

Navi-Trainer Professional 6

Instructor Manual

Version 6.0

Date of issue: February 2024

Copyright © 2023-2024 Wärtsilä Voyage Oy. All rights reserved.

The information contained herein is proprietary to Wärtsilä Voyage Oy and shall not be duplicated in whole or in part. The technical details contained in this manual are accurate at the date of issue but are subject to change without prior notice. Wärtsilä Voyage Oy pursues a policy of continuous development. This may mean that the product delivered has additional enhancements not yet covered by the latest version of this manual.

The names of actual companies and products mentioned herein may be the trademarks of their respective owners.

Wärtsilä uses data gathered from equipment and software to improve and develop its products and services.

 $\mbox{NVIDIA} \ \mbox{\@}$ and $\mbox{PhysX} \ \mbox{\@}$ are registered trademarks of \mbox{NVIDIA} Corporation and are used under license.

Disclaimer: Supplier's Products and Services related to the operation and monitoring of vessels (including but not limited to solutions relating to onboard navigation and remote ship traffic control) are provided only as an aid to decision making. The safe voyage planning, navigation, manoeuvring, operation and monitoring of and instructions and advice given to vessels are dependent on human skill and judgement and are the responsibility of the relevant appropriately qualified personnel responsible for such tasks in accordance with all applicable laws, regulations and best practices. Accordingly, Supplier will not be liable for any accident, damage or delay caused by or to any vessel owned or operated by Customer or any other person whether or not the accident or damage is related to the operation or failure of the Products or Services.

Table of Contents

		ction	
?riı	nting	House Conventions	12
1.	Arra	angement of Instructor Workplace	13
	1.1	Main Window	14
		1.1.1 Menu Bar	15
		1.1.2 Toolbars	15
		1.1.3 Customising Menu Bar and Toolbars	16
		1.1.4 Chart Window	21
		1.1.5 Status Bar	22
	1.2	Workplace Configuration	23
		1.2.1 Saving Configuration	23
		1.2.2 Loading Configuration	23
		1.2.3 Default Configuration	24
		1.2.4 Autosaving Configuration	24
2.	Pre	paring Exercise	25
	2.1	Creating New Exercise	28
		2.1.1 Selecting Training Area	28
		2.1.2 Information on Training Area	29
		2.1.3 Changing Training Area	
	2.2	Chart Setup	
		2.2.1 Chart Centering	
		2.2.2 Zooming Chart Fragment	
		2.2.3 Changing Chart Scale	
		2.2.4 Canceling Operations on Charts	
		2.2.5 Unloading and Loading Area Charts	
		2.2.6 Chart Information	
		2.2.7 Measuring Bearing and Range	
		2.2.8 Setting Reference Point	
		2.2.9 Changing Window Orientation	
		2.2.10 Display of Overlays	
		2.2.11 Depth Section	
		2.2.12 Changing Colour Palette	
		2.2.13 Readonly Mode	
		2.2.14 Selecting Motion Display Modes	
	2.3	Adding New Object to Exercise	
		2.3.1 Adding Own Ship	
		2.3.2 Adding Target Ship	
	2.4	Manipulations with Objects	
		2.4.1 Selecting Object on Chart	
		2.4.2 Opening Object Context Menu	
		2.4.3 Moving Object	
		2.4.4 Rotating Object	
		2.4.5 Renaming Object	
		2.4.6 Setting Ship Tactical Image	
		2.4.7 Opening Object Properties Panel	
		2.4.8 Deleting Object	
		2.4.9 Objects Panel	
	2.5	Selection of Measurement Units	

2.6	General Exercise Settings	53
	2.6.1 Displaying Ship Track as Series of Dots	53
	2.6.2 Displaying Ship Track as Series of Contours	54
	2.6.3 Speed Vectors Settings	55
	2.6.4 Automatic Sound Signals Settings	57
	2.6.5 Model Effects Settings	57
	2.6.6 Miscellaneous Settings	58
2.7	Setting Environmental Sailing Conditions	60
	2.7.1 Environment Settings	61
	2.7.2 Overall Environmental Conditions	71
	2.7.3 Environment Conditions Template	82
	2.7.4 Local Environmental Conditions	84
	2.7.5 Importing Tides and Currents	91
	2.7.6 Creating Vector Field of Current	94
	2.7.7 Setting Longshore Current	96
	2.7.8 Setting Clouds	98
	2.7.9 Setting Atmosphere Fronts	
	2.7.10 Editing Wave Spectrum	
2.8	Setting Ship Properties	
	2.8.1 General Ship Properties	
	2.8.2 Ship Position	
	2.8.3 Autopilot Settings	
	2.8.4 Radar Settings	
	2.8.5 UAIS Initial Settings	
	2.8.6 Navigational Signals Settings	
	2.8.7 Specifying Options	
	2.8.8 Contours Settings	
	2.8.9 Fenders Settings	
	2.8.10 Cargo Settings	
2.9	Ship Data	
	0 Route Planning	
	2.10.1 Plotting Route on Chart	
	2.10.2 Assigning Object to Route	
	2.10.3 General Route Settings	
	2.10.4 Editing Route on Chart	
	2.10.5 Setting Route Properties	
	2.10.6 Route Tabular Form	
	2.10.7 Conditions for Following on Route	
	2.10.8 Route Export and Import	
	2.10.9 Importing KML or KMZ File	
2.1	1 Faults Panel	
	2.11.1 Setting Faults	
	2.11.2 Example of Fault Scenario	
2.1	2 Setting Custom Alarms	
••	2.12.1 Creating Custom Alarms	
	2.12.2 Template of Custom Alarms	
	2.12.3 Activating Custom Alarms	
2.1:	3 Setting Synchronization with External GPS Source	
	4 Magnetic Deviation	
•	2.14.1 Setting Magnetic Deviation	
	2.14.2 Magnetic Deviation Template	
2.1	5 Aid-to-Navigation Objects (Buovs)	146

2.16 Land Objects	147
2.16.1 Setting Land Objects	147
2.16.2 Tidal Stream Signal Station	147
2.16.3 Traffic Control Signal Station	148
2.17 Floating Objects	149
2.17.1 Setting Floating Objects	149
2.17.2 Designation of Floating Objects on Chart	154
2.18 Creature Objects	156
2.18.1 Whales	156
2.19 Radar Echo Objects	159
2.19.1 Setting Solid Radar Object	159
2.19.2 Setting Bridge Radar Object	161
2.19.3 Setting Voltage Line Radar Object	164
2.20 E-Tutor Objects	167
2.20.1 Using Safety Lines	167
2.21 AIS AtoN	171
2.22 Smart Targets	173
2.22.1 Variants of Maneuvering	174
2.22.2 Setting Smart Targets Globally	176
2.22.3 Setting Smart Targets Individually	177
2.22.4 Monitoring Maneuver Events	178
2.22.5 Maneuvers of Ships Assigned on Separate Routes	179
2.22.6 Maneuvers of Ships Assigned on Same Route	180
2.22.7 Maneuvers of Ships Moving on Course	182
2.22.8 Cooperative Maneuvers of Ships in High Traffic Areas	183
2.22.9 Avoiding Collisions with Buoys	184
2.22.10 Avoiding No-Go Zones	186
2.22.11 Avoiding Depth Zones	189
2.23 Using Objects Templates	192
2.23.1 Template of Single Object	192
2.23.2 Template of Group of Objects	193
2.24 Setup of Exchange Recording	195
2.25 Setup of Video Log File Recording	196
2.26 Creating Comments on Exercise	197
2.27 Adding User Information to Chart	198
2.28 Pre-Playing Exercise	201
2.29 Saving and Loading Exercise	202
2.29.1 Saving Exercise	202
2.29.2 Loading Exercise	203
Simulator Session	205
3.1 Simulator Session Management	207
3.1.1 Assignment of Exercises to Bridges	207
3.1.2 Starting Exercise	210
3.1.3 Pausing Exercise	210
3.1.4 Rewinding Exercise	
3.1.5 Stopping Exercise	
3.1.6 Switching Between Bridges	
3.1.7 Saving Current State of Exercise	
3.1.8 Session Overloading	
3.1.9 Pending Tasks	212
3.2 Control of Ships	
3.2.1 Control of Own Ship and Target Ship (6-DoF Model)	214

3.

	3.2.2 Control of Target Ship (Simplified Model)	214
	3.3 Control of Autopilot	217
	3.4 Giving Sound Signals	219
	3.4.1 Giving Sound Signals by Target Ships	219
	3.4.2 Giving Siren Signals	
	3.5 Motion Parameters	
	3.6 Control of Navigation Signals	
	3.6.1 Selecting Signals Preset Combination	
	3.6.2 Setting Lights	
	3.6.3 Setting Shapes	
	3.6.4 Change of Lights and Shapes	
	3.7 Flashing Light Signalling	
	3.8 UAIS Message Exchange	
	3.8.1 Sending AIS Messages	
	3.8.2 Monitoring of UAIS Messages	
	3.9 Using Virtual Force	
	3.9.1 Setting Virtual Force	
	3.9.2 Changing Virtual Force	
	3.9.3 Setting Virtual Force Properties	
	3.10 Ship Information Panel	
	3.11 Events Panel	
	3.12 Monitoring of Dangerous Targets	
	3.12.1 CPA/TCPA Monitoring Panel	
	3.12.2 General CPA/TCPA Settings	
	3.12.3 Graphic Presentation of CPA/TCPA Data	
	3.13 Ship Diagram	
	3.13.1 Control of Ship Diagram	
	3.13.2 Diagram Parameter Properties	
	3.13.3 Diagram Settings Template	
	3.13.4 Export of Diagram Data to CSV-Format	
	3.13.5 Diagram Printout	
	3.14 Instructor Visualization	
	3.14.1 Adding Camera to Exercise	
	3.14.2 Control of Instructor Visualization	
	3.15 Visualization Screen Capture	
	3.16 Loading Visual Profile on Bridge	
	3.17 Visual Settings	
	3.18 Setting Radar Position	
	3.19 Loading ECDIS Data	259
	3.20 Transmitting NAVTEX Messages	260
	3.21 Control of Auxiliary Machinery	263
	3.22 Control of Remote Sensors	265
	3.23 Creating Log Scripts	267
4.	Debriefing	269
	4.1 General	270
	4.2 Loading Log File	271
	4.3 Log Playback	272
	4.4 Log Script Playback	
	4.5 Exercise Window Printout	
	4.6 Playback of Audio Log Files	
	4.7 Playback of Video Log Files	
	•	280

		4.8.1 Report Generator Wizard	
		4.8.2 Report Setup Wizard	. 283
		4.8.3 Report Settings	. 285
		4.8.4 Reports Panel	.285
5.	Spe	ecial Operations	287
	5.1	Mooring Operations	.289
		5.1.1 Mooring Walls	.289
		5.1.2 Fender Protection	.289
		5.1.3 Operations with Tow/Mooring Lines	.293
		5.1.4 Control of Mooring Winches	
		5.1.5 Setting Mooring Objects	
		5.1.6 Operations with Inflatable Fenders	
	5.2	Operations with Anchors	
		5.2.1 Setting Anchor Properties	
		5.2.2 On-Chart Control of Anchors	
		5.2.3 Anchor Control During Exercise	
	5.3	Towing Operations	
		5.3.1 Setting Bridle	
		5.3.2 Creating Barge Tow	
		5.3.3 Operations with Tow Winches	
	5.4	Towing in Inland Waterways	
	U	5.4.1 Articulated Tug Barge	
		5.4.2 Pusher-Barge Train	
		5.4.3 Breaking Connection	
		5.4.4 Attaching Remote Sensor to Barge	
		5.4.5 Barge Navigational Signals	
		5.4.6 Applying Assist Force	
	55	Operations with Automatic Tugs	
	0.0	5.5.1 Automatic Tug General Settings	
		5.5.2 Tug Reserving and Releasing	
		5.5.3 Basic Operations	
		5.5.4 Automatic Tug Control Page	
	56	Pilot Boat Operations	
	5.0	5.6.1 Pilot Boat Maneuvers	
	5 7	5.6.2 Pilot Transfer Operations in Ice Conditions	
	5.7	·	
		5.7.1 Using Ice Object	
		5.7.2 Using Ice Generator Object	
		5.7.3 Control of Ice Resistance	
		5.7.4 Setting Ice Accretion	
		5.7.5 Ice Channel Evaporation Visual Effect	
		5.7.6 Operations with Icebergs	
	г о	5.7.7 Monitoring Trainee Performance	
	5.8	Fishing Operations	
		5.8.1 Shoal Objects	
		5.8.2 Pinnacle Objects	
		5.8.3 Fishing Buoys	
		5.8.4 Setting Trawl	
		5.8.5 Setting Purse Seine	
		5.8.6 Setting Long Line	
		5.8.7 Watching Trainee Performance	
	5.9	Search and Rescue Operations	∴379

	5.9.1 SAR Objects	379
	5.9.2 Flag Signalling	390
	5.9.3 Giving Distress Signals	
	5.9.4 Use of Rescue Boat	394
	5.9.5 Life Raft Operations	
	5.9.6 Using Lifesling for MOB Recovery	
	5.10 Operations with Helicopters and Aircrafts	
	5.10.1 Helicopter Initial Parameters	
	5.10.2 Helicopter Control	
	5.10.3 Operations with Aircraft	
	5.11 Underway Replenishment Operations	
	5.11.1 Setting Underway Replenishment Criteria	
	5.11.2 Underway Replenishment Control Panel	
	5.11.3 P&D Line and Span Wire Rendering/Heaving	
	5.12 Joint Ship Maneuvering in Formations	
	5.12.1 Creating Fleet Formation	
	5.12.2 Fleet Formation Management	
	~	
	5.13 Passing of Locks	
	5.13.1 Setting Lock Initial Properties	
	5.13.2 Control of Lock	
	5.13.3 Monitoring Fulfilment of Exercise with Locks	
	5.14 Control of Semaphores	
	5.15 Ro-Ro Operations	
	5.16 ECDIS Training	
	5.16.1 Ordering Charts, Chart Updates and Weather Forecasts	
	5.16.2 Order Processing	
	5.17 FOS Data Import	
	5.18 Operations with Submarine	
	5.19 Pirate Attacks	
	5.20 Operations with Moveable Bridges	
	5.21 GNSS Control	
	5.22 Control of Motion Platforms	
	5.22.1 Control of Motion Platforms via UHI Task	
	5.22.2 Control of Motion Platforms via MPI Task	
6.	Ship Communication Aids	455
	6.1 Intercom	
	6.1.1 Telephone Communication with Subscribers	456
	6.1.2 Playback of Telephone Communications	
	6.2 VHF Communication	460
	6.2.1 Radio Traffic Monitoring	460
	6.2.2 Keeping of Listening Watch	461
	6.2.3 Playback of Radiotelephone Communications	463
	6.2.4 VHF Background Noise	463
	6.2.5 VHF Notifications	465
	6.2.6 VHF Instructor Panel	466
7.	Exercise Remote Control	469
	7.1 General	
	7.2 Running Remote Instructor Panel	
	7.3 Exercise Control	
	7.4 Displaying Ship Properties	
	7.5 Displaying Speed Vectors	474

7.7	' Adding Event Markers	476
Anney	Menu Commands Description	477

Introduction

This Manual is intended for use by the Instructor of the Navi-Trainer Professional 6 navigational simulator(NTPRO 6).

The Instructor workplace is based on the Navi-Trainer Instructor application intended for:

- Creation and editing of exercise scenarios;
- Starting and conducting simulator sessions;
- · Debriefing.

An "exercise scenario" or an "exercise" is understood as a description of location and motion of objects within the scene.

An "exercise scene" or a "scene" is understood as a model or actual geographic area.

The scene is an aggregate of:

- The terrain spatial model (submerged and surface part);
- Spatial models of coastal structures (buildings, bridges, berths, etc.);
- Spatial models of aids to navigation (lighthouses, buoys, etc.).

Scenes of different geographic areas on the earth are supplied together with the simulator.

An exercise can contain the following objects: ships, tugs, barges, aircraft, helicopters, environment objects, and others.

The exercise scenario creation procedure implies:

- · Selection of a geographic area;
- Setting environment sailing conditions;
- · Adding objects to exercise;
- Setting initial object properties (course, speed, position, and others).
- · Plotting routes of objects;
- · Setting navigational equipment faults .

The simulator session procedure includes the following main stages:

- · Exercise assignment to bridge and starting;
- Exercise monitoring and control.

Exercise debriefing implies:

- · Log playback;
- · Report generation.

The Manual consists of seven chapters and an annex. Each of the chapters has its own list of contents.

For work with the Manual you may need the following related documents:

- Navi-Trainer Professional 6. Navigational Bridge. Version 6.0;
- Navi-Trainer Professional 6. Technical Description and Installation Manual. Version 6.0;
- E-Tutor 5000. User Manual.

Printing House Conventions

SAMPLE OF NOTATION	USAGE COMMENTS Messages, commands, files, folders and other Windows OS info		
Setup.exe			
<enter></enter>	Keyboard key names		
Menu	Menu items		
Interface	Windows names, tabs, icons, checkboxes, radio buttons and text box captions, and other interface elements		
NOTE	Notes		
WARNING	Warnings		
Document	Document names		
NTPRO_6\Exercise	Path to a file, folder		
see on page 12	Cross reference		

1. Arrangement of Instructor Workplace

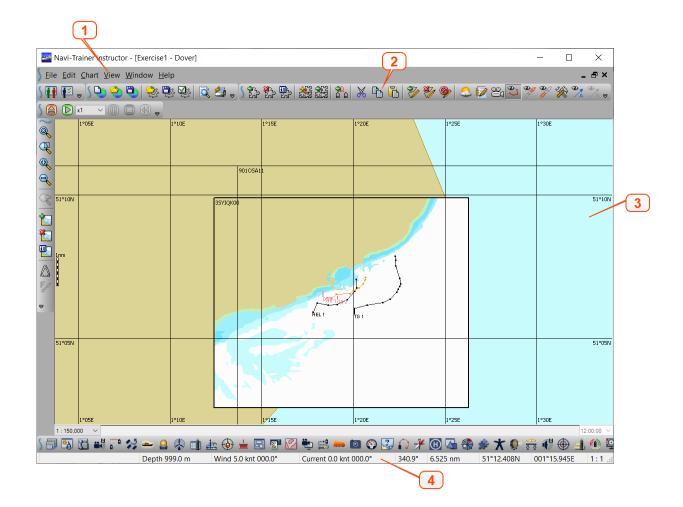
1.1	Main \	Window	14
	1.1.1	Menu Bar	15
	1.1.2	Toolbars	15
	1.1.3	Customising Menu Bar and Toolbars	16
	1.1.4	Chart Window	21
	1.1.5	Status Bar	22
1.2	Work	place Configuration	23
	1.2.1	Saving Configuration	23
	1.2.2	Loading Configuration	23
	1.2.3	Default Configuration	24
	1.2.4	Autosaving Configuration	24

1.1 Main Window

After the application start, the Navi-Trainer instructor main window appears.

The Main window contains (see picture below):

- 1 Menu bar (see "Menu Bar" on page 15)
- 2 Toolbars (see "Toolbars" on page 15)
- 3 Sailing area chart window (see "Chart Window" on page 21)
- 4 Status bar (see "Status Bar" on page 22)



NOTE: The Instructor can customize the menu bar and toolbars (see "Customising Menu Bar and Toolbars" on page 16).

1.1.1 Menu Bar

Menu bar contains the following menu commands:

- File handling exercise files;
- Edit editing exercises;
- · Chart handling charts;
- View setting the instructor station elements for the display;
- Window handling windows;
- Help calling context help and obtaining additional information.

NOTE: For more information, see "Annex. Menu Commands Description" on page 477.

To change the arrangement of the menu bar, click the sign in the left-hand part of the menu bar, drag and drop the menu bar in the desired position on the tool ribbon. When the menu bar is dragged beyond the tool ribbon, the menu bar assumes the form of a window.

1.1.2 Toolbars

The main window has the following toolbars:

· Chart - select main chart operation commands



Control - select exercise control functions



· Bridge select - select the active bridge



Editor - use the exercise editor



• e-Tutor - use the e-Tutor Scenario Editor and monitor the trainee performance



External connections - set parameters and status of external connections



• Loading ECDIS data - transfer data from the Trainee ECDIS to the Instructor ECDIS



Main - select main exercise handling tools



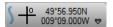
• Panels - open control panels



· Plotting - display ship speed vectors, routes, tracks



Reference point - set a reference point on the chart and show its coordinates



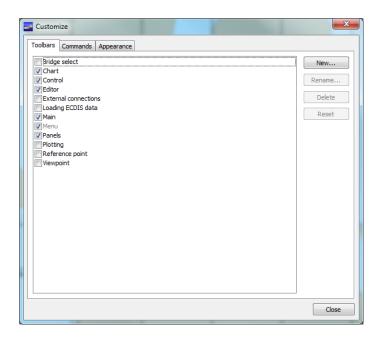
Viewpoint - control the visualization camera and the radar



To change the arrangement of the toolbar, click the sign in the left-hand part of the toolbar, drag and drop the toolbar in the desired position on the tool ribbon. When the toolbar is dragged beyond the tool ribbon, the toolbar assumes the form of a window.

1.1.3 Customising Menu Bar and Toolbars

The Instructor can customize the menu bar and toolbars on the Customize panel (the View\Customize... menu item).



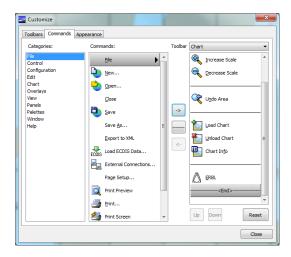
The Customize panel contains the following pages:

- Toolbars handle default toolbars (showing, hiding, restoring) and custom toolbars (creating, renaming, showing, hiding, deleting);
- Commands change contents of the menu bar and toolbars;
- Appearance change toolbar colour palette and design.

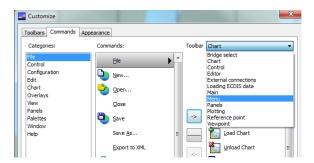
1.1.3.1 Changing Toolbar Commands

To change commands of the toolbars and menu bar, use the following procedure:

- 1. Choose the View\Customize... menu item or click the button in the right-hand part of any toolbar on the ribbon.
- 2. Select the Commands tab of the Customize panel.



 From the Toolbar drop-down list, select the name of the toolbar (or Menu for the menu bar) which should be changed. The list of tools and commands of the selected toolbar/menu will be displayed under the Toolbar dropdown list in the right part of the Customize panel.



- 4. To add a tool or command to the selected toolbar/menu bar:
 - a. In the Categories list, select the required functional group of tools.
 - b. From the Commands list, select the required tool or command.
 - c. Click the button. The tool or command will be added to the list of tools and commands of the selected toolbar in the right part of the **Customize** panel.
- 5. To add a separating line to the selected toolbar/menu bar:
 - a. From the list of tools in the right part of the panel, select the tool before which the separating line should be added.
 - b. Click the button.
- 6. To delete a tool/command/separate line from the selected toolbar/menu bar:
 - a. Select the tool/command/separate line in the list of tools in the right part of the panel.
 - b. Click the button.

- 7. To change the position of a tool/command/ separate line on the toolbar/menu bar:
 - a. Select the tool/command/separate line from the list of tools in the right part of the panel.
 - b. To move it up, click the **Up** button the required number of times.
 - c. To move it down, click the **Down** the required number of times.

To restore the toolbar/menu bar, choose one of the following ways:

 Select the Commands tab of the Customize panel. From the Toolbar drop-down list select the name of the toolbar (or Menu for a menu bar), which the default settings should be restored for and click the Reset button.

Or

 Select the Toolbars tab of the Customize panel. Click the name of the toolbar which the default settings should be restored for (except for menu and custom toolbars) and click the Reset button.

To close the **Customize** window, click the **Close** button.

1.1.3.2 Showing and Hiding Toolbars

To hide/show the toolbar in the main window, use the following procedure:

- 1. Choose the View\Customize... menu item or click the button in the right-hand part of any toolbar.
- 2. Select the Toolbars page.
- Check/clear the name of the toolbar which should be shown/hidden. The toolbar will appear/disappear in/from the main window of the Navi-Trainer Instructor application.

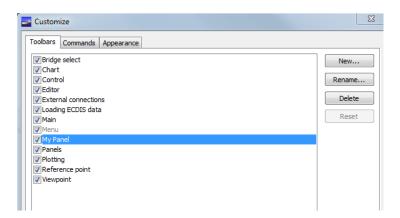
To close the Customize window, click the Close button.

NOTE: All the toolbars can be hidden, except for the menu bar.

1.1.3.3 Creating Custom Toolbars

To create custom toolbar, use the following procedure:

- 1. Choose the View\Customize... menu item or click the button in the right-hand part of any toolbar.
- 2. Open the Toolbars tab of the Customize panel.
- 3. Click the New... button.
- 4. In the **Toolbar name** dialog box, enter the toolbar name and click the **OK** button. The **Toolbar name** dialog box closes. The name of the new custom toolbar will appear on the list of toolbars on the **Toolbars** tab.





The custom toolbar appears in the top left corner of the screen in the form of an empty window

5. To place the custom toolbar, click the left mouse button on the toolbar heading and drag and drop the toolbar in the desired position on the tool ribbon.

To rename the custom toolbar, use the following procedure:

- 1. On the Toolbars tab of the Customize window, select the required custom toolbar from the list.
- 2. Click the Rename button. The Toolbar name dialog box appears.



3. Enter a new toolbar name and click the OK button.

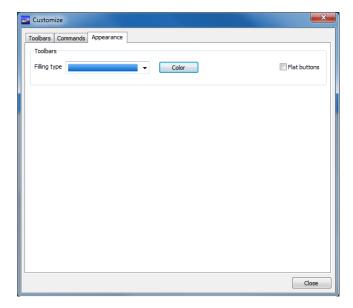
To delete a custom toolbar, use the following procedure:

- 1. On the Toolbars tab of the Customize window, select the custom toolbar from the list.
- 2. Click the Delete button.
- 3. Click the OK button.

1.1.3.4 Changing Toolbar Appearance

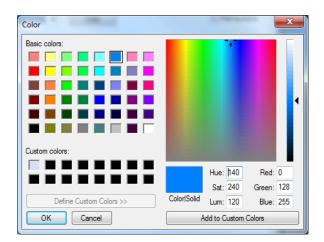
To change the toolbar appearance, use the following procedure:

- 1. Choose the View\Customize... menu item or click the button in the right-hand part of any toolbar.
- 2. Select the Appearance tab.

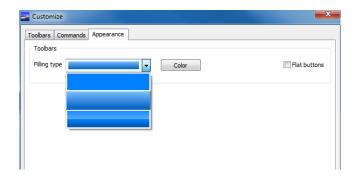


3. Click the Color button.

4. In the Basic colors group box, select the necessary design colour and click the OK button.



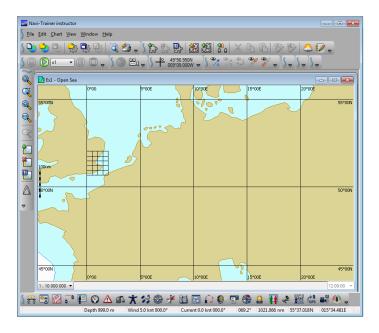
- 5. If the **Basic colors** group box does not contain the necessary colour, to select the required hue, use the following procedure:
 - a. Click the Define Custom Colors>> button.
 - b. By moving the sliders along the colour field and the brightness scale in the right-hand part of the Color window, select the necessary colour and its saturation, or set the colour coordinates: Hue, Sat, Lum, Red, Green, Blue.
 - c. Click the Add to Custom Colors button. The selected colour will appear in the Custom colors group box.
 - d. Click OK.
- 6. To change the interface design, use the following procedure:
 - a. Choose the View\Customize... menu item or click the button in the right-hand part of any toolbar on the ribbon.
 - b. Select the Appearance tab.
 - c. Select the interface design option from the Filling type drop-down list.
 - d. If necessary, select the **Flat buttons** checkbox to set flat type buttons (the convex type buttons are selected by default).



7. To close the **Customize** panel, click the **Close** button.

1.1.4 Chart Window

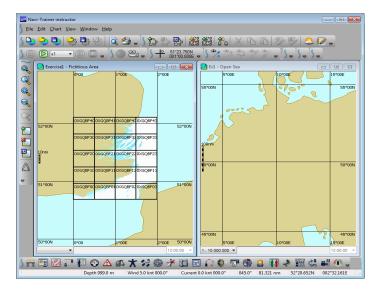
At the initial stage of work, it is necessary to create a window with a chart of the training area (see "Selecting Training Area" on page 28) or to open a saved exercise (File\Open... menu item).



The chart window can be expanded to the width of the main window , returned to the initial state , collapsed or closed .

To add a new chart window for an exercise: choose the Window\NewWindow menu item.

To arrange several chart windows on the screen, use the following Window menu items: Cascade, Tile Horizontally, Tile Vertically.



To switch from one window to another, use the list of open windows in the Window menu.



NOTE: Parameters and scale can be set for each window individually.

In the left corner under the chart, there is a drop-down list for selecting the chart scale (see "Changing Chart Scale" on page 32).

After the exercise has been assigned to the simulator bridges, the left corner under the chart will also display tabs which have names of simulator bridges. Each tab contains a chart of the training area centered in the own ship assigned to the relevant bridge.

If the log file is running, a control tool for log playback appears at the bottom of the chart (see "Log Playback" on page 272).



1.1.5 Status Bar

After the chart window has been opened, a status bar appears in the bottom part of the main window:



The status bar shows the following information:

- · depth in the chart point under the mouse cursor;
- wind velocity in the chart point under the mouse cursor;
- · current speed in the chart point under the mouse cursor;
- bearing and distance from the reference point to the point under the mouse cursor;
- absolute position of the chart point under the mouse cursor;
- current chart display scale;
- window orientation (North up by default. 000.0 Nord Up);
- absolute or relative time of the exercise beginning (absolute time of 12:00:00 by default).

NOTE: Measurement units are set on the Unit settings panel (see "Selection of Measurement Units" on page 52).

1.2 Workplace Configuration

The workplace configuration is an arrangement of windows, panels and toolbars of the Navi-Trainer instructor application as well as names of the loaded exercises and initial parameters settings.

The Navi-Trainer instructor application allows storing any number of the user workplace configurations in files for their future use.

1.2.1 Saving Configuration

To save the configuration, use the following procedure:

1. After the workplace configuration is completed, choose the File\Configuration\ Save As... menu item.



2. Enter the configuration name in Name field and click OK.

To save changes in the current configuration, use the following procedure:

- 1. Choose the File\Configuration\Save menu item.
- 2. To rewrite the configuration, click Yes. All the changes you have made will be saved in the current configuration.
- 3. To cancel saving changes in the configuration, click No.

After the configuration has been saved, it can be loaded for further work.

1.2.2 Loading Configuration

To load the Instructor workplace configuration, use the following procedure:

 Choose the File\Configuration\Open... menu item. The Open configuration panel with the list of available configurations appears.



Select the required configuration from the list and click the OK button. The Instructor workplace will be arranged in accordance with the selected configuration.

1.2.3 Default Configuration

To set the default configuration, choose the File\Configuration\Set as Default menu item. The selected configuration will be loaded automatically every time the Instructor workplace is run.

1.2.4 Autosaving Configuration

To save the changes in the current configuration automatically, choose the File\Configuration\Auto Save configuration menu item.

To cancel the autosaving of the configuration, clear the checkbox next to the File\Configuration\Auto Save configuration menu item.

2. Preparing Exercise

2.1	Creat	ing New Exercise	28
	2.1.1	Selecting Training Area	28
	2.1.2	Information on Training Area	29
	2.1.3	Changing Training Area	30
2.2	Chart	Setup	31
	2.2.1	Chart Centering	31
	2.2.2	Zooming Chart Fragment	31
	2.2.3	Changing Chart Scale	32
	2.2.4	Canceling Operations on Charts	32
	2.2.5	Unloading and Loading Area Charts	33
	2.2.6	Chart Information	33
	2.2.7	Measuring Bearing and Range	34
	2.2.8	Setting Reference Point	34
	2.2.9	Changing Window Orientation	34
	2.2.10	Display of Overlays	34
	2.2.11	Depth Section	37
	2.2.12	Changing Colour Palette	37
	2.2.13	Readonly Mode	37
	2.2.14	Selecting Motion Display Modes	38
2.3	Addin	g New Object to Exercise	39
	2.3.1	Adding Own Ship	42
	2.3.2	Adding Target Ship	44
2.4	Manip	oulations with Objects	46
	2.4.1	Selecting Object on Chart	46
	2.4.2	Opening Object Context Menu	46
	2.4.3	Moving Object	46
	2.4.4	Rotating Object	46
	2.4.5	Renaming Object	47
	2.4.6	Setting Ship Tactical Image	47
	2.4.7	Opening Object Properties Panel	49
	2.4.8	Deleting Object	50
	2.4.9	Objects Panel	50
2.5	Selec	tion of Measurement Units	52
2.6	Gene	ral Exercise Settings	53
	2.6.1	Displaying Ship Track as Series of Dots	53
	2.6.2	Displaying Ship Track as Series of Contours	54
	2.6.3	Speed Vectors Settings	55
	2.6.4	Automatic Sound Signals Settings	57
	2.6.5	Model Effects Settings	57
	2.6.6	Miscellaneous Settings	58
2.7	Settin	g Environmental Sailing Conditions	60
	2.7.1	Environment Settings	61

	2.7.2 Overall Environmental Conditions	71
	2.7.3 Environment Conditions Template	82
	2.7.4 Local Environmental Conditions	84
	2.7.5 Importing Tides and Currents	91
	2.7.6 Creating Vector Field of Current	94
	2.7.7 Setting Longshore Current	96
	2.7.8 Setting Clouds	98
	2.7.9 Setting Atmosphere Fronts	100
	2.7.10 Editing Wave Spectrum	102
2.8	Setting Ship Properties	105
	2.8.1 General Ship Properties	107
	2.8.2 Ship Position	107
	2.8.3 Autopilot Settings	108
	2.8.4 Radar Settings	109
	2.8.5 UAIS Initial Settings	111
	2.8.6 Navigational Signals Settings	112
	2.8.7 Specifying Options	114
	2.8.8 Contours Settings	116
	2.8.9 Fenders Settings	117
	2.8.10 Cargo Settings	118
2.9	Ship Data	119
2.10	Route Planning	120
	2.10.1 Plotting Route on Chart	120
	2.10.2 Assigning Object to Route	121
	2.10.3 General Route Settings	122
	2.10.4 Editing Route on Chart	123
	2.10.5 Setting Route Properties	123
	2.10.6 Route Tabular Form	127
	2.10.7 Conditions for Following on Route	128
	2.10.8 Route Export and Import	128
	2.10.9 Importing KML or KMZ File	130
2.11	Faults Panel	132
	2.11.1 Setting Faults	132
	2.11.2 Example of Fault Scenario	134
2.12	Setting Custom Alarms	139
	2.12.1 Creating Custom Alarms	139
	2.12.2 Template of Custom Alarms	141
	2.12.3 Activating Custom Alarms	142
2.13	Setting Synchronization with External GPS Source	143
2.14	Magnetic Deviation	144
	2.14.1 Setting Magnetic Deviation	144
	2.14.2 Magnetic Deviation Template	144
2.15	Aid-to-Navigation Objects (Buoys)	146
2.16	Land Objects	147

	2.16.1 Setting Land Objects	147
	2.16.2 Tidal Stream Signal Station	147
	2.16.3 Traffic Control Signal Station	148
2.17	Floating Objects	149
	2.17.1 Setting Floating Objects	149
	2.17.2 Designation of Floating Objects on Chart	154
2.18	Creature Objects	156
	2.18.1 Whales	156
2.19	Radar Echo Objects	159
	2.19.1 Setting Solid Radar Object	159
	2.19.2 Setting Bridge Radar Object	161
	2.19.3 Setting Voltage Line Radar Object	164
2.20	E-Tutor Objects	167
	2.20.1 Using Safety Lines	167
2.21	AIS AtoN	171
2.22	Smart Targets	173
	2.22.1 Variants of Maneuvering	174
	2.22.2 Setting Smart Targets Globally	176
	2.22.3 Setting Smart Targets Individually	177
	2.22.4 Monitoring Maneuver Events	178
	2.22.5 Maneuvers of Ships Assigned on Separate Routes	179
	2.22.6 Maneuvers of Ships Assigned on Same Route	180
	2.22.7 Maneuvers of Ships Moving on Course	182
	2.22.8 Cooperative Maneuvers of Ships in High Traffic Areas	183
	2.22.9 Avoiding Collisions with Buoys	184
	2.22.10 Avoiding No-Go Zones	186
	2.22.11 Avoiding Depth Zones	189
2.23	Using Objects Templates	192
	2.23.1 Template of Single Object	192
	2.23.2 Template of Group of Objects	
2.24	Setup of Exchange Recording	195
2.25	Setup of Video Log File Recording	196
2.26	Creating Comments on Exercise	197
2.27	Adding User Information to Chart	198
2.28	Pre-Playing Exercise	201
2.29	Saving and Loading Exercise	202
	2.29.1 Saving Exercise	202
	2.29.2 Loading Exercise	203

2.1 Creating New Exercise

Creation of a new exercise begins with a scene selection. An exercise scene is a model of actual geographic area.

2.1.1 Selecting Training Area

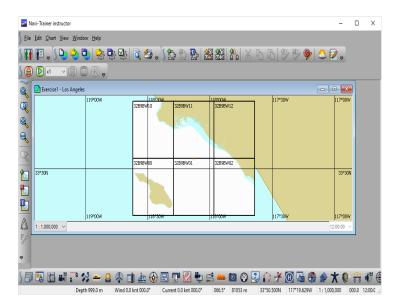
To select a training area, use the following procedure:

1. Choose the File\New... menu item, or click the toolbar button, or press <Ctrl>+<N> key combination. The Select area window that contains Area list of all the available geographic areas appears.



NOTE: To show areas that were not purchased under the license, select the **Show not licensed areas** checkbox. Area names will appear in grey in the **Area** list. These areas will not be selectable unless a license is purchased.

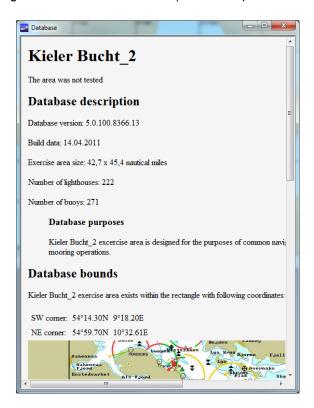
- 2. Click the area name in the list. The green marker shows the selected area position on the chart.
- Click the OK button. The chart window appears. It contains charts covering the scene area.



NOTE: The chart display scale is selected so that the entire scene area should fit in the program window. To change the chart scale, see "Changing Chart Scale" on page 32.

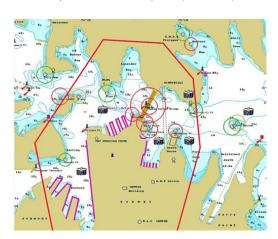
2.1.2 Information on Training Area

To obtain information on the training area, choose the **Help\Area**... menu item. The **Database window** will display main data, charts of the area (with zones of recommended sailing tracks, piers for mooring operations and zones with a high/low/medium level of detail) and some pictures of the area.



For some areas, you can obtain a larger scale of zones with a high level of detail:

1. Click the zone red boundary. The screen will display a chart of the zone with a high level of detail, showing the borders (in the red colour) and piers for performing mooring operations (magenta).



2. To view some fragment of the scene, click the button on the chart.

2.1.3 Changing Training Area

The Instructor can change the current training area to another area. In this case, the coordinates of the exercise objects are saved.

To change area, use the following procedure:

1. Before the exercise start, choose the **Edit\Change area...** menu item. The list of all the available geographic areas appears in the **Select area** window.



- 2. Click the area name. The green marker shows the position of selected area on the chart.
- 3. Click the OK button.

2.2 Chart Setup

2.2.1 Chart Centering

To center a chart with respect to the selected point, use the following procedure:

- 1. Choose the Chart\Centre menu item, or click the toolbar button, or press the <Ctrl>+<Alt>+<C> key combination. The cursor will switch to the chart centering mode.
- 2. Click the required place on the chart, or press one of the following key combinations:
 - move to the left <Shift>+<←>;
 - move to the right <Shift>+<→>;
 - o move up <Shift>+<↑>;
 - o move down <Shift>+<↓>.

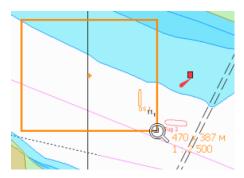
NOTE: The selected point will be displayed in the centre of the window. The screen centre shifts to a distance equal to 1/1000 of the current display scale.

3. To return to the default cursor mode, click the right mouse button.

2.2.2 Zooming Chart Fragment

To zoom a chart fragment, use the following procedure:

- 1. Choose the Chart\Zoom Area menu item, or click the toolbar button, or press the <Ctrl>+<Z> key combination.
- 2. Click on the chart and move the cursor to select a chart fragment.

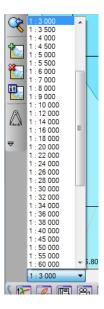


- 3. Click to zoom the selected chart fragment to the entire window size.
- 4. Right-click to return to the default cursor mode.

2.2.3 Changing Chart Scale

The chart scale is shown in the status bar at the bottom of the screen, and on the graduated scale to the left on the chart.

To select the scale, use the drop-down list in the left bottom corner under the chart.



The chart scale can be changed from 1:10 000 000 till 1:50.

There are several ways to change the scale (see Table 1).

Table 1. Ways to increase or decrease the scale of the chart

INCREASE SCALE	DECREASE SCALE
Choose the Chart\Increase Scale menu item	Choose the Chart\Decrease Scale menu item
Click the toolbar button	Click the toolbar button
Press the <ctrl>+ <↑ > key combination</ctrl>	Press the <ctrl>+ <↓ > key combination</ctrl>
Press the <+> key on the digital keyboard	Press the <-> key on the digital keyboard
Rotate the mouse scroll wheel backward (toward). In this case, the chart will be centered in the cursor position	Rotate the mouse scroll wheel forward (away from you). In this case, the chart will be centered in the cursor position

2.2.4 Canceling Operations on Charts

To cancel operations performed on the charts (scaling, shifting, etc.), choose the **Chart\Undo Area** menu item, or click the toolbar button, or press the <Ctrl>+<Backspace> key combination.

2.2.5 Unloading and Loading Area Charts

Usually, all area charts are displayed on the screen. Any chart can be removed from display as required. To unload a chart, use the following procedure:

- Choose the Chart\Unload Chart menu item, or click button on the toolbar, or press <Ctrl>+<Shift>+<L> key combination. The mouse cursor will switch to the chart unloading mode .
- 2. Click on the boundary of the chart required to be unloaded. The chart will disappear, and its boundary will be shown as a thin line.
- 3. Right-click to return to the default cursor mode.

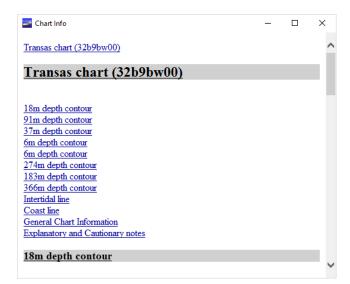
To return the unloaded chart or to display a loaded chart on top of other charts, use the following procedure:

- 1. Choose the Chart\Load Chart menu item, or click button on the toolbar, or press <Ctrl>+<L> key combination. The mouse cursor will switch to the chart loading mode.
- 2. Click on the boundary of the chart which should be loaded or shown on top of others. The selected chart will be displayed on top of the other charts. The chart boundary will be shown in the bold line.
- 3. Right-click to return to the default cursor mode.

2.2.6 Chart Information

The Instructor can obtain general information on any chart fragment. To do this, use the following procedure:

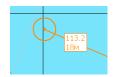
- 1. Choose the Chart\Chart Info menu item, or click the toolbar button, or press the <Ctrl>+<I> key combination.
- 2. By moving the Lucursor, select the chart fragment which the information should be obtained on, and click. The Chart Info window appears.



2.2.7 Measuring Bearing and Range

The bearing and range can be measured by using the electronic range and bearing line (ERBL). To do this, use the following procedure:

- 1. Choose the Chart\ERBL menu item, or click the toolbar button, or press the <Ctrl>+<E> key combination.
- 2. Position the cursor on the chart point, which the measurements are made relative to, and click. The ERBL origin appears on the chart.



- 3. Move the cursor on the chart. Bearing and range values will change in the pop-up window.
- 4. Right-click to return to the default cursor mode.

2.2.8 Setting Reference Point

To set a reference point on the chart, use the following procedure:

- 1. Choose the Chart\Reference Point menu item, or click the toolbar button, or press the <Ctrl>+<Shift>+<C> key combination.
- 2. Position the cursor over the chart where the reference point is required to be set, and click. The reference point appears on the chart. Reference point coordinates appear on the toolbar over the chart. Reference point coordinates appear on the toolbar over the chart. By moving the cursor on the chart, the Instructor can get readings of the bearing and distance from the reference point to the cursor in the status bar.
- 3. Right-click to return to the default cursor mode.

2.2.9 Changing Window Orientation

By default, the window is oriented North Up.

To rotate the window clockwise, press the $\langle Ctrl \rangle + \langle \rightarrow \rangle$ key combination.

To rotate it counter clockwise, press the <Ctrl>+<←>key combination.

The window orientation will be changing with a step of 15 degrees.

NOTE: The orientation angle will be shown in the status bar.

2.2.10 Display of Overlays

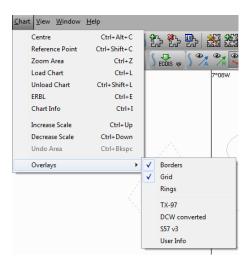
It is possible to show and hide on the chart the following auxiliary elements (overlays):

- chart borders, grid and range rings;
- TX-97 chart elements;
- DCW converted chart elements;
- · depth contours on the S-57 charts;
- user information (line, zone, text, ellipsis).

2.2.10.1 Chart Borders, Grid and Range Rings

To show/hide chart borders, grid and range rings, use the following procedure:

1. Choose the Chart\Overlays menu item.



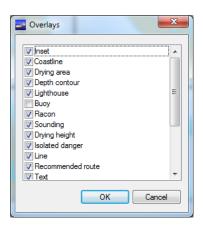
2. Select/clear checkboxes next to the chart elements in the sub menu: Borders, Grid, Rings.

NOTE: The range rings have origin in the reference point.

2.2.10.2 TX-97 Chart Elements

To show/hide the display of TX-97 chart elements, perform the following procedure:

- 1. Choose the Chart\Overlays\TX-97 menu item.
- 2. In the Overlays window, select/clear checkboxes next to the relevant items.

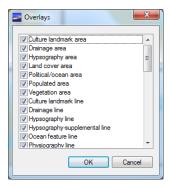


3. Click the OK button.

2.2.10.3 DCW Converted Chart Elements

To show/hide the display of DCW converted chart elements, use the following procedure:

- 1. Choose the Chart\Overlays\DCW converted menu item.
- 2. Select/clear checkboxes next to the relevant items.

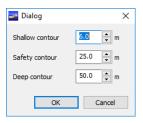


3. Click the OK button.

2.2.10.4 Depth Contours on S-57 Charts

To show/hide depth contours on S-57 charts, use the following procedure:

1. Choose the Chart\Overlays\S57 v3 menu item.

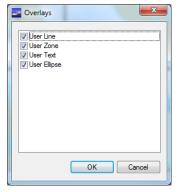


- 2. In the **Dialog** window, set the depth in meters to display **Shallow contour**, **Safety contour** and **Deep contour** on the chart.
- 3. Click the OK button.

2.2.10.5 User Information on Chart

To show/hide user information on the chart, perform the following procedure:

- 1. Choose the Chart\Overlays\User Info menu item.
- 2. Select/clear checkboxes next to the relevant items.



3. Click the OK button.

NOTE: To add user information to the chart, see "Adding User Information to Chart" on page 198.

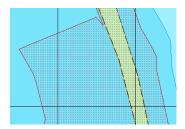
2.2.11 Depth Section

To select chart fragments where the depth values are equal to or smaller than the set value, use the following procedure:

1. Choose the View\Panels\Depth Section menu item or click the toolbar button. The Depth section window appears.



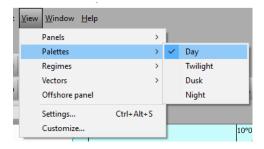
- 2. Select the Show section checkbox.
- 3. Enter the depth level (m) in the Section depth field.
- 4. Click the Set button.



2.2.12 Changing Colour Palette

The colour palette of the Instructor display can be selected to suit the current illumination.

To change the palette, choose the View\Palettes menu and select the required sub-menu item: Day, Twilight, Dusk, Night.



2.2.13 Readonly Mode

The Readonly mode is used to prevent accidental editing of objects as zones, routes, lines, points on the chart.

To switch on the Readonly mode, choose the Edit\Readonly Mode menu item or click the with toolbar button.

After the mode is switched on, the limits, dimensions and position of chart objects cannot be changed.

NOTE: This mode can be switched on/off anytime, regardless of the exercise status.

2.2.14 Selecting Motion Display Modes

Chart display may be turned into one of the following modes:

- TM 'Follow Up' true motion, Follow Up chart orientation (set by default);
- RM 'Nord Up' relative motion, Nord Up chart orientation;
- RM 'Head Up' relative motion, Head Up chart orientation;
- RM 'Fixed Heading' relative motion, fixed heading (the ship with selected heading is displayed at the center of the screen).

To select the true motion mode, check the View\Regimes\TM 'Follow Up' menu item, or click the View\TM 'Follow Up' menu item, or click the View\TM 'Follow Up' menu item, or click the View\TM 'Follow Up' menu item, or click the

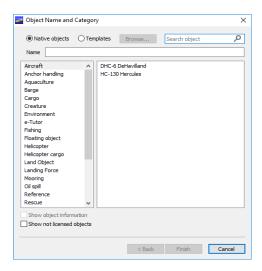
To select relative motion mode:

- 1. On the exercise chart, select the reference vessel.
- 2. To select the RM 'Nord Up' mode, check the View\Regimes\ RM 'Nord Up' menu item, or click the button, or press the <Ctrl>+<Alt>+<N> key combination.
- 3. To select the RM 'Head Up' mode, check the View\Regimes\ RM 'Head Up' menu item, or click the button, or press the <Ctrl>+<Alt>+<H> key combination.
- 4. To select the RM 'Fixed Heading' mode, check the View\Regimes\RM 'Fixed Heading' menu item, or click the toolbar button, or press the <Ctrl>+<Alt>+<X> key combination.

2.3 Adding New Object to Exercise

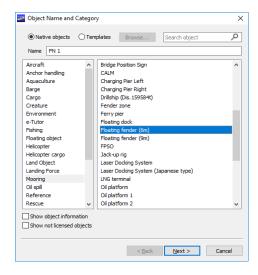
To add a new object to the exercise, use the following procedure:

1. Choose the Edit\New Object... menu item, or click the toolbar button, or press the <Ctrl>+<Shift>+<Ins> key combination. The Object Name and Category window appears. Objects are categorized according to their functionality. For more information about object categories, see Table 2.



NOTE: The **Object Name and Category** window can be opened from the **Objects** panel (for more information, see "Objects Panel" on page 50).

- 2. Make sure that the Native objects option is selected.
- 3. Select the object category from the list in the left-hand part of the window. If necessary, type the category or object name in the Search object part of the window. Search object is displayed in the right-hand part of the window.
- 4. Select the object from the list of objects.



NOTE: The symbol * shows that it is advisable to reinstall the object model.

- 5. If necessary, in the **Name** field, enter the object name that will be shown on the chart (the incorrect value is marked in red, and the further object setting will be impossible).
- 6. To show more information about the object, select the **Show object information** checkbox (if available). To close the information window, click
- 7. To show in the list the objects that were installed but not purchased under the license, select the Show not licensed objects checkbox. Object names appear in grey in the list. These objects will not be selectable unless a license is purchased.
- 8. Click the Next button.
- 9. Set the object initial parameters. If the object parameters are set correctly, the **Finish** button is available. If it is necessary to return to the parameter setting, click the **Back** button.
- 10. Click the Finish button.
- 11. Click the desired place on the chart. The object's contour and its name will be fixed on the chart.
- 12. Point the cursor on the object contour. In a second, the screen will display a pop-up window containing the object information.

Table 2. Categories of objects

CATEGORY	DESCRIPTION		
Aircraft	The category includes a collection of aircraft models which can be used in an exercise as target aircraft		
Anchor handling	The category contains objects which can be used for anchor handling operations		
Aquaculture	The category contains objects which can be used for fish farming operations		
Barge	The category contains barge models which can be used as targets in an exercise		
Creature	The category contains a set of animated creature objects. All the objects of this category can be seen both, visually and on the radar		
Environment	The category contains objects for adjusting the environment settings		
e-Tutor	The category contains the objects which can be used for trainee competence assessment		
Fishing	The category contains objects which can be used for fishing operations		
Floating objects	Objects of this category are used for preventing collisions with floating objects		
Helicopter	The category contains helicopter models which can be used as targets and deck helicopters		
Helicopter cargo	The category contains objects which can be used for helicopter vertical replenishment and medical evacuation operations		
Land Object	The category contains a set of 3D objects which can be added to a simulator scene as visual or radar landmarks		
Mooring	Objects of this category can be used in training of mooring operations		
Oil spill	The category contains objects which can be used in operations of the collection and pumping of oil in the areas of oil spills on the water surface		
Reference	The category contains the objects which can be used in exercises with DP system: acoustic transponder, acoustic transponder array, ROV. Also the category contains the remote sensor object which is used to monitor a remote object movement paramters		
Rescue	Objects of this category can be used in the search and rescue operations. Some objects are visible on the radar		

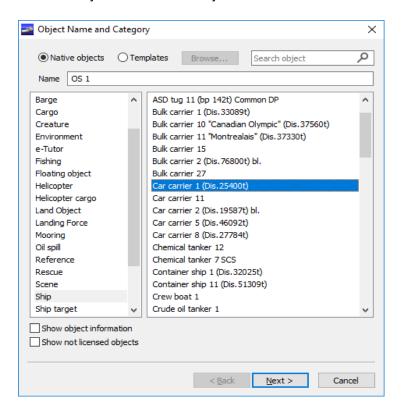
Table 2. Categories of objects (continued)

CATEGORY	DESCRIPTION
Scene	The category contains the following objects:
	AtoN station;
	Barge fleet - for the visual presentation of a group of barges (3 types);
	Camera - for setting the instructor visual observation camera;
	Route - for route creating;
	Wind turbine - for the visual presentation of a wind turbine (2 types)
Ship	The category contains ship models which can be used as own ships on bridge
Ship target	The category contains ship models which can be used as target ships in an exercise
Transport	The category contains a set of vehicles in the form of animated objects which can be used in exercises for creating environment. All the objects of this category can be seen both, visually and on the radar
Tug	The category contains tug models which can be used as own tugs on bridge
Tug target	The category contains tug models which can be used as tug-targets in an exercise
User objects	The category contains objects which can be used for adding user information to the chart in the form of symbol/line/text/ellipse/zone
VTS	This category contains the objects for joint operations with the VTS simulator

2.3.1 Adding Own Ship

To add a new own ship or tug object to the exercise, use the following procedure:

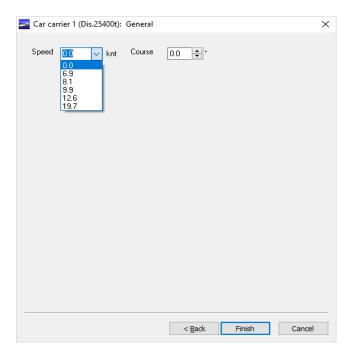
- 1. Choose the Edit\New object... menu item or click the toolbar button. The Object Name and Category window appears.
- 2. Select the **Ship** or **Tug** category from the category list in the left-hand part of the window. If necessary, type the object name in the Search object bar to search. The list of objects appears in the right-hand part of the window.
- 3. Select the object from the list of objects.



NOTES:

- To show more information about an object, select the Show object information checkbox (if available).
- To show the objects that were installed but not purchased under the license, select the Show not licensed objects checkbox. Object names appear in grey in the list. These objects will not be selectable unless a license is purchased.
- 4. In the **Name** field, enter the object that will be shown on the chart (if necessary). By default, the own ships are designated on the chart as "OS 1, 2..."; the own tugs are designated on the chart as "Tug 1, 2...").
- 5. Click the Next button.

Set the initial speed in the Speed field (it is advisable to select its value from the drop-down list corresponding to the discrete telegraph positions).



- 7. Set the initial course in the Course field.
- 8. For some objects it's possible to set the draft/trim in the Draft/Trim drop-down list.
- 9. For some vessels fitted out with controllable pitch propellers (CPP), you can select the "Combinator" mode by checking the **Combinatorial** checkbox. In this case, another telegraph table is used for the initial speed setting. An appropriate indicator (Combinator) lights up on the trainee bridge.
- 10. Click the Finish button.
- 11. Click the required place on the chart. The object's contour and its name will be fixed on the chart.

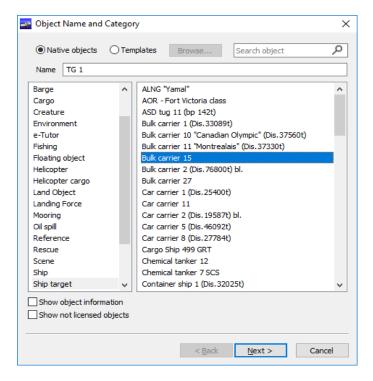


NOTE: To change the initial ship parameters, use the **General** tab of the ship properties panel (see "General Ship Properties" on page 107).

2.3.2 Adding Target Ship

To add a new target ship or target tug object to the exercise, use the following procedure:

- Choose the Edit\New object... menu item or click the toolbar button. The Object Name and Category window appears.
- 2. Select the **Ship target** or **Tug target** category from the category list in the left-hand part of the window. If necessary, type the object name in the search object bar to search. The list of objects appears in the right-hand part of the window.
- 3. Select the object from the list of objects.



NOTE: To show more information about the object, select the Show object information checkbox (if available).

- 4. In the **Name** field, enter the object name that will be shown on the chart (by default, "TG 1, 2..." for target ship and "TT 1, 2..." for target tug).
- 5. Click Next.
- Set the initial speed in the Speed field (it is advisable to select speed value from the drop-down list corresponding to the discrete telegraph positions).
- Set the initial course in the Course field.
- 8. Select the exercise time when the object will be created and added to the chart:
 - Start delay object creation time delay from the beginning of exercise;
 - Creation time object creation time (12.00.00 by default).
- 9. Select the type of a target model: simplified model or advanced (6-DoF) model:
 - To select a simplified target model, clear the 6 DoF model checkbox.
 - To select an advanced target model, select the 6 DoF model checkbox. In this case, the target ship's modeling level will correspond to the level of the own ship motion model.

- 10. If the simplified target model is selected:
 - a. Select the Pitch and roll checkbox to simulate pitch and roll (optional).
 - b. Click the Finish button.
- 11. If the 6-DoF target model is selected:
 - a. Set the Draft/Trim value (optional).
 - b. Select the Combinatorial mode for models with CPP (optional).
 - c. Click the Finish button.
- 12. Click on the chart. The object's contour and its name will be fixed on the chart:



NOTES:

- To change initial object parameters, use the **General** tab of the target ship properties panel (see "General Ship Properties" on page 107).
- To enable and set the Smart Target feature, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

2.4 Manipulations with Objects

2.4.1 Selecting Object on Chart

To perform any operations with the object (move, rotate, rename, delete, set the properties, etc.), it should be made active, i.e. selected on the chart.

To select an object on the chart, click the object contour. The object contour is shown in orange.



2.4.2 Opening Object Context Menu

To open the object context menu, right-click the object contour on the chart.

The object context menu contains the list of commands which can be carried out with this object.



To close the object context menu, click any place on the chart.

2.4.3 Moving Object

The move the object to another place on the chart, use the following procedure:

- 1. Right-click the object and choose the Move context menu item.
- 2. Move the cursor to a new place and click on the chart.

2.4.4 Rotating Object

To rotate the object, use the following procedure:

- 1. Right-click the object and choose the Rotate context menu item.
- 2. By moving the cursor, set the object's direction on the chart.



Click on the chart.

2.4.5 Renaming Object

To rename an object, use the following procedure:

1. Right-click the object and choose the **Rename** context menu item. The **Rename object** window appears.

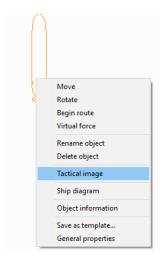


2. Enter a new object name and click the OK button.

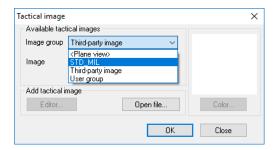
2.4.6 Setting Ship Tactical Image

To set a ship tactical image on the chart, use the following procedure:

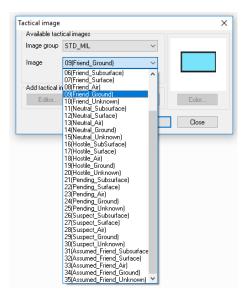
1. Right-click the object and choose the **Tactical image** context menu item.



- 2. From the Image group drop-down list, select the group of tactical images:
 - < Plane view > object's default image.
 - Third-party image downloaded image files (jpg, png or bmp format).
 - STD_MIL standard military tactical symbols.
 - User group tactical images created in the internal editor.



3. From the Image drop-down list, select a tactical image.



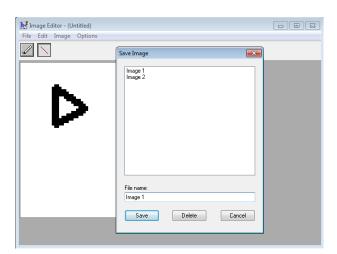
4. Click OK in the Tactical image window. The object tactical image appears on the chart.

To load an image in the jpg, png or bmp format:

- 1. In the Tactical image window, click the Open file... button.
- 2. In the Open window, select the required file and click the Open button. The image will be added to the Third-party image group of the Tactical image window.

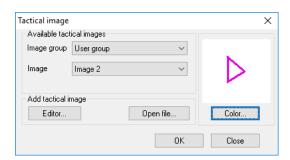
To create a tactical image in the internal editor:

- 1. In the **Tactical image** dialogue box, select the **User group** and click the **Editor...** button. The **Image Editor** window appears.
- 2. On the white field, use the buttons to draw the object tactical image.
- 3. Choose the File\Save As... menu item.
- 4. In the Save Image dialogue box, enter the tactical image name and click the Save button.



5. Close the Image Editor window.

- 6. In the Tactical image window, select the User group from the Image group drop-down list.
- 7. From the Image drop-down list, select the created tactical image.
- 8. To change an image colour, click the Color... button and select the desired colour in the Color dialog box.

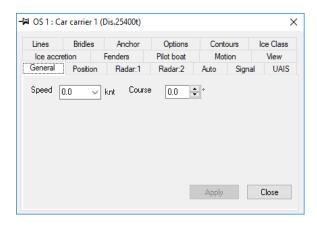


9. Click the OK button in the Tactical image window. The object tactical image appears on the chart.



2.4.7 Opening Object Properties Panel

The object properties panel contains tabs for setting the object properties.



To open the object properties panel, use the following procedure:

- 1. Select object on the exercise chart.
- 2. Use one of the following actions:
 - o Right-click the object and choose the General properties context menu item.

NOTE: To open the properties panel for route waypoints, segments, and line attachment points, choose the **Properties** context menu item.

- o Choose the Edit\Object properties... menu item.
- Click the toolbar button.
- Use the Objects panel (for more information, see "Objects Panel" on page 50).

NOTE: The appearance of the object properties panel before and after the start of the exercise may differ (for more information about the ship properties, see "Setting Ship Properties" on page 105).

2.4.8 Deleting Object

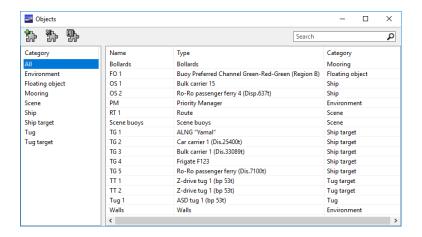
To delete an object, use the following procedure:

- 1. Select an object on the exercise chart.
- 2. Use one of the following actions:
 - Open the object's context menu and choose the Delete Object item.
 - Click the button on the toolbar.
 - Choose the Edit\Delete Object menu item.
 - Press the <Ctrl>+<Shift>+ key combination.
 - Use the **Objects** panel (for more information, see "Objects Panel" on page 50).
- 3. Click the OK button to confirm the object deletion.

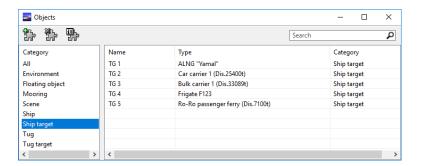
2.4.9 Objects Panel

The **Objects** panel contains a list of all exercise objects. To use the panel, perform the following operations:

1. Choose the View\Panels\Objects...menu item, or click the toolbar button to open the Objects panel.

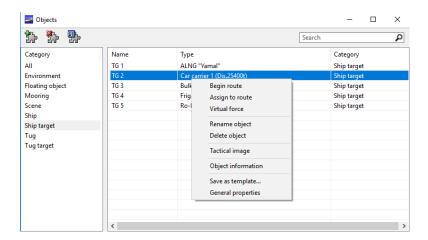


2. Select the object category from the **Category** list. A list of objects of this category is displayed in the right part of the window.



- 3. To sort the objects by the name, type and category, click on the respective table header.
- 4. To quickly find an object or an object category in the **Objects** window, enter the full or the partial name of the object or the category in the search bar . To clear the search line, click .

5. To open the object context menu, double-right-click the name of the object in the table.

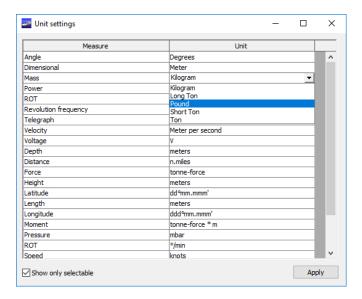


- 6. To add a new object to the exercise, click the button. The **Object Name and Category** window appears on the screen (for more information, see "Adding New Object to Exercise" on page 39).
- 7. To delete an object: select the name of the object in the table, click the button and click OK.
- 8. To open the object properties panel: select the name of the object in the table and click the button.
- 9. To display the object in the center of the chart, double-left click on the object name in the table.

2.5 Selection of Measurement Units

To select measurement units in an exercise, use the following procedure:

- 1. Choose the Edit\Units Settings... menu item. The Unit settings window appears.
- 2. To show only selectable values of measurement units in the window, select the Show only selectable checkbox.
- To change the measurement unit, click the unit in the Unit column and select the required value from the dropdown list. Click the Apply button.



4. To close the window, click X.

2.6 General Exercise Settings

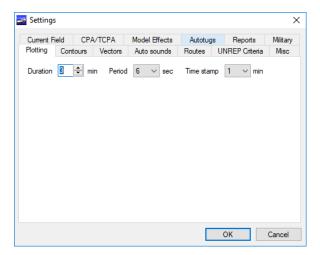
The **Settings** panel (**View\Settings...** menu) is used for adjusting general exercise settings. It contains the following tabs:

- Plotting displaying ship track as a series of dots on the chart (see "Displaying Ship Track as Series of Dots" on page 53);
- Contours displaying ship track as a series of ship contours on the chart (see "Displaying Ship Track as Series of Contours" on page 54);
- Vectors speed vector settings (see "Speed Vectors Settings" on page 55);
- Routes route settings (see "General Route Settings" on page 122);
- Auto sounds enabling/disabling automatic sound signals (see "Automatic Sound Signals Settings" on page 57);
- CPA/TCPA CPA/TCPA settings (see "General CPA/TCPA Settings" on page 234);
- UNREP Criteria underway replenishment criteria (see "Setting Underway Replenishment Criteria" on page 410);
- Current Field current field object settings (see "Current Field General Settings" on page 94);
- Autotugs auto tugs settings (see "Automatic Tug General Settings" on page 331);
- Reports report settings (see "Report Settings" on page 285);
- Model Effects setting hydrodynamic interaction model effects (see "Model Effects Settings" on page 57);
- Misc miscellaneous settings (see "Miscellaneous Settings" on page 58).

2.6.1 Displaying Ship Track as Series of Dots

To show the ship track as a series of dots, use the following procedure:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press <Ctrl>+<Alt>+<S> key combination.
- 2. Open the Plotting tab on the Settings panel.



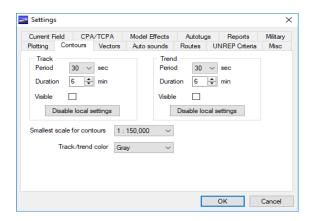
- 3. Set the following parameters:
 - o Duration history length, min;
 - o Period plot interval, s;
 - Time stamp time tag interval, min.
- 4. Click the OK button.



2.6.2 Displaying Ship Track as Series of Contours

To show the ship track as a series of ship contours, use the following procedure:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press the <Ctrl>+<Alt>+<S> key combination.
- 2. Open the Contours tab on the Settings panel.



- 3. In the Track group box, adjust settings to show the ship track as series of ship contours:
 - a. Set the contour display interval from the Period drop-down list, sec.
 - b. Set the track length in the **Duration** field, min.
 - c. To display contours, check the Visible checkbox, or click the toolbar button
 - d. To disable local settings, click the Disable local settings button (as required).
- 4. In the **Trend** group box adjust settings to show the estimated ship track as series of ship contours:
 - a. Set the contour display interval from the Period drop-down list, sec.
 - b. Set the track length in the **Duration** field, min.
 - c. To display contours, check the Visible checkbox, or click the loolbar button.
 - d. To disable local settings, click the Disable local settings button (as required).

- 5. Select the minimum contour display scale from the **Smallest scale for contours** drop-down list. If the scale is less, than set value, the ship contours assume the form of a dot.
- 6. Select colour of the contours from the Track/trend color drop-down list (Grey as default).
- 7. Click the OK button.

The ship track will be shown as a series of the ship contours, in the form of:

· tracks;



trends.

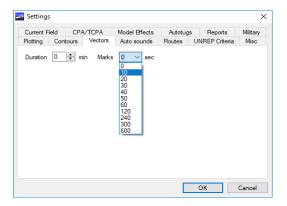


NOTE: For more information about local settings of the contours display, see "Contours Settings" on page 116.

2.6.3 Speed Vectors Settings

To set ship's speed vector parameters, use the following procedure:

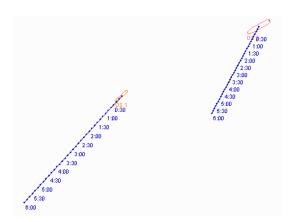
- 1. Choose the View\Settings... menu item or click the toolbar button, or press the <Ctrl>+<Alt>+<S> key combination.
- 2. Open the Vectors tab on the Settings panel.



- 3. Set the following parameters:
 - Duration time duration;
 - Marks interval between time marks.
- 4. Click the OK button.

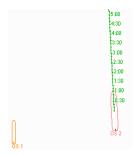
To switch on the display of absolute speed vectors, choose the View\Vectors\Absolute menu item, or press the <Ctrl>+<Alt>+<A> key combination, or click the toolbar button.

Absolute speed vectors will be displayed on the chart in blue.

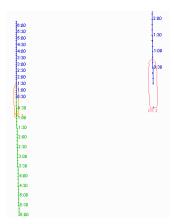


To switch on the display of relative speed vectors, do the following:

- 1. On the exercise chart, select the reference vessel.
- 2. Choose the View\Vectors\Relative menu item, or press the <Ctrl>+<Alt>+<R> key combination, or click the toolbar button. The relative speed vectors will be displayed on the chart in green.



It is possible to display absolute and relative speed vectors simultaneously.



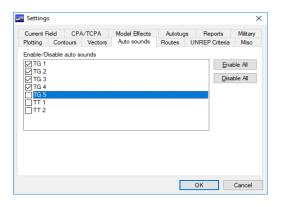
To switch off the display of speed vectors, clear the **Absolute** and **Relative** menu item, or click the and toolbar buttons again.

2.6.4 Automatic Sound Signals Settings

During a simulator session, the target ship automatically produces sound signals in low visibility (fog) regardless of the time of the day.

To disable/enable automatic sound signals, use the following procedure:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press the <Ctrl>+<Alt>+<S> key combination.
- 2. Open the **Auto sounds** tab on the **Settings** panel. Select checkboxes the target ships which are allowed to make automatic sound signals and click the **OK** button.

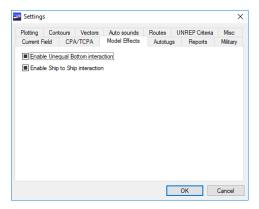


- 3. To disable automatic sound signals for all target ships, click the Disable All button and click the OK button.
- 4. To enable automatic sound signaling for all target ships, click the Enable All button and click the OK button.

2.6.5 Model Effects Settings

To enable/disable the "Unequal Bottom" and "Ship to Ship" hydrodynamic interaction effects:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press the <Ctrl>+<Alt>+<S> key combination.
- 2. Open the **Model Effects** tab of the **Settings** panel.

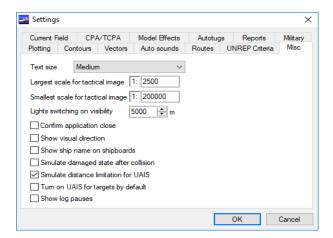


- 3. Click several times the **Enable Unequal Bottom interaction** and **Enable Ship to Ship interaction** checkbox buttons to select one of three effect states:
 - the effect is applied depending on the coefficient set on the **Options** tab of the ship properties panel (see "Specifying Options" on page 114);
 - \Box the effect is disabled for all ships;
 - the effect is enabled for all ships.
- 4. Click the OK button to apply settings.

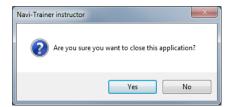
2.6.6 Miscellaneous Settings

To adjust miscellaneous settings, use the following procedure:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press <Ctrl>+<Alt>+<S> key combination.
- 2. Open the Misc tab on the Settings panel.



- 3. From the Text size drop-down list, select the text size on the chart: small, medium, large, very large.
- 4. In the Largest scale for tactical image field, set the maximum scale for the tactical image. If the chart scale is larger than the selected value, the ship contour assumes a standard view.
- 5. In the **Smallest scale for tactical image** field, set the minimum scale for the tactical image. If the chart scale is smaller than the selected value, the ship contour assumes the form of a dot.
- 6. In the **Light switching on visibility** field, set the visibility distance of navigation lights. If the visibility distance is smaller than the set value, signal lights are automatically turned on; if it is larger than the set value, signals are given with the aid of shapes.
- 7. To confirm the Navi-Trainer Instructor application closing, select the **Confirm application close** checkbox. When you are closing the application, in the dialogue box which will appear, click **Yes** to exit the program, or click **No** to return to the work with the application.



8. To set the display of visual direction on the bridge, select the **Show visual direction** checkbox. After the simulation session start, an arrow on the chart will show the visual direction on the bridge.



NOTE: The visual direction is displayed if the **ShowDirection** visualization parameter is specified in the Configuration Editor.

- 9. To show the ship names on the ship side, select the Show ship names on shipboards checkbox.
- 10. To trigger the damaged or wrecked state for the vessels after serious impact, select the **Simulate damaged state after collision** checkbox.
- 11. To simulate UAIS distance limitations, select the **Simulate distance limitations for UAIS** checkbox. If the checkbox is removed, the targets will be tracked at any distance.
- 12. For UAIS transponders to be turned on by default on all the target ships, select the **Turn on UAIS for targets by default** checkbox.
- 13. To show exercise pauses periods on the log time bar during the log playback, select the **Show log pauses** checkbox.
- 14. Click the **OK** button to apply settings.

2.7 Setting Environmental Sailing Conditions

Environmental sailing conditions can be specified for the entire exercise scene or within a certain zone on the exercise chart.

The Instructor can set the following environmental conditions:

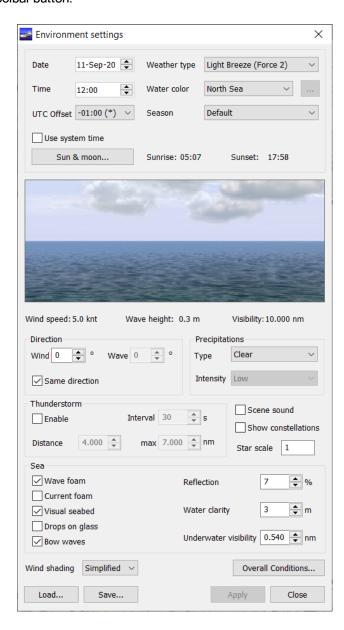
- Environment settings: date and time, weather type, colour of water, season, etc. (see "Environment Settings" on page 61)
- Overall environmental conditions: current, wind, wave, visibility, sky, tide, etc. (see "Overall Environmental Conditions" on page 71)
- Local environmental conditions which are set within the composite condition zone, such as: current, depth, fog, muddy strata, sky, tide, wind, etc. (see "Local Environmental Conditions" on page 84)
- Tides and currents from the common databases (see "Importing Tides and Currents" on page 91)
- Vector field of currents (see "Creating Vector Field of Current" on page 94)
- Longshore current (see "Setting Longshore Current" on page 96)
- Clouds (see "Setting Clouds" on page 98)
- Atmosphere fronts (see "Setting Atmosphere Fronts" on page 100)
- Wave spectrum (see "Editing Wave Spectrum" on page 102)
- Ice zones (see "Operations in Ice Conditions" on page 344).

2.7.1 Environment Settings

Environment settings are specified for the entire sailing area before the exercise start and during the simulator session.

To open the Environment settings window, choose the Edit\Environment Settings... menu item, or click the toolbar button.





In the Environment settings window, the following environmental conditions can be specified:

- exercise date and time (see on page 62);
- weather type (see on page 63);
- colour of water (see on page 66);
- season (see on page 67);
- the Sun and Moon position (see on page 67);
- wind and wave direction (see on page 68);

- precipitations (see on page 69);
- thunderstorm conditions (see on page 69);
- sea visual presentation (see on page 69);
- scene sounds (see on page 70);
- constellation display (see on page 70);
- wind shading (see on page 70).

2.7.1.1 Exercise Date and Time

Exercise start date and time in combination with the area latitude and longitude determine angular positions of heavenly bodies (sun, moon, stars) and, accordingly, exercise scene illumination. Besides, the exercise start date and time affect the time dependent hydrometeorological data values (e.g., tides and currents).

NOTE: The Instructor can select exercise date and time only before the exercise start.

Setting Exercise Date

To set the exercise start date, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the Date field, set the exercise date and click Apply.

Setting Exercise Time

To set the exercise start time, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Time** field, set the exercise start time (12.00.00 is a default time). Or select the **Use system time** checkbox to use the current system time.
- 3. From the UTC Offset drop-down list, select UTC offset (the symbol * marks the UTC offset for the current sailing area).
- 4. Click Apply.

There are two types of time readings: absolute or relative. The absolute time is the time of the day; relative time is the time from the exercise start.

To select the time type:

- 1. Right-click the time reading in the status bar at the bottom of the screen.
- 2. Choose the **Absolute** or **Relative** time type.

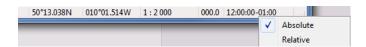
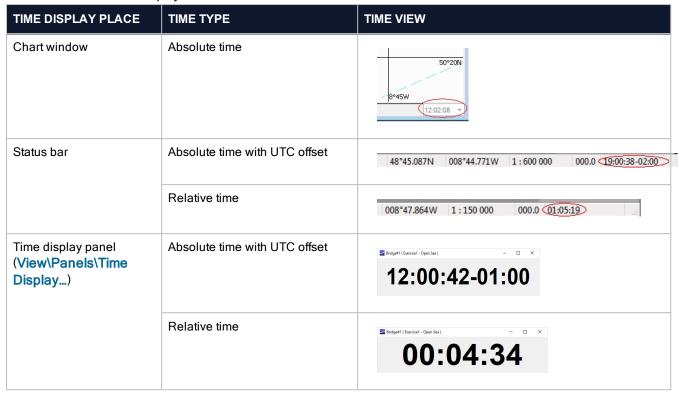


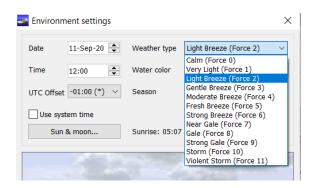
Table 3. Exercise time display



2.7.1.2 Weather Type

To select the type of weather conditions, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. From the **Weather type** drop-down list, select the necessary type of weather conditions (for more information about weather types, *see Table 4*).



In the centre of the **Environment settings** window, a sample of the selected weather condition is displayed and main weather parameters are specified: **Wind speed**, **Wave height**, **Visibility**.



3. Click Apply.

Table 4. Weather condition types

WEATHER TYPE (WIND FORCE)	PARAMETERS		VIEW
Calm	Cloud layer:	Cirrus	
(force 0)	Wind speed (knt):	0	
	Wave height (m):	0	
	Sky clarity (%):	100	
	Visibility (nm):	10	
Very Light	Cloud layer:	Cirrus	
(force 1)	Wind speed (knt):	2	
	Wave height (m):	0.1	
	Sky clarity (%):	100	
	Visibility (nm):	9.0	
Light Breeze	Cloud Layer:	Altocumulus	
(force 2)	Wind speed (knt):	5	
	Wave height (m):	0.4	
	Sky clarity (%):	100	
	Visibility (nm):	8.0	
Gentle Breeze	Cloud Layer:	Cirrocumulus	and the second
(force 3)	Wind speed (knt):	8	
	Wave height (m):	0.8	
	Sky clarity (%):	100	
	Visibility (nm):	10	

Table 4. Weather condition types (continued)

WEATHER TYPE (WIND FORCE)	PARAMETERS		VIEW
Moderate Breeze	Cloud Layer:	Cumulus mediocris	
(force 4)	Wind speed (knt):	13	
	Wave height (m):	1.3	
	Sky clarity (%):	70	
	Visibility (nm):	5.0	
Fresh Breeze	Cloud Layer:	Cirrostratus	
(force 5)	Wind speed (knt):	19	
	Wave height (m):	2.5	
	Sky clarity (%):	70	
	Visibility (nm):	10	
Strong Breeze	Cloud Layer:	Stratocumulus	
(force 6)	Wind speed (knt):	25	
	Wave height (m):	3.5	
	Sky clarity (%):	70	
	Visibility (nm):	10	
Near Gale	Cloud Layer:	Stratus with lighting	
(force 7)	Wind speed (knt):	31	
	Wave height (m):	4.7	
	Sky clarity (%):	50	
	Visibility (nm):	10	
Gale (force 8)	Cloud Layer:	Stratus	
	Wind speed (knt):	37	
	Wave height (m):	5.5	
	Sky clarity (%):	40	
	Visibility (nm):	7.0	
Strong Gale	Cloud layer:	-	-
(force 9)	Wind speed (knt):	44	
	Wave height (m):	7.8	
	Sky clarity (%):	10	
	Visibility (nm):	5.0	

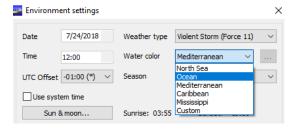
Table 4. Weather condition types (continued)

WEATHER TYPE (WIND FORCE)	PARAMETERS		VIEW
Storm	Cloud layer:	-	
(force 10)	Wind speed (knt):	51	
	Wave height (m):	9.0	
	Sky clarity (%):	0	
	Visibility (nm):	2.0	
Violent storm	Cloud layer:	-	
(force 11)	Wind speed (knt):	58	
	Wave height (m):	11.0	
	Sky clarity (%):	0	the shift comment of the shift
	Visibility (nm):	0.5	

2.7.1.3 Colour of Water

To set the colour of water, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. From the Water color drop-down list select the geographic area: North Sea (by default), Ocean, Mediterranean, Caribbean, Mississippi.



3. Or set the colour of water for a specific area: select **Custom** from the **Water color** drop-down list, click the button, select water colour from the **Color** window, and click **OK**.

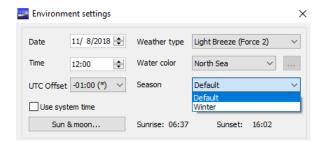


4. Click Apply. The colour of water will be changed on the visualization screen.

2.7.1.4 Season

To select a season, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the latest toolbar button.
- 2. From the **Season** drop-down list select the season:
 - Default spring and summer time;
 - o Winter winter time.



3. Click Apply.

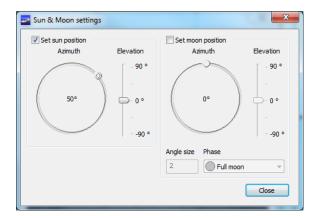
2.7.1.5 Sun and Moon Position

The Instructor can set positions of the Sun and Moon. The Sun and Moon settings affect the exercise scene illumination and, accordingly, the turning on of the navigational lights at the dark time of the day. In this case, the exercise date and time remain unchanged.

NOTE: The Sun and Moon positions are set relative to the training area centre.

To set the Sun position, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. Click the Sun & moon... button. The Sun & Moon settings window appears.
- 3. Select the **Set sun position** checkbox.
- 4. Set an angle between the horizon and position of the Sun disk by moving the slider along the **Elevation** scale (90° corresponds to the summer solstice point, 0° to the sun position on the horizon).
- 5. Set the azimuth to the Sun position by moving the slider along the Azimuth scale.

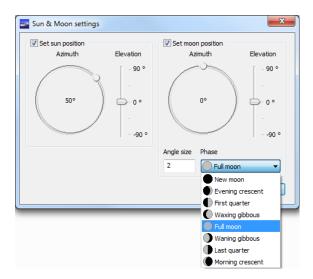


- Click the Close button in the Sun & Moon settings window.
- 7. Click the Apply button in the Environment settings window.

NOTE: For reference, in the **Environment settings** window near the **Sun & moon...** button the **Sunrise** and **Sunset** time is specified as of the set exercise fulfillment date.

To set the Moon position, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. Click the Sun & moon... button. The Sun & Moon settings window will appear.
- 3. Select the **Set moon position** checkbox.
- 4. Set an angle between the horizon and position of the Moon by moving the slider along the **Elevation** scale (0° corresponds to the Moon position on the horizon).
- 5. Set the azimuth to the Moon position by moving the slider along the Azimuth scale.
- 6. Use the Angle size field to enter the Moon disk aperture angle (0°-15°).
- 7. From the Phase drop-down list, select the required moon phase.



- 8. Click Close in the Sun & Moon settings window.
- 9. Click Apply in the Environment settings window.

2.7.1.6 Wind and Wave Direction

To set the wind and wave direction, use the following procedure:

 Choose the Edit\Environment Settings... menu item, or click the toolbar button. The Direction group box of the Environment settings window is used for setting parameters.

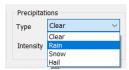


- 2. Set the wind direction in the Wind field.
- Set the wave direction in the Wave field, or select the Same direction checkbox to set the wave direction identical to the wind direction.
- 4. Click Apply.

2.7.1.7 Precipitations

To enable precipitations, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. In the Precipitations group box, open the Type drop-down list and select the precipitation type: Rain, Snow, Hail.



NOTE: To disable precipitations, select Clear and click Apply.

- 3. From the Intensity drop-down list, select the precipitation intensity: Low, Medium, High.
- 4. Click Apply.

2.7.1.8 Thunderstorm Conditions

To set the thunderstorm conditions, use the following procedure:

1. Choose the Edit\Environment Settings... menu item, or click the toolbar button. The Thunderstorm group box of the Environment settings window is used for setting parameters.



- 2. Select the Enable checkbox.
- 3. In the Interval field, set thunderstorm time interval (s).
- 4. In the **Distance min/max** fields, set the minimum and maximum distance from the ship to the display of lightning (nm).
- 5. Click Apply.

2.7.1.9 Sea Visual Presentation

To set the sea visual presentation, use the following procedure:

1. Choose the Edit\Environment Settings... menu item, or click the toolbar button. The Sea group box of the Environment settings window contains the set of parameters for adjusting the sea visual presentation.



- 2. Select the Wave foam checkbox to display sea foam generated by waves.
- 3. Select the Current foam checkbox to display sea foam generated by the difference of currents.
- 4. Select the Visual seabed checkbox to display detailed seabed.
- 5. Select the **Drops on glass** checkbox to display water drops on the cabin glass.

- 6. Select the Bow waves checkbox to display bow waves.
- 7. In the **Reflection** field, set the value of reflection of the ship, clouds and other objects on the water surface (%).
- 8. In the Water clarity field, set the depth through the water surface that is visible (m).
- 9. In the **Underwater visibility** field, set the visibility range underwater (nm).
- 10. Click Apply.

2.7.1.10 Scene Sounds

The scene sounds can be switched on/off for some areas. To do this, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. Select/clear the Scene sound checkbox.
- 3. Click Apply.

2.7.1.11 Constellation Display

To display constellations on the visualization screen, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the at toolbar button.
- 2. Select the Show constellations checkbox.
- 3. In the Star scale, enter the star scale (1-5).
- 4. Click Apply.

2.7.1.12 Wind Shading

To enable the wind shadow effect, when the wind speed is reduced near an obstacle (for example, near the ship hull), use the following procedure:

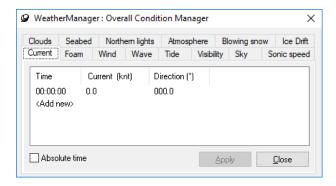
- 1. Choose the Edit\Environment Settings... menu item, or click the length toolbar button.
- 2. From the Wind shading drop-down list, select the required option:
 - Simplified to create the wind shadow zone near the ship hull.
 - Advanced to create the wind shadow zone near the ship hull, berth, bridges supports.
 - o Off to disable the wind shadow effect.
- 3. Click Apply.

2.7.2 Overall Environmental Conditions

Overall environmental conditions are set for the entire sailing area in the Overall Condition Manager window.

To open the Overall Condition Manager window, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button. The Overall Condition Manager window appears.



The Overall Condition Manager window contains tabs for setting the following overall condition parameters:

- Atmosphere (see "Atmosphere Properties" on page 71)
- Blowing snow (see "Blowing Snow Properties" on page 72)
- Clouds (see "Clouds Properties" on page 73)
- Current (see "Properties of Current" on page 73)
- Foam (see "Foam Properties" on page 74)
- Ice Drift (see "General Ice Drift Settings" on page 356)
- Northern lights (see "Northern Lights Properties" on page 74)
- Seabed (see "Seabed Properties" on page 75)
- Sky (see "Sky Properties" on page 76)
- Sonic speed (see "Sonic Speed Properties" on page 76)
- Tide (see "Tide Properties" on page 77)
- Visibility (see "Visibility Properties" on page 78)
- Wave (see "Wave Properties" on page 79)
- Wind (see "Wind Properties" on page 81).

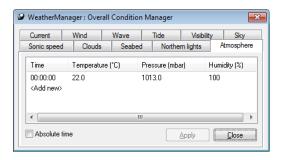
NOTE: Values of some weather conditions previously set in **Environment settings** window (see "Environment Settings" on page 61), are passed automatically to the relevant tabs of the **Overall Condition Manager** window.

2.7.2.1 Atmosphere Properties

To set the atmosphere properties, use the following procedure:

- 1. In the Overall Condition Manager window, open the Atmosphere tab.
- 2. In the Temperature column, enter the atmospheric temperature.
- 3. In the Pressure column, enter the atmospheric pressure.

4. In the Humidity column, enter relative humidity.

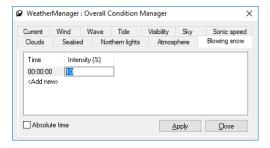


- 5. Select the **Absolute time** checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the < Delete > key.
- 7. Click the **Apply** and **Close** buttons.

2.7.2.2 Blowing Snow Properties

To set the blowing snow properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Blowing snow tab.

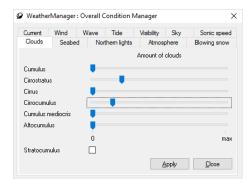


- 4. In the Intensity column, enter the snowdrift intensity (%).
- 5. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - Set the parameter value.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the < Delete > key.
- 5. Click the Apply and Close buttons.

2.7.2.3 Clouds Properties

To set the clouds properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the latest toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager box, open the Clouds tab.
- 4. By using the sliders, set amount of clouds for the required cloud type.

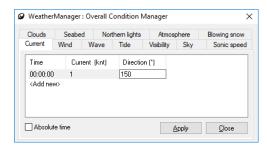


- 5. To set a stratocumulus cloud layer, select the Stratocumulus checkbox.
- 6. Click the Apply and Close buttons.

2.7.2.4 Properties of Current

To set properties of current, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Current tab.
- 4. In the Current column, enter the current velocity (knots).
- 5. In the Direction column, enter the current direction (deg).



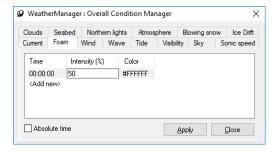
- 6. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 7. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete > key.

8. Click the Apply and Close buttons.

2.7.2.5 Foam Properties

To set foam properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Foam tab.
- 4. In the Intensity column, enter the percentage of foam on the water surface.

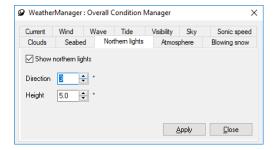


- 5. To change the default foam colour (white): click the colour value in the Color column, select the desired colour in the Color window and click OK. The new colour value appears in the Color column.
- 6. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 7. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the < Delete > key.
- 5. Click the Apply and Close buttons.

2.7.2.6 Northern Lights Properties

To set the northern lights properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Northern lights page.
- 4. Select the Show northern lights checkbox to display the northern lights on the visualization screen.

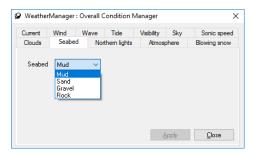


- 5. In the **Direction** field, set the northern lights direction (0-359.9 degrees).
- 6. In the **Height** field, set an angle between the horizon and the bottom limit of the northern light display on the visualization screen (0-60 degrees).
- 7. Click the Apply and Close buttons.

2.7.2.7 Seabed Properties

To select the seabed type for the entire sailing area:

- 1. Choose the Edit\Environment Settings... menu item, or click the ____ toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Seabed tab.
- 4. Select the seabed type from the Seabed drop-down list: Mud, Sand, Gravel, Rock.



5. Click the Apply and Close buttons.

The dangerous ground touch speed depends on the seabed type if the Simulate damaged state after collision checkbox is selected on the Misc tab of the Settings panel (see "Miscellaneous Settings" on page 58). In this case, the vessel's touching the ground at a speed that exceeds the dangerous speed for this type of seabed will cause damage to the vessel (see Table 5).

Table 5. Dangerous speeds for different types of seabed

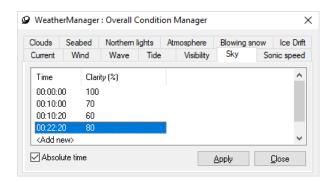
SEABED TYPE	MAX SPEED (M/S)
Mud	2
Sand	1.5
Gravel	1
Rock	0.5

NOTE: The Instructor can set the anchor holding power for all seabed types, see "Setting Anchor Properties" on page 306.

2.7.2.8 Sky Properties

To set the sky properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Sky tab.
- 4. In the Clarity column, enter the sky clarity (%).
- 5. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set the parameter value.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.

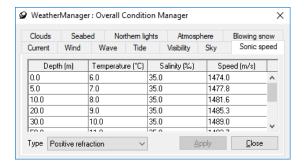


5. Click the Apply and Close buttons.

2.7.2.9 Sonic Speed Properties

To set sonic speed properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- In the Overall Condition Manager window, open the Sonic speed tab.



4. Select one of standard vertical sonic speed distribution (VSSD) from the **Type** drop-down list: **Negative refraction**, **Positive refraction**, **Channel**, **Thermocline**.

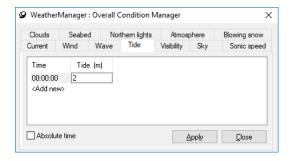
- 5. To change the standard settings, enter the temperature value in the Temperature column and/or enter the salinity value in the Salinity column for any selected depth level. With the input of temperature and/or salinity, the corresponding sonic speed will be re-calculated automatically and the User-defined type of VSSD appears in the Type field.
- 6. Click the Apply and Close buttons.

NOTE: The sonic speed parameter values have an effect on fish-finding and hydro-acoustic devices.

2.7.2.10 Tide Properties

To set the tide properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Tide tab.
- 4. In the Tide column, enter the tide height (m).

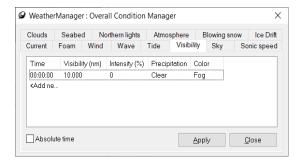


- 5. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set the parameter value.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.
- 5. Click the Apply and Close buttons.

2.7.2.11 Visibility Properties

To set the visibility properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Visibility tab.
- 4. In the Visibility column, enter the visibility range (nm).



- 5. In the Intensity column, enter the precipitation intensity from 0 to 100 %.
- 6. In the **Precipitation** column, open the drop-down list and select the type of precipitations: **Rain**, **Snow**, **Hail**. Or select **Clear** to disable precipitations.
- 7. In the Color column, open the drop-down list and select the parameter to reduce visibility: Fog or Dust.
- 8. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 9. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete > key.
- 5. Click Apply and Close.

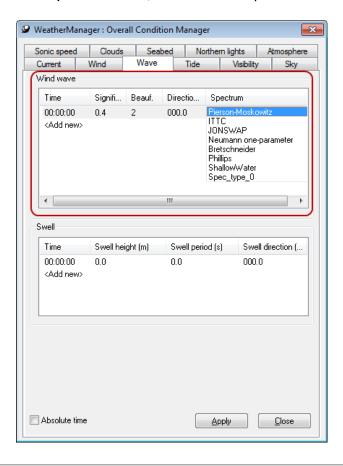
2.7.2.12 Wave Properties

On the Wave tab of the Overall Condition Manager window, the Instructor can set the wind wave and swell properties.

Setting Wind Wave

To set the wind wave properties, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the ____ toolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Wave tab.
- 4. In the Wind wave group box, in the Significant height column, enter the wind wave significant height in meters.
- 5. In the Beauf. column, enter the Beaufort force value.
- 6. In the **Direction** column, enter the wind wave direction in degrees.
- 7. In the Spectrum column, select the wave spectrum from the drop-down list.



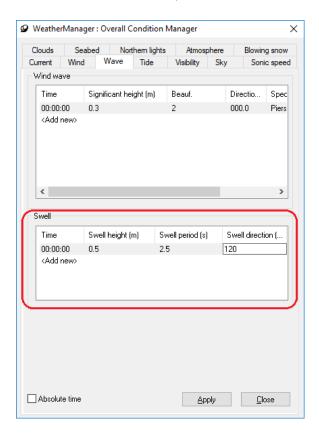
NOTE: The user can create and edit wave spectrum on the **Spectrum Editor** panel (see "Editing Wave Spectrum" on page 102).

- 8. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 9. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - In the Time column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete > key.
- 5. Click the **Apply** and **Close** buttons.

Setting Swell

To set the swell properties, use the following procedure:

- 1. In the Overall Condition Manager window, open the Wave tab.
- 2. In the Swell group box, in the Swell height, enter the swell height in meters.
- 3. In the Swell period column, enter swell period in seconds.
- 4. In the Swell direction column, enter swell direction in degrees.



- 5. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.
- 5. Click the Apply and Close buttons.

2.7.2.13 Wind Properties

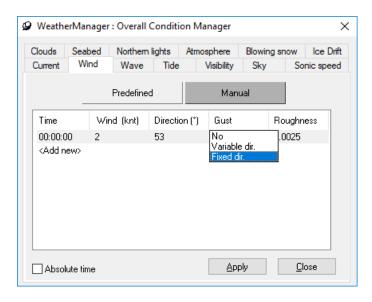
To set the wind properties, perform the following actions:

- Choose the Edit\Environment Settings... menu item, or click the loolbar button.
- 2. In the Environment settings window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Wind tab.
- 4. On the Predefined page, set the wind speed, direction, and wind gust effect (see "Procedure 1").
- After the start of the exercise, on the Manual page, change the wind speed and direction (see "Procedure 2").

Procedure 1. Predefined Settings

To set the wind speed, direction, and wind gust:

- 1. Click the Predefined button (the Predefined page is active by default).
- 2. In the Wind column, enter the wind speed.
- 3. In the **Direction** column, enter the wind direction.
- 4. To enable the wind gust effect:
 - a. In the Gust column, open the drop-down list and select the required option: No to disable the wind gust effect; Variable dir. - to select variable wind gust direction; Fixed dir. - to select fixed wind gust direction as set in the Direction column.



b. In the **Roughness** column, set a surface roughness coefficient within a range of 0.0001 to 0.005 (typically 0.0025 for sea surface with large waves).

NOTE: The wind gust is calculated as per Davenport Spectrum.

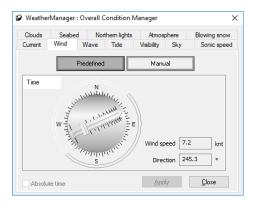
- 5. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 6. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the < Delete > key.

5. Click the Apply and Close buttons.

Procedure 2. Manual Settings

To change the wind direction and speed immediately (not available until the exercise start):

- 1. Click the Manual button.
- 2. Set the wind speed by moving the scroll box on the arrow indicator logarithmic scale. The **Wind speed** indicator shows the current wind speed.
- Set the wind direction by clicking the compass scale and rotating the arrow indicator. The **Direction** indicator shows the current wind direction.

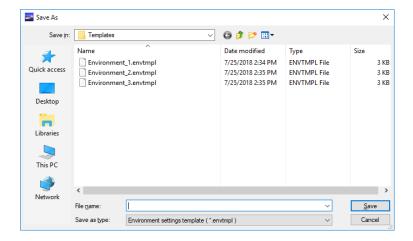


- To edit predefined wind parameters, click the Predefined button (settings made on the Manual page are not saved).
- 5. To close the Overall Condition Manager window, click the Close button.

2.7.3 Environment Conditions Template

To create an environment conditions template, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the late toolbar button.
- 2. In the **Environment settings** window, set the required properties and click the **Apply** button (for more information, see "Environment Settings" on page 61).
- Click the Overall Conditions button and set properties of overall conditions parameters in the Overall Condition
 Manager window. Click the Apply and Close buttons (for more information, see "Overall Environmental
 Conditions" on page 71).
- 4. In the Environment settings window, click the Save... button.

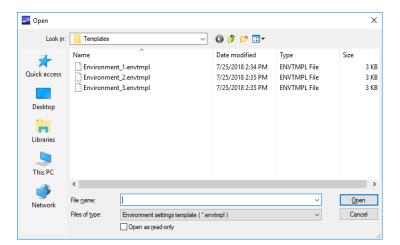


- 5. In the File name field, enter the file name and click the Save button. By default, the file with the environment settings template will be saved to the \Nti\Templates folder with the *.envtmpl extension.
- 6. In the Environment settings window, click the Close button.

NOTE: The exercise start date or time is not saved in the environment settings template.

To load a template in an exercise, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item.
- 2. In the Environment settings window, click the Load... button.
- 3. In the Open window, select the required template.



4. Click the Open button.

NOTE: Select the **Open as read-only** checkbox in the **Open** window if you do not want to make any changes to the environment settings template.

5. In the Environment settings window, click the Close button.

2.7.4 Local Environmental Conditions

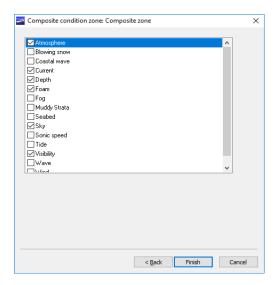
The Instructor can set environmental conditions within a certain zone by using the **Composite condition zone** object belonging to the **Environment** category.

NOTE: Local environmental conditions (determined within zone) are always superseding overall conditions and parameters of the "Tides and Currents" database.

2.7.4.1 Creating Composite Condition Zone

To create a composite condition zone, use the following procedure:

- 1. Choose the Edit\New Object... menu item, or click the toolbar button.
- 2. In the Environment category, select the Composite condition zone object and click the Next button.
- Select parameters which should be added to the composite zone (for more information about parameters, see "Composite Condition Zone Parameters" on page 86).



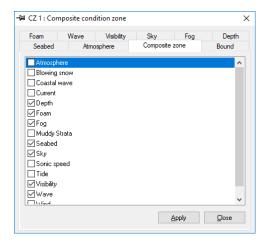
- 4. Click the Finish button.
- 5. Click on the chart to fix the first point of the zone.
- 6. Move the cursor and click on the chart to fix other zone points on the chart.
- Right-click to complete the zone creating. The last zone point will be connected with the first point, and a polygon delimiting the area appears on the chart.
- 8. Position the cursor on the zone boundary. In a second, a prompt containing the zone information appears.



2.7.4.2 Editing Composite Condition Zone

To add/remove environment parameters to/from the composite condition zone:

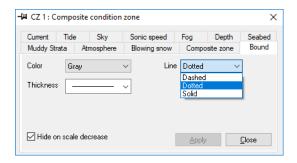
- Right-click the zone boundary and choose the General properties context menu item. The Composite condition zone window appears.
- 2. Open the Composite zone tab.
- 3. Select/clear checkboxes of the parameters you want to add/remove.



4. Click the Apply button.

To change presentation of the zone boundary:

- Right-click the zone boundary and choose the General properties item. The Composite condition zone window appears.
- 2. Open the **Bound** tab of the **Composite condition zone** window.



- 3. From the Color drop-down list, select the boundary line colour. The colour of the zone boundary is visible if the zone is not selected on the chart (inactive).
- 4. From the Line drop-down list, select the bound line type (dashed, dotted, solid).
- 5. From the **Thickness** drop-down list, select the bound line thickness.
- 6. Clear the Hide on scale decrease checkbox if you do not want to hide the zone when the chart scale is decreased.
- 7. Click the Apply button.

To move a zone node, use the following procedure:

- 1. Click the required zone node and move the cursor in the selected direction on the chart. The zone node will be moving together with the cursor.
- Click to fix the zone node in a new position.

To add a zone node, use the following procedure:

- 1. Click the zone boundary where a new node is required to be added. The new zone node appears.
- 2. Move the cursor with a new point to the required place on the chart and click. The new zone node will be fixed on the chart.

To delete a zone point, use the following procedure:

- 1. Choose the Edit\Delete Point menu item, or click the toolbar button, or press the <Ctrl>+ key combination.
- 2. Click the zone point which is required to be deleted.

2.7.4.3 Composite Condition Zone Parameters

The composite condition zone can contain the set of parameters listed in the table below.

Table 6. Parameters of the composite condition zone

PARAMETER	PROPERTIES	DESCRIPTION
Atmosphere	Temperature, pressure, humidity	Properties are set as described above for overall environmental conditions, see "Atmosphere Properties" on page 71
Blowing snow	Intensity	Properties are set as described above for overall environmental conditions, see "Blowing Snow Properties" on page 72
Coastal wave	No properties	Visual effect imitates a surf zone
Current	Depth, velocity, direction	For more information, see "Creating Multilevel Currents" on page 87
Foam	Intensity, colour	For more information, see "Foam Properties" on page 74
Fog	Height, visibility range	For more information, see "Fog Properties" on page 88
Muddy Strata	Thickness, density	For more information, see "Muddy Strata Properties" on page 88
Seabed	Type: rock, mud, sand, gravel	Properties are set in the same way as described above for overall environmental conditions, see "Seabed Properties" on page 75
Depth	Depth	For more information,see "Depth Properties" on page 89
Sky	Clarity	Properties are set as described above for overall environmental conditions, see "Sky Properties" on page 76
Sonic speed	Speed of sound distribution in the water	Properties are set as described above for overall environmental conditions, see "Sonic Speed Properties" on page 76
Tide	Height	Properties are set as described above for overall environmental conditions, see "Tide Properties" on page 77
Visibility	Visibility range, precipitation type (rain, snow, hail), intensity and color (fog or dust)	Properties are set as described above for overall environmental conditions, see "Visibility Properties" on page 78)
Wave	Wind wave: significant height, Beaufort force, direction, spectrum. Swell: height, period, direction.	Properties are set as described above for overall environmental conditions, see "Wave Properties" on page 79

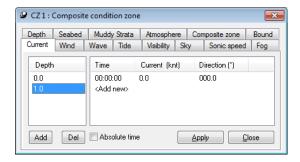
Table 6. Parameters of the composite condition zone (continued)

PARAMETER	PROPERTIES	DESCRIPTION
Wind	Speed, direction, gust, roughness	Properties are set as described above for overall environmental conditions, see "Wind Properties" on page 81

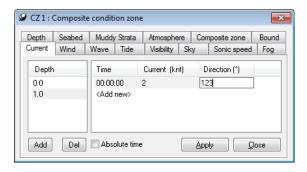
Creating Multilevel Currents

To create multilevel currents within composite zone, use the following procedure:

- 1. In the Composite condition zone window, open the Current tab.
- 2. Click the Add button to add a level of current.
- 3. In the Depth column, enter the depth of the current level.



- 4. In the Current column, enter the current velocity.
- 5. In the Direction column, enter the current direction (deg).



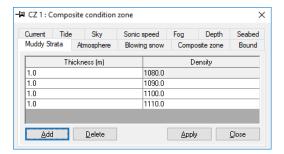
- 6. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 7. To add the change of the current velocity and direction at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.
- Click the Apply and Close buttons.

To delete a level of current, select the necessary depth in the **Depth** column, click the **Del** button and click the **Apply** button.

Muddy Strata Properties

To set the muddy strata properties, use the following procedure:

- 1. In the Composite condition zone window, open the Muddy Strata tab.
- 2. To add a muddy stratum, click the Add button.
- 3. In the **Thickness** column, enter the thickness value (m); and in the **Density** column, enter the density value (kg/cum) of the muddy stratum above the bottom.



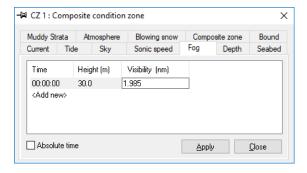
4. Click the Apply button.

To delete a muddy stratum, click its row in the table and click the Delete button.

Fog Properties

To set the fog properties, use the following procedure:

- 1. In the Composite condition zone window, open the Fog tab.
- 2. In the Height field, enter the height of fog propagation.
- 3. In the Visibility field, enter the fog visibility range.

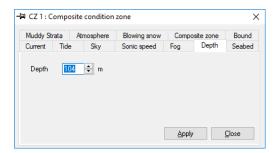


- 4. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 5. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - In the Time column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.
- 5. Click the Apply and Close buttons.

Depth Properties

To set the depth properties, use the following procedure:

- 1. In the Composite condition zone window, open the Depth tab.
- 2. In the **Depth** column, enter the depth value (m).



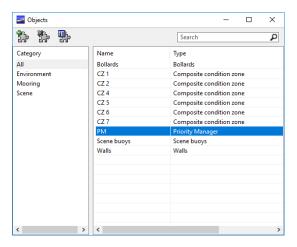
3. Click the Apply button.

2.7.4.4 Priority of Composite Condition Zones

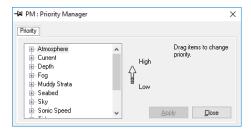
As two or more composite condition zones are overlapping, it is possible to set a priority in the use of parameters in the overlapping area.

To do this, use the following procedure:

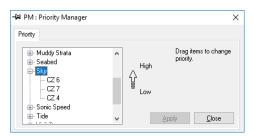
- 1. Choose the View\Panels\Objects... menu item, or click the toolbar button.
- 2. On the Objects panel, select the Environment category in the Category column.
- 3. Select the **Priority Manager** object in the **Name** column.



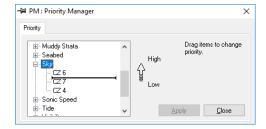
4. Click the button. The **Priority Manager** window appears.



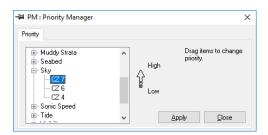
5. Double-click to select the environmental parameter. A list of condition zone names appears.



6. Click and drag the zone name in the list. A horizontal dividing line appears above the zone name. The higher the name of the zone in the list, the higher its priority.



7. Repeat the procedure for other parameters and click the Apply button.



NOTE: When the ship gets into the overlapping area, conditions will be taken from the zone that is the first in the list.

2.7.5 Importing Tides and Currents

To import tides and currents to the exercise, Instructor can use the following databases:

- "Tides and Currents" world database;
- "Total Tide" database of tides and tidal currents;
- · Databases of currents in xml-format.

NOTE: It is not advisable to use different tides and currents databases at the same time.

2.7.5.1 Tides and Currents World Database

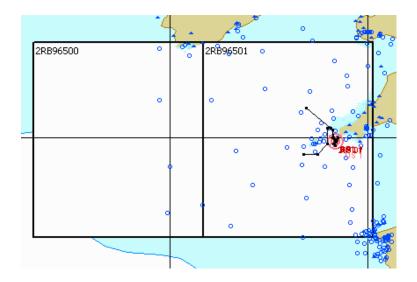
Tide and surface current distribution can be specified in the entire area with the use of "Tides and Currents" world database.

Tide and currents are imported in accordance with: area coordinates, exercise date and time. Tide and current data is updated automatically in the course of exercise.

Actual values of tides and currents in any geographic point of area are calculated by interpolation of three closest points from database.

To import tides and currents to the exercise, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select Tides & currents object from the Environment category.
- 3. Click the Finish button. Tide and current data will be displayed on the chart.



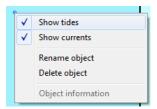
4. Position the cursor on the tide mark on the chart. In a second, a prompt containing the tide height appears.



5. Position the cursor on the current mark on the chart. In a second, a prompt containing the current speed and direction appears.



6. Position the cursor on the current or tide mark on the chart, right-click to call the context menu. In the context menu, select the type of data which you want to be displayed on chart.



2.7.5.2 Total Tide Database

The Instructor can use the "Total Tide" automatically updated database of tides and tidal currents.

NOTE: The "TotalTide" database is available if the licensed "Admiralty TotalTide" application is installed at the Instructor workstation, and the **TotalTide Server** task is run in the configurator (see *Navi-Trainer 6 Professional*. *Technical Description and Installation Manual*. *Version 6.0*).

To import the database:

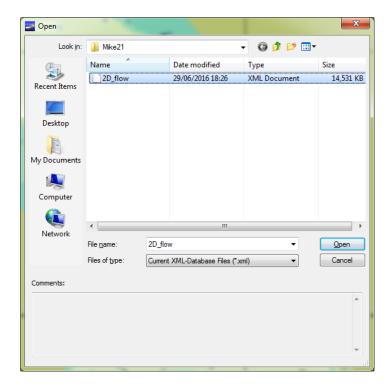
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select the **TotalTide** object from the **Environment** category.
- 3. Click the Finish button.

2.7.5.3 Databases of Currents in Xml Format

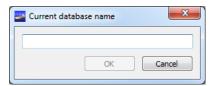
A database of currents in xml-format (MIKE 21 and others) can be imported for certain areas.

To import the database, use the following procedure:

- 1. Choose the Edit\Import Current Database menu item.
- 2. In the Open window, specify the path to the database xml-file and click Open button.

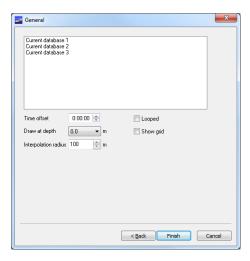


3. In the Current database name window, enter the database name and click the OK button.



To load the database:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- Select the External currents DB object from the Environment category. Click the OK button. The General window appears.



- 3. Select the necessary database in the list box.
- 4. In the Time offset field, set the time offset. The time offset allows using the database at any time of the exercise.

NOTE: For example, the lifetime of the currents database from 16:00 till 18:00, the start time of the exercise is 12.00. After the exercise start the currents will be enabled only in four hours. To activate the currents just after the exercise start, set the time offset from four to six hours.

- 5. In the Draw at depth field, set the depth (the level at which the currents data will be displayed).
- 6. In the Interpolation radius field, set the radius of action of one current point.
- 7. Select the **Looped** checkbox if it is necessary to re-use the current database in the exercise after the database expiration.
- 8. Select the Show grid checkbox if it is necessary to draw a grid of currents vectors on the Instructor's chart.
- 9. Click the Finish button.

2.7.6 Creating Vector Field of Current

The Current field object of the Environment category is used for creating a vector field of current.

There are two ways to set a vector field of current:



– setting the surface current direction and velocity in an isolated point in the space. Several Current field objects will determine a field of current in a certain zone on the exercise chart. The actual value of the current velocity and direction in the ship position will be calculated from the closest points by the interpolation method.



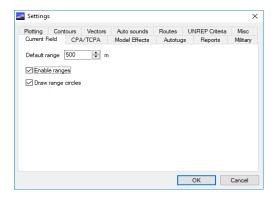
– setting the surface current direction and velocity in a circular area (Range). This method allows models of currents in the river areas or in the narrow straits to be created effectively. In the places where the objects' circular areas intersect each other, data on current will be calculated by the interpolation.

NOTE: The vector field of current can be used in combination with the field of currents loaded from the Tides and Currents database.

2.7.6.1 Current Field General Settings

To set the range of the Current Field objects, use the following procedure:

- 1. Choose the View\Settings... menu item and open the Current Field tab.
- 2. Check the Enable ranges checkbox.
- 3. Set the object default range in the Default range field.
- 4. If it is necessary to hide the range circles on the chart, remove the Draw range circles checkbox.



5. Click the OK button.

To disable ranges, remove the Enable ranges checkbox and click the OK button.

2.7.6.2 Setting Current Field on Chart

To set the Current Field object on the chart, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select Current field object from the Environment category and click the Finish button.
- 3. Click on the chart. The current vector origin will be fixed on the chart.



4. By moving the cursor, set the direction and velocity. The end of the vector will be moving together with the cursor, and the velocity value will be shown near.



- 5. Click on the chart again. The end of the vector will be fixed on the chart.
- 6. Position the cursor on the object's image, a hint specifying the object properties will subsequently appear.



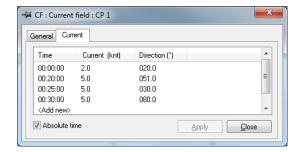
2.7.6.3 Changing Current Field Properties

To change the current field properties:

- 1. Open the Current field object context menu and choose the Properties item. The properties panel appears.
- 2. On the General tab, set the object range.



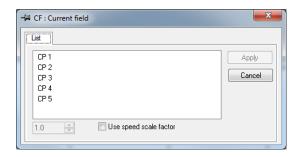
- 3. On the Current tab, set the current velocity and direction values .
- 4. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.
- 5. To add the parameter change at the specified time (within the 24 hours):
 - a. Double-click the <Add new> line. A new row of table appears.
 - b. In the **Time** column, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set parameter values.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete> key.



5. Click the Apply and Close buttons.

To quickly find an object on the chart:

1. Open the context menu of any Current field object and choose the General properties menu item.



2. Double-click the required object in the list. The object will be shown in the centre on a chart, and the object properties panel will be displayed.

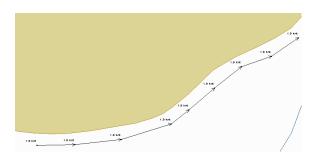
For the current speed scaling-up, select the **Use speed scale factor** checkbox, set the scaling-up coefficient and click the **Apply** button. In this case, the speed of current will be scaling-up for all Current field objects.



2.7.7 Setting Longshore Current

The Stream line object allows the Instructor to set longshore current.

Stream line is a line consisting of one or more segments where each segment determines the current vector (velocity and direction of current).



To set a stream line, use the following procedure:

- Choose the Edit\New object... menu item and select Stream line object from the Environment category. Click the Finish button.
- 2. Click on the chart several times to plot the stream line segments . Right-click to complete the plotting.



NOTE: To add a new point to the stream line, double-click the stream line. To delete a point from the line, click the toolbar button and click this point.

To set stream line properties:

1. Right click the start point of the segment and choose the **Properties** context menu item. The **Point property** tab appears.



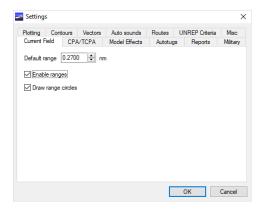
2. In the Speed field, set the stream speed at the start point of the segment (default speed is 1.9 knots).

NOTE: The stream speed in the closest points of segment will be calculated by the interpolation method.

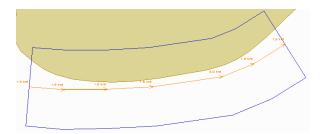
- 3. To apply the "Speed" value to all points on the stream line, select the Apply to all checkbox.
- 4. To set the stream reversed direction, click the Revert button.
- 5. Click the Apply button.

To set properties of the stream tube, use the following procedure:

- 1. Before the exercise start, choose the View\Settings... menu item and open the Current Field tab.
- 2. Select the Enable ranges checkbox. Do not clear the Draw range circles checkbox. Click OK.



The contour of the stream tube appears on the chart.



3. Right click the segment of the stream line and choose the **Properties** context menu item. The **Segment property** tab appears.



4. In the Speed field, set the stream speed on the selected segment of the stream line.

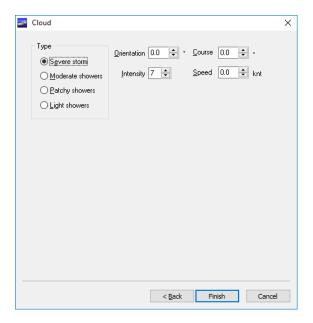
NOTE: The value of the stream speed in all points of this segment is constant.

- 5. To apply the "Speed" value to all stream line segments, select the Apply to all checkbox.
- 6. To set the stream line reversed direction, click the Revert button.
- 7. In the Stream tube field, set the width of a stream tube area (from 50 to 1000 m).
- 8. Click the Apply button.

2.7.8 Setting Clouds

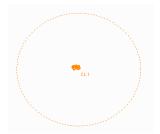
The Cloud object can be seen visually and is shown on the radar. To set the object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select Cloud object in Environment category and click Next.



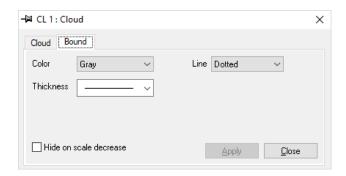
- 3. In the **Type** group box, select the cloud type:
 - Severe storm;
 - Moderate showers;
 - Patchy showers;
 - o Light showers.

- 4. In the Orientation field, set the object orientation.
- 5. In the Intensity field, set the precipitation intensity.
- 6. In the Course field, set the object course.
- 7. In the **Speed** field, set the speed of the cloud motion.
- 8. Click the Finish button and click on the chart.
- 9. By moving the cursor on the screen, set the object area and left click. The object will be fixed on the chart.



To change the object appearance on the chart:

1. Open the object properties panel and select the **Bound** tab.

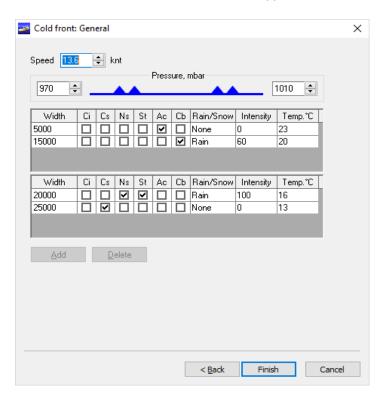


- 2. From the Color drop-down list, select the object boundary colour. The colour of the object boundary is visible if the object is not selected on the chart (inactive).
- 3. From the Line drop-down list, select the boundary line type (dashed, dotted, solid).
- 4. From the Thickness drop-down list, select the boundary line thickness.
- 5. Select the **Hide on scale decrease** checkbox if you want to hide the object boundary when the chart scale is decreased.
- 6. Click the Apply button.

2.7.9 Setting Atmosphere Fronts

To set up atmosphere fronts, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. In the Environment category, select the Cold front or Warm front object.
- Click the Next button. The General window appears.



- 4. In the upper part of the window, set the front parameters:
 - Speed speed of the front;
 - o Pressure atmosphere pressure in extreme points of the front line, mbar.
- 5. In the tables, set the front zones parameters:
 - Width width of the front zone;
 - Types of clouds in the front zone: Ci (cirrus), Cs (cirrostratus), Ns (nimbostratus), St (stratus), Ac (altocumulus),
 Cb (cumulonimbus). Select the required parameters by using checkboxes.
 - Rain/Snow precipitations in the front zone (rain, snow). Select the parameter value from the drop-down list:
 None, Rain, Snow;
 - Intensity intensity of precipitation in the front zone, %;
 - Temp. temperature in the front zone, °C.

NOTE: The upper table is used for setting parameters in zones behind the front line, the bottom table - for setting the parameters in zones before the front line.

- 6. To create a new front zone, click the Add button. In the added table row, set the new zone parameters.
- 7. To delete a front zone, click on the table row with the zone parameters and click the **Delete** button.
- 8. Click the Finish button.

- 9. Set the front line position on the chart:
 - a. Click on the chart to set the first extreme point of the front line.
 - b. Move the cursor and click to set the second extreme point of the front line.

The cold front line is marked with blue and blackened triangles, pointing at the front advance:



The warm front line is marked with red and blackened semicircles, pointing at the front advance:

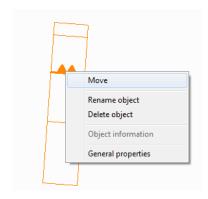


To extend or shorten the front line:

- 1. Click the extreme point of the front line.
- 2. Move the cursor to another point and click.

To move the front line:

1. Right-click the front line and choose the Move context menu item.



2. Move the cursor to the necessary point and click.

To edit properties of the front:

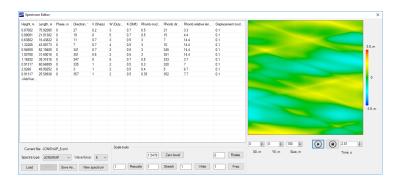
- 1. Choose the **General properties** menu item.
- 2. In the General window, set the properties of the front (see the description above).
- 3. Click the Apply and Close buttons.

NOTE: Atmospheric fronts cannot be created or edited after the exercise start.

2.7.10 Editing Wave Spectrum

The **Spectrum Editor** panel is used for editing wave spectrum parameters.

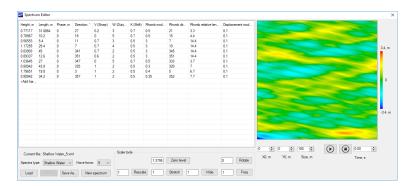
To open the panel, choose the View\Panels\Spectrum Editor... menu item or click the toolbar button.



2.7.10.1 Selecting Wave Spectrum

To select the required wave spectrum:

- 1. Choose the View\Panels\Spectrum Editor... menu item or click the toolbar button.
- 2. From the **Spectra type** drop-down list, select the required spectrum type (three standard wave spectra are available for editing).
- 3. From the Wave force drop-down list select the wave force (0-12).



The table shows values of parameters for each harmonic of a wave spectrum:

- Height wave height: the difference between the elevations of a crest and a neighbouring trough in meters.
- Length wavelength: the distance between consecutive corresponding points of the same phase, such as crests or troughs, in meters.
- Phase phase offset, in meters.
- Direction wave direction: from the north, clockwise, in degrees; 0 head sea, ±90 beam sea, 180 following sea.
- V (Sharp) parameter in the range of [0...1] that modifies wave shape from sine wave (V=1) to triangle wave (V=0).
- W (Duty cycle) parameter in the range of [1...5] that indicates the slope of the wave crest, the higher the value, the sharper the crest of a wave.
- K (Shift) parameter in the range of [0.5...1] that indicates wave crest shift due to the wind and/or shallow water, larger means that wave crest will be shifted more to the forward direction.

- Rhomb modulation parameter in the range of [0...1000] that indicates the wave packet factor. "1000" means that there is no packet factor and "0" means that every N-th wave will have the largest amplitude.
- Rhomb direction- direction from which wave packets are coming. In general, it should coincide with wave direction
 or be very close to it.
- Rhomb relative length parameter indicates, how many wave crests make a wave packet. In general, this should be between 5 and 15.
- **Displacement modulation** parameter in the range of [0...1] does not affect wave shape. It indicates that visualization should draw wave crests sharper and move texture in wave trough faster. In general, it should be close to 0.1.

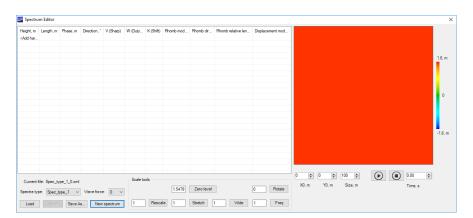
To change the parameter value on the Spectrum Editor panel, double-click the table cell and enter the new value.

To add a new harmonic of wave spectrum, double-click the **<Add harmonic>** row of the table. The new harmonic will contain default parameter values. Enter the new parameter values as required.

2.7.10.2 Creating New Wave Spectrum

To create a new wave spectrum, use the following procedure:

- 1. Choose the View\Panels\Spectrum Editor... menu item or click the toolbar button.
- 2. Click the New spectrum button.
- 3. In the Create newspectrum dialogue box, enter the spectrum name and click OK.



- 4. To add a new harmonic of a wave spectrum, double-click the <Add harmonic> row of the table. The new harmonic will contain default parameter values.
- 5. Enter new parameter values.

NOTE: If values of the parameters are set incorrectly, they will be marked in red. When you hover cursor over these values, the pop-up message appears.

6. To delete a harmonic from the table, right-click on the required row and choose the Delete menu item.

2.7.10.3 Wave Spectrum Saving and Loading

The specified wave spectrum values can be saved in a file in xml-format. To do this:

1. On the Spectrum Editor panel, click the Save As button if it is necessary to save the file under a new name.

NOTE: All the standard spectra can be saved only under a new name if edited.

2. In the Save As window, select a folder, enter the file name and click the Save button.

To save the changes, you can use the Save button on the Spectrum Editor panel.

To load the wave spectrum:

- 1. Click the Load button on the Spectrum Editor panel.
- 2. In the Open window, select the required file and click the Open button.

2.7.10.4 Scaling Wave Spectrum

In the **Scale Tools** group box of the **Spectrum Editor** panel, it is possible to set the scaling for all harmonics of the wave spectrum at the same time.

For scaling the following tools are used:

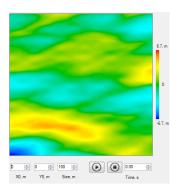
- Zero level shift zero level, in meters;
- · Rescale multiply wave height;
- Stretch multiply of wavelength;
- Wide multiply wave direction range;
- Rotate rotate the spectrum by a specified angle;
- Freq frequency range.

To scale the wave spectrum, enter the required value and click the corresponding button, for example:



2.7.10.5 Wave Heights Map

The right part of the **Spectrum Editor** panel contains a map of wave heights showing an animated example what the selected spectrum looks like:



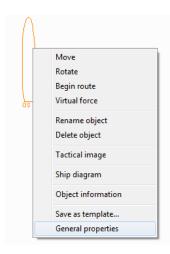
Below the wave heights map, are the following controls:

- X0, Y0 coordinates of the bottom-left corner of the map, in meters.
- Size size of a map, in meters.
- Buttons to control animation of wave heights map:
 - start animation;
 - pause animation;
 - stop animation.
- Time time of animation, in seconds.

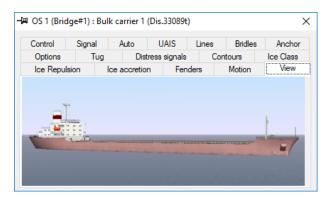
2.8 Setting Ship Properties

The ship properties can be set on the ship's properties panel.

To open the ship properties panel: select the ship on the exercise chart, right click and choose the **General** properties menu item or click the toolbar button.



The panel contains tabs for setting different properties of the own ship or target ship. Each ship model has various set of specified parameters.



The most common set of the own ship properties is as follows:

- Anchor setting anchor properties (see "Setting Anchor Properties" on page 306);
- Auto setting autopilot parameters (see "Autopilot Settings" on page 108);
- Bridles displaying bridle parameters (see "Setting Bridle" on page 310);
- Contours setting display of the ship contours (tracks and trends), display of wind and current diagrams, display of rotation centre and pivot point (see "Contours Settings" on page 116);
- Control control of the own and target (6 DoF) ships during the exercise (see "Control of Own Ship and Target Ship (6-DoF Model)" on page 214);
- Davit setting davit parameters (see "Use of Rescue Boat" on page 394);
- Distress signals giving distress signals (see "Giving Distress Signals" on page 393);
- Fenders setting fender parameters (see "Fenders Settings" on page 117);

- General setting initial ship's speed and course before the exercise start (see "General Ship Properties" on page 107);
- Ice accretion setting ice accretion parameters (see "Setting Ice Accretion" on page 361);
- Ice Class setting ship ice class (see "Setting Ship Ice Class" on page 349);
- Lines setting initial state of the winch before the exercise start and control the mooring winches during the
 exercise (see "Control of Mooring Winches" on page 296);
- Motion displaying current ship data (see "Motion Parameters" on page 221);
- Options setting options (see "Specifying Options" on page 114);
- Position setting ship position (see "Ship Position" on page 107);
- Radar setting radar parameters (see "Radar Settings" on page 109);
- Signal setting navigational signals (see "Navigational Signals Settings" on page 112);
- Tug reserving target tugs for operation with own ship or target ship (see "Tug Reserving and Releasing" on page 331):
- UAIS setting UAIS parameters (see "UAIS Initial Settings" on page 111);
- UNREP Criteria setting underway replenishment criteria (see "Setting Underway Replenishment Criteria" on page 410);
- View ship view;
- · Visual Scripts selecting ship visual script.

2.8.1 General Ship Properties

The main characteristics of the own ship (target ship) can be set on the **General** tab of the object properties panel. To change values of the main parameters, use the following procedure:

- 1. Before the exercise start, open the own ship or target ship properties panel and open the General page.
- 2. Select the recommended ship speed value from the drop-down list, or enter an own value.



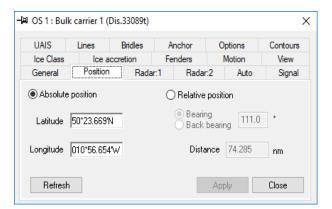
- 3. Set the own ship (target ship) course by using the arrow buttons, or enter an own value.
- 4. For the own ship (target ship) with the variable pitch propeller, check the Combinator checkbox as required.
- 5. For the target ships:
 - a. In the Creation time field, set the object creation time; or in the Start delay field, set the time interval whereupon the object will be created;
 - b. Check the Pitch and roll option as required (for simplified target ship models only);
 - c. Select the 6-DoF model mode as required.
- 6. Click the Apply and Close buttons.

2.8.2 Ship Position

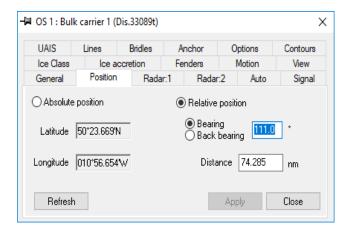
The own ship or target ship position can be set in absolute geographic coordinates or relative the reference point.

To set the object position, use the following procedure:

- 1. Open ship properties panel and select the Position page.
- 2. To set the object position in absolute geographic coordinates:
 - a. Select the Absolute position option.



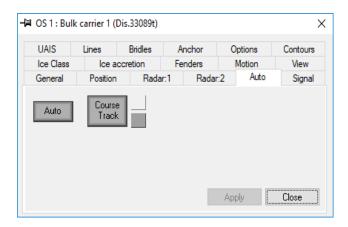
- b. Enter Latitude and Longitude.
- c. Click the Apply button.
- 3. To set the object position relative to the reference point:
 - a. Select the Relative position option.
 - b. Select the Bearing or Back bearing option and enter the bearing value in the field to the right.
 - c. In the Distance field, enter the object distance from the reference point.
 - d. Click the Apply button.
 - e. Click the Refresh button each time to change the reference point position.



2.8.3 Autopilot Settings

To make the own ship autopilot initial settings, use the following procedure:

1. Open the own ship properties panel and select the Auto tab.



- To switch on the autopilot, click the Auto button. By default, the autopilot will be operating in the Course Control mode (the top indicator is lighted).
- To switch over the autopilot to the Track Control mode, click the Course/Track button. In the Track Control mode, the bottom indicator is lighted.
- 4. Click the **Apply** and **Close** buttons. After the exercise has been started, the ship will be controlled by the autopilot in the selected mode.

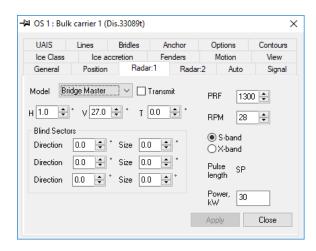
NOTES:

- The own ship autopilot will operate in Track Control mode unless the own ship is assigned to a route.
 The autopilot will operate only at speeds higher than 4 knots.
- The autopilot of a target ship operates in the Track Control mode only. The autopilot is switched on automatically after the exercise has been started if the ship-target has a route and its speed is higher than 0 knots.

2.8.4 Radar Settings

To set initial radar parameters before the simulator session start, use the following procedure:

1. Open the Radar tab of the ship's properties panel.

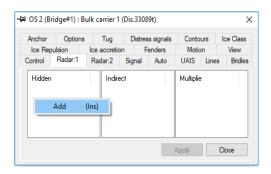


- 2. Select the radar model from the Model drop-down list.
- 3. Select the Transmit checkbox to switch the radar on.
- 4. In the H field, set the horizontal beam width.
- 5. In the V field, set the vertical beam width.
- 6. In the T field, set the antenna tilt.
- 7. In the PRF field, set the pulse repetition frequency (the Pulse length value will be changed depending on the chosen PRF value);
- 8. In the RPM field, set the antenna revolution rate (revolutions per minute);
- 9. In the **Power** field, set the transmitter power, kW;
- 10. Select the S-band or X-band option. The PRF, RPM and Power parameters will accept the default values.
- 11. In the Blind Sectors group box, set the Direction and Size of blind sectors.
- 12. Click the Apply and Close buttons.

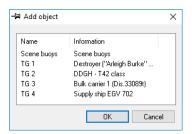
NOTE: The Horizontal beam width, Vertical beam width and Pulse length can also be set by using the RadarImageIni file (see *Navi-Trainer Professional 6. Technical Description and Installation Manual. Version 6.0*).

To set the radar errors during the simulator session, use the following procedure:

- 1. Open the Radar tab of the ship properties panel.
- Add the object to the category of the radar error: Hidden hidden signals; Indirect indirect signals; Multiplie multiple echoes from a large-tonnage ship passing close to the own ship. To do this:
 - a. Right-click the category's column and choose the Add menu item, or press the < Insert> key.

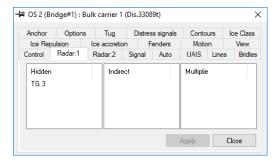


b. Select the required object from the list of the **Add object** dialog box and click **OK**. The **Cancel** button will cancel the selection and close the dialogue box.



NOTE: To delete an object from the category of the radar error, select the required object and choose the **Delete** menu item or press the <**Delete**> key.

3. Click the Apply button.



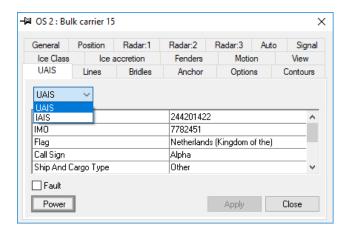
NOTE: The multiple echo effect can also be turned on for all targets by the parameter in the RadarImageIni file. If it is so, this effect will be shown for all targets which are high enough (both target and own ship) and are close enough to the own ship inside the side sector, even though there are no targets in the Multiple column (see Navi-Trainer Professional 6. Technical Description and Installation Manual. Version 6.0).

2.8.5 UAIS Initial Settings

The NTPRO 6 simulator imitates message exchange between the Universal Automatic Identification Systems of own and target ships, and VTS stations. The information required for the Position Report, Static and Voyage Ship data, should be specified preliminarily by the Instructor.

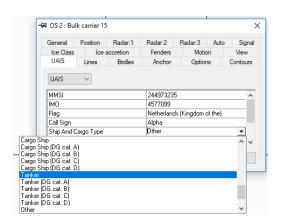
To set the UAIS initial parameters, use the following procedure.

- 1. Open the UAIS tab of the ship properties panel.
- 2. Select the type of AIS-station from the drop-down list: UAIS, IAIS (Inland AIS).



NOTE: After the exercise start the AIS type can be changed for target ships only . The transponder should be switched off.

3. Set the UAIS data in the right column of the table and click the Apply button.



4. To switch on the transponder (the own ship's transponder is turned on by default), click the Power and Apply buttons. After the exercise start, the transponder will be ready for sending and receiving text messages (see "Sending AIS Messages" on page 227).

NOTE: For the target ship's UAIS transponder to be turned on by default, open the **Misc** tab of the **Settings** panel (menu **View\Settings...**) and select the **Turn on UAIS for targets by default** checkbox (see on page 58).

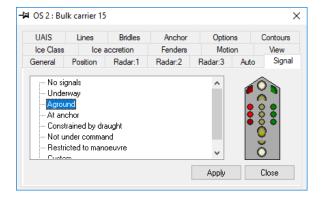
- 5. To set transponder malfunction, select the Fault checkbox and click the Apply button.
- 6. To close the window, click the Close button.

2.8.6 Navigational Signals Settings

Navigational signals (lights, shapes) can be specified for any own and target ship/tug. The specified light combination is switched on automatically at dark time or in low visibility.

All own or target ships are carrying the combination of navigational lights and shapes in accordance with vessel navigational status. To select the vessel navigational status, use the following procedure:

- 1. Open the Signal tab of the ship's properties panel.
- 2. Select a navigational situation from the list (the list of navigational situations may differ depending on the vessel type). The appropriate light will be turned on the mnemonic diagram on the right.

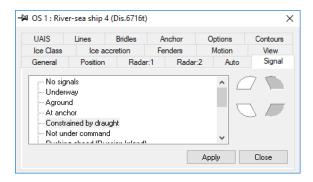


3. Click the Apply and Close buttons.

NOTE: If the Trainee switches on/off other navigational lights, the **Custom** type is highlighted on the **Signal** tab of the ship properties panel.

For river/sea ships:

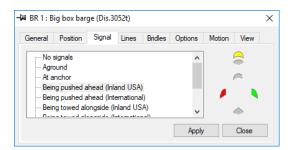
- 1. Open the Signal tab of the ship properties panel.
- 2. Select the navigational situation from the list.
- 3. Switch on/off the passing side indicating signals by clicking four light symbols (in right part of the tab).



4. Click the Apply and Close buttons.

For barges which can be used in a combination tow:

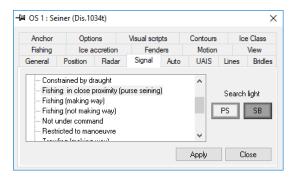
- 1. Open the ship's properties panel and select the Signal tab.
- 2. Select a navigational situation from the list.



- 3. Switch on/off the lights by clicking light symbols (in the right part of the tab).
- 4. Click the Apply and Close buttons.

For fishing vessels, the Instructor can switch on/off the port or starboard searchlight. The searchlight imitates fish attracting process by using bright lights:

- 1. Open the Signal tab of the ship's properties panel.
- 2. Select a navigational situation from the list.
- 3. To switch on the searchlight, click the PS (port) or SB (starboard) button in the right part of the tab. To switch off the searchlight, click the button again.



4. Click the Apply and Close buttons.

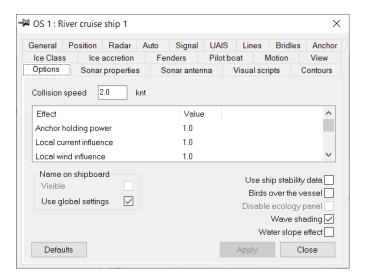
2.8.7 Specifying Options

The options can be set for own ship/tug or target models (6-DoF) before and after the exercise start.

Setting options before the exercise start

To set the object options before the exercise start, use the following procedure:

1. Open the object properties panel and select the **Options** tab.

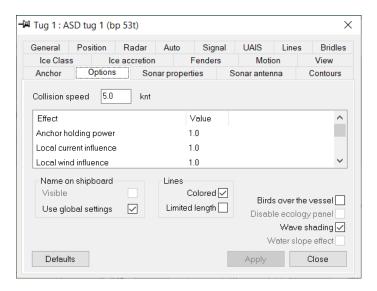


2. In the Collision speed field, set the maximum speed of mechanical interaction with other objects and mooring walls (default speed is 2 knots for own/target ships and 5 knots for own/target tugs).

NOTE: In case of a mechanical contact with another object, the ship is damaged if the ship speed is more than the specified **Collision speed**. The status of the ship will be changed to "Damaged". The corresponding messages appear on the **Event** panel. There will be no ship damaged state if the **Simulate damaged state after collision** option is disabled on the **Misc** tab of the **Settings** panel (see "Miscellaneous Settings" on page 58).

- Set the coefficients of the hydrodynamic interaction and aerodynamic effects (the zero coefficients turn off the calculation of forces and moment of hydrodynamic interaction):
 - Anchor holding power;
 - Local current influence distributed current effect;
 - Local wind influence effect of the wind distributed along the ship hull;
 - Ship to ship interaction hydrodynamic interaction with other ships; can be enabled/disabled on the Model Effects tab of the Settings panel (see "Model Effects Settings" on page 57);
 - Unequal bottom interaction hydrodynamic interaction with unequal bottom (bank). By default, the coefficient is
 0 for the target ship models; the interaction with unequal bottom can be enabled/disabled on the Model Effects
 tab of the Settings panel (see "Model Effects Settings" on page 57).
- 4. Click the **Defaults** button if you want to keep the default values.
- 5. To display the ship's name on the ship side:
 - a. Clear the Use global settings checkbox (if you want to switch off the display of other ships' names).
 - b. Select the Visible checkbox. The ship's name will be displayed on the visualization screen.

- 6. For the tug, target tug objects which have special line's attachment points (winch points), it's possible to set the following parameters in the **Lines** group box (for more information, see "Setting Winch Point Properties" on page 316):
 - a. Select the Colored checkbox to display the coloured towing lines on the visualization screen;
 - b. Select the Limited length checkbox to fix the line length.



7. To monitor the ship stability data, select the Use ship stability data checkbox and click the Apply button.

NOTE: The option is licensed and set only for the own ships. In the Configurations Editor, add the Ship Stability System Interface task. In the task properties, specify the path to the xml-file which contains the following data: draft, trim, heel. For detailed description, see Navi-Trainer Professional 6 (Version 6.0). Technical Description and Installation Manual.

- 8. To display a flock of birds over the ship on the visualization screen, select the Birds over the vessel checkbox.
- To hide the ecology panel on the Conning Display, select the Disable ecology panel checkbox. Indicators of eco parameters are not displayed on the ECO page of the Conning Display.
- 10. To enable the wave shading effect, select the **Wave shading** checkbox. In this case, the ship hull will dump the wave and create the wave shadow behind itself.
- 11. To enable the water slope effect in rivers, select the Water slope effect checkbox. The effect can only be applied to certain areas and ship models.

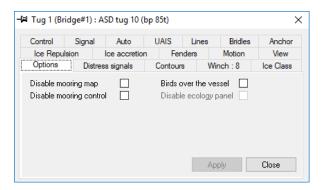
NOTE: For obtaining correct results of using the water slope effect, it is strongly recommended to specify the river currents. To do this, you can import the databases of currents into NTPRO, or use such objects as: Composite condition zone, Current field or Stream line.

12. Click the Apply button to apply all settings.

Setting object options after the exercise start

To set the object options after the exercise start, use the following procedure:

1. Open the own ship properties panel and select the **Options** tab.

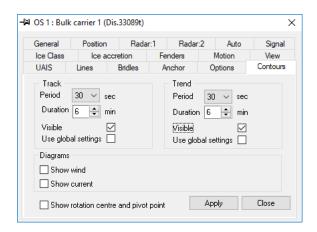


- 2. To hide/show a mooring chart on the **Moor** page of the Conning Display, select/clear the **Disable mooring map** checkbox.
- 3. To hide/show lines on the **Line** control panel on the **Moor** page of the Conning Display, select/clear the **Disable** mooring control checkbox.
- To hide/show flock of birds over the vessel on the visualization screen, select/clear the Birds over the vessel checkbox.
- To hide/show the ecology panel (ECO) on the Instrum page of the Conning Display, select/clear the Disable ecology panel checkbox.
- 6. Click the Apply button to apply all settings.

2.8.8 Contours Settings

To display the contours of the ship track and estimated ship track (trend), use the following procedure:

Open the Contours tab of the ship properties panel.

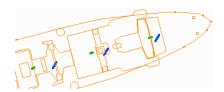


- 2. In the Track/Trend group box, clear the **Use global settings** checkbox to disable the general settings that were specified on the **Contours** tab of the **Settings** panel.
- 3. Set the Track and Trend parameters:
 - Period time interval to display the contours on the chart;
 - Duration track length.
- 4. Select the Visible checkbox to show the contours.

5. Click the Apply button.

To show the curves of the wind and current effect on the ship hull:

- 1. Open the Contours tab of the ship properties panel.
- 2. Select the Show wind checkbox, to show relative wind velocity and course on the chart.
- 3. Select the Show current checkbox, to show relative current velocity and course on the chart.
- 4. Click the **Apply** button. On the exercise chart the following marks will display: blue curve of the wind aerodynamic effect forces, green curve of the current hydrodynamic effect forces on the ship hull.



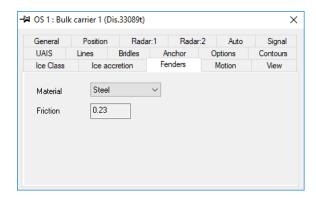
To show the ship's centre of rotation and its projection on the centreline plane (pivot point):

- 1. Open the Contours tab of the ship properties panel.
- 2. Select the Show rotation centre and pivot point checkbox.
- 3. Click the Apply button. During the ship's curvilinear motion, the following points will appear on the chart:
 - RC centre of rotation;
 - PP pivot point.

2.8.9 Fenders Settings

To set the fenders properties, use the following procedure:

- 1. Open the Fenders tab of the ship properties panel.
- 2. Select the material of fenders from the Material drop-down list. The friction coefficient appears in the Friction field.

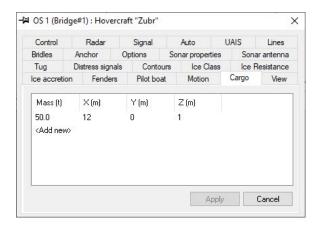


3. Click X to close the window.

2.8.10 Cargo Settings

For some ship models, the Instructor can set properties of cargo before and after the exercise start.

- 1. Open the object properties panel and select the Cargo tab.
- 2. In the Mass column, specify the mass of the load.
- 3. In the X, Y, Z columns, specify the coordinates of the load.

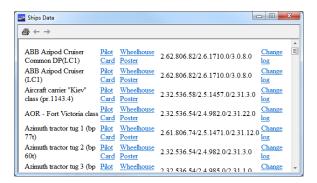


- 4. To add another load, double-click on the <Add new>. A table row appears for setting the properties of the load.
- 5. To delete a load, select the corresponding row in the table and press the <Delete> key.
- 6. Click the Apply button.

2.9 Ship Data

To obtain information on any ship, use the following procedure:

Choose the Help\Ships... menu item. The screen will display the Ships Data window with a list of all the ship
models in the simulator.



- 2. To view the ship Pilot Card, click on the Pilot Card link.
- 3. To view a table of the ship manoeuvring characteristics, click on the Wheelhouse Poster link.
- 4. To view information about all changes made with this ship model, click on the Change log link.

NOTE: If it is necessary to inform the technical support center about the incorrect model performance, in the e-mail message, specify the number of the current version of the ship model (you can see the version number between the Wheelhouse Poster and Change log links).

- 5. To return to the previous page, click
- 6. To turn to the following page, click →.
- 7. To print out a page, click ...
- 8. To close the window, click _____.

2.10 Route Planning

Routes can be plotted for the own ships (tugs), target ships (target tugs), Shoal, Ice zone, Helicopter, Aircraft, Creature, Reference and Transport objects.

Several objects can be assigned to a route.

The route consists of a set of waypoints. The route segment connects two neighbouring waypoints. For some objects, you can set properties of the route waypoints and segments.

The route is initially set graphically on the instructor chart. The route can be subsequently edited in the tabular form. Sailing is along the rhumb line.

2.10.1 Plotting Route on Chart

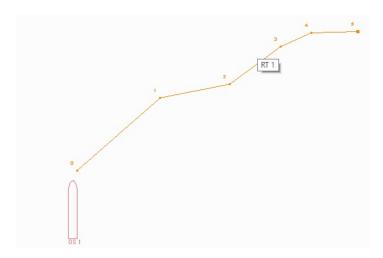
- 1. Route plotting is started in one of the following ways:
 - Right-click the object and choose the Begin route context menu item. The object will be automatically assigned to the route.
 - Choose the Edit\New object menu item. From Scene category, select the Route object and click the Finish button.
- 2. Click on the chart to fix the first waypoint.
- 3. Move the cursor on the chart and click to fix the second waypoint. A segment line connects the first and second waypoints.

NOTE: Press the <+> key or <-> key while moving the cursor to zoom in/out of the chart.

4. Move the cursor on the chart and click to fix other waypoints. The route line connecting the waypoints appears.

NOTE: If the cursor reaches the chart screen boundary during the plotting, after a short delay it will assume the form of a large white arrow. Left-click as many times as necessary to shift the chart fragment the number of steps you wish.

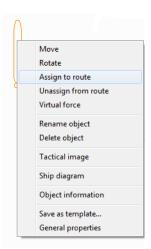
In the terminal point of the route, right-click to complete the route plotting. Numbers of all the waypoints appear on the chart. As the cursor is positioned on any route point or segment, the prompt window shows the name of the route.



2.10.2 Assigning Object to Route

The object can be assigned to any route plotted on the chart. To do this:

1. Right-click the object and choose the Assign to route menu item.

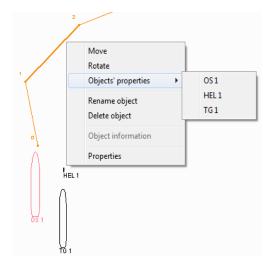


2. Click the route.

NOTE: After the exercise start, the own ship will not proceed along the route unless the autopilot is switched to the Track Control mode. For target ships, the Track Control mode is turned on automatically.

To set properties of objects assigned to the route:

1. Right-click the object and choose the Objects' properties item.



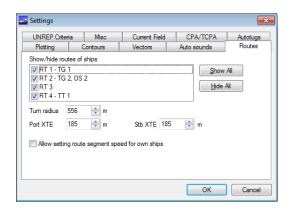
2. Select the necessary object from the sub-menu. The panel with properties of the selected object appears.

To cancel the assigning of the object to the route: rght-click the object and choose the **Unassign from route** item.

2.10.3 General Route Settings

To change the general route settings, use the following procedure:

Choose the View\Settings... menu item and open the Routes tab. The list of routes appears.

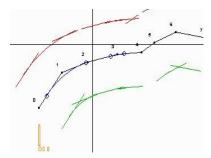


- 2. To show/hide the route on the chart, select/clear the checkbox next to the route name and click OK.
- 3. To show/hide all the routes on the chart, click the Show All/Hide All button and click OK, or click the toolbar button

NOTE: The route will be displayed in either case, if it is selected on the chart, or an object assigned to this route is selected.

- 4. In the Turn radius field, set the ship turn radius and click the OK button.
- 5. In the Port XTE, Stb XTE fields, set a cross track error (for own ships only) and click the OK button.
- 6. To allow setting the own ship speed on the route segments, select the Allow setting route segment speed for own ships option and click the OK button.

If the own ship assigned to the route is selected, the chart will display the turn start and end points (WOP's), turn curve and cross track error lines (red for Port XTE and green for Stb XTE). If due to its maneuvering characteristics, the ship cannot proceed using the set turn radius, the turn curve is not drawn on the chart, whereas in the table this point is marked.



2.10.4 Editing Route on Chart

To shift a route waypoint, use the following procedure:

- 1. Click the route waypoint.
- 2. Move the cursor to a new position on the chart and click again. The waypoint will be fixed in a new position.

To add a new waypoint in the middle of the route, use the following procedure:

- 1. Click the route segment. A new waypoint appears, dividing the selected segment into two parts.
- 2. Move the cursor to the required position on the chart and click. The point will be fixed on the chart.

To add a new waypoint to the end or beginning of the route, use the following procedure:

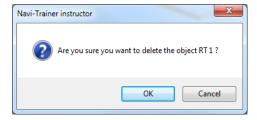
- 1. In main menu, choose the **Edit\Append point** item or click the toolbar button.
- 2. Click the last (first) route waypoint.
- 3. Move the cursor and click on the chart. The new waypoint will be fixed on the chart.
- 4. Right-click to complete adding new waypoints.

To delete a waypoint, use the following procedure:

- 1. Choose the Edit\Delete point menu item or click the toolbar button.
- 2. Click the waypoint you want to delete.
- 3. Right-click to complete deleting waypoints.

To delete a route, use the following procedure:

1. Open the context menu of the route, and choose the **Delete object** item. The screen will display the object deleting confirmation window.



2. Click the **OK** button. The ship route will disappear from the chart.

2.10.5 Setting Route Properties

Route properties allow programming the behavior of an object as it proceeds along the route, and are determined by the properties of waypoints and segments (segment is a line between two waypoints).

Waypoint properties determine the behavior of an object as it approaches this waypoint. Waypoint properties can be set for target ship, target tug, Aircraft, Helicopter, Acoustic transponder objects.

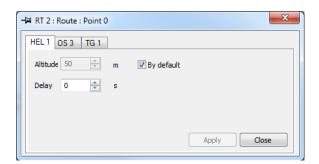
Route segment properties determine the speed of the objects as it is proceeding from one waypoint to the other. Route segment properties can be set for target ship, target tug, Helicopter, Acoustic transponder objects.

NOTE: Also the segment properties of own ships routes can be set if the Allow setting route speed for own ships option is selected in the general route settings (menu View\Settings\Routes).

2.10.5.1 Setting Route Waypoint Properties

To set the route waypoint properties for Aircraft and Helicopter category objects, use the following procedure:

1. Open the route waypoint context menu and choose the **Properties** item. The screen will display a window with the route waypoint properties.



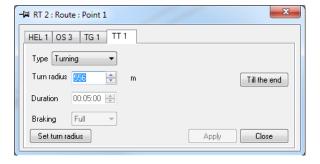
- 2. Select the tab with the name of the necessary object.
- 3. Clear the By default checkbox.
- 4. In the Altitude field set the flight altitude.
- For the Helicopter category object, set the time of staying in a waypoint as required.

NOTE: If the parameter value exceeds the permitted one, the screen will display a prompt specifying the maximum parameter values.

6. Click the Apply and Close buttons.

To set the route waypoint properties for target ship/tug and Acoustic transponder category objects, use the following procedure:

1. Open the route waypoint context menu and choose the **Properties** item. The screen will display a window with the route waypoint properties.



- 2. Select the tab with the name of the necessary target ship.
- 3. Set the waypoint properties (see tables below).

NOTE: If the parameter value exceeds the permitted one, the screen will display a prompt specifying the maximum parameter values.

- 4. For the Acoustic transponder object:
 - To set the depth of immersion, clear the from initial value checkbox and type the required value in the Depth field.
 - b. To set the selected depth of immersion for all the waypoints, click the lower Till the end button.
- 5. Click the Apply and Close buttons.

Table 7. Waypoint properties (simplified target models)

POINT POSITION	POINT TYPE	DESCRIPTION	PARAMETER
Start and intermediate points	Turning	The target continues proceeding along the route	Turn radius
	Stand	The target stops in the waypoint for some time	Duration - stop time
End point	Disappear	The target is deleted from the exercise after passing the route's end point	
	Follow	After passing the end point, the target continues its motion	
	Restart	After passing the end point, the target jumps to the route start	
	Follow back	Shuttle-like motion of the target ship on the route is set	Duration - the target stop time before the U-turn in the route end point

Table 8. Waypoint properties (6-DoF target models)

Table 6. Traypoint proporties (6 Dor taiget modelle)				
POINT TYPE	DESCRIPTION	PARAMETER		
Turning	The target continues its motion making a turn to the next route segment	Turn radius		
Anchorage	The target is going into anchor	Braking - stopping speed during the approach to the waypoint (Dead, Slow, Half, or Full). Duration - time of staying at anchor		
Adrift	The target is heaving-to	Braking - stopping speed (Dead, Slow, Half, or Full). Duration - time of staying adrift		

By default, all the waypoints are set in the Turning mode.

As the Turning type point is selected, the following buttons are activated:

- Till the end to assign the set turn radius value to all waypoint on the route from current point to the end of the route;
- Set turn radius to restore the default turn radius value in the current waypoint.

2.10.5.2 Setting Route Segment Properties

As the object's route is plotted, the same speed equal to the initial object speed is set for each route segment.

It is possible to set desired object's speed on any route segment. It may be used for such objects as a target ship/tug, Helicopter, Acoustic transponder.

Also, you can set the speed of own ship on the route segment if the **Allow setting route speed for own ships** option is selected in the general route settings (see "General Route Settings" on page 122).

To change the speed on a route segment for such objects as target ship/tug, own ship/tug and Acoustic transponder, use the following procedure:

- 1. Open the route segment context menu and choose the **Properties** item.
- 2. Select the tab with the name of the necessary object.
- From the Speed drop-down list, select the object speed on the route segment.



- 4. Click the **Till the end** button as necessary. The object will be proceeding at a speed selected for the route segment to the end of the route.
- 5. Click the **Set ship speed** button as necessary. The object will be proceeding along the segment at the current speed.

NOTE: The speed of own ships on route segments cannot be set after the exercise start.

To change the speed on a route segment for Helicopter object, use the following procedure:

- 1. Open the segment context menu and choose the **Properties** item. The screen will display a dialogue box with the route segment properties.
- 2. Select the tab with the name of the necessary object.

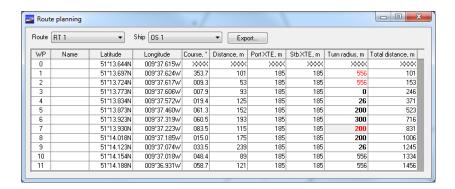


- 3. Clear the By default checkbox.
- 4. Select the object speed on the route segment from the drop-down list.
- 5. Click the Apply and Close buttons. The object will be proceeding along the route segment at the selected speed.

2.10.6 Route Tabular Form

The route tabular form is created automatically after the route has been plotted on the chart.

The Route planning panel (the View\Panels\Route planning... menu item) is intended for editing the route parameter values in the tabular form.



The panel consists of:

- The Route drop-down list for selecting the route plotted on the chart;
- The Ship drop-down list which contains all the ships assigned to this route;
- · Route parameters table.

The route parameters table contains the following information on each route segment:

- WP waypoint number;
- Name waypoint name (optional);
- Latitude and Longitude waypoint coordinates;
- Speed ship speed on the route segment (used for target ships and own ships if the Allow setting route speed for own ships option is selected in the general route settings);
- Course direction to the given point;
- Distance distance between the previous and current waypoint (current route segment);
- Port XTE and Stb XTE cross track error value on the given route segment (used for own ships only);
- · Turn radius turn radius in the waypoint;
- Stay type stay type (optional parameter used for target ships only);
- Stay time stay time (optional parameter, set for target ships only in selecting the stay type: Anchorage, Adrift, Stand, Follow back;
- ETA estimated time of arrival in the given point (used for target ships only);
- Total time summary estimated time of arrival in the given point (used for target ships only);
- Total distance summary distance of route segments to the given point;
- Altitude altitude applies only to the Aircraft and Helicopter objects;
- Depth depth applies only to the Acoustic transponder objects.

In the case of automatic re-calculation of the route segment, the following conditions are fulfilled:

- with the change of coordinates, the distance and direction are re-calculated;
- with the change of distance, point coordinates are re-calculated;
- with the change of direction, point coordinates are re-calculated.

To set and edit the route parameters in the tabular form, use the following procedure:

- 1. Choose the View\Panels\Route planning... menu item, or click the toolbar button.
- 2. From the Routedrop-down list, select the necessary route.
- 3. From the Ship drop-down list, select the necessary ship (target ship) assigned to the route.
- 4. Make a double mouse click in the WP field to select the waypoint number.
- Click the mouse to select a field for editing.
- 6. Enter a new value.
- 7. Press < Enter > or click on any other table row.

NOTE: Incorrect parameter values are marked in red colour. For the ship not to deviate from the route, it is advisable to set other parameter values.

Bold type is used for marking the parameter values different from the standard values set on the Routes tab (menu View\Settings...).

The current waypoint in whose direction the ship is proceeding, is marked with an asterisk.

2.10.7 Conditions for Following on Route

A route has the following limitations:

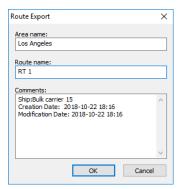
- Critical values of route parameters, indicated in red in the table on the Route planning panel, should be replaced.
- The distance from the first waypoint to the ship should not be more than 0,33 n.m.
- The difference between the ships initial heading and the course on the first route segment should not be more than 30 degrees.
- The ship autopilot is switched on, and the Track control mode is selected.

The ship speed should be 4 knots or more (except if the **Allow setting route speed for own ships** option is selected in the general route settings).

2.10.8 Route Export and Import

To export a route, use the following procedure:

- 1. Choose the View\Panels\Route planning... menu item, or click the toolbar button.
- 2. From the Route and Ship drop-down lists, select the necessary route and the ship assigned to it.
- 3. Click the Export... button. The screen will display the Route planning panel.



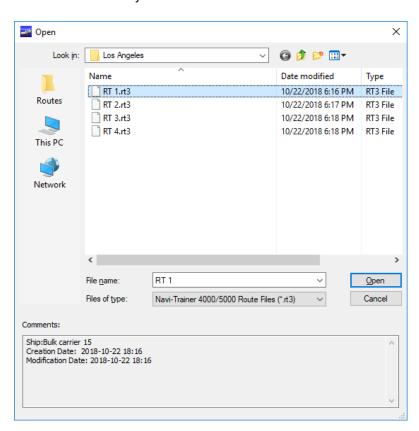
4. In the Area name field, type the name of the area where the route will be saved.

- 5. In the Route name field, type the name of the route file.
- 6. In the Comments field type comments as necessary.
- 7. Click the OK button.

NOTE: All the files with routes are stored in the *Routes* directory and have the *rt3 extension.

To import a route, use the following procedure:

- 1. Choose the Edit\Import Route... menu item or press the <Ctrl>+<Shift>+<R> key combination.
- 2. Open the Routes folder and the folder of the area that contains the file with the route you want to open:
- 3. Click the file name. The bottom part of the window will contain comments:
 - the ship assigned to the route;
 - file creation and modification dates;
 - information added by the Instructor.



4. Click the Open button.

NOTE: Route referencing is made by the geometric position of points on the chart.

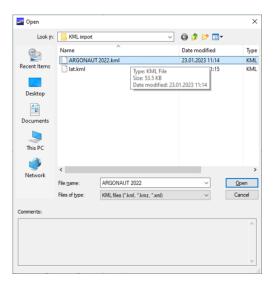
2.10.9 Importing KML or KMZ File

A search and rescue pattern is prepared by the Trainee as a KML/KMZ file. It is used for navigation and coordination of joint search and rescue group operation.

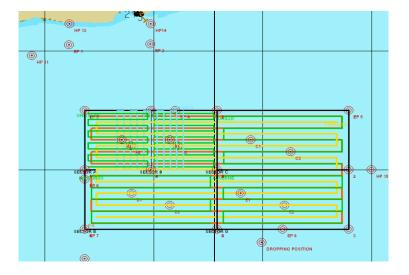
The Instructor can import the available KML/KMZ file into the NTPRO 6 simulator to understand what is happening and to speed up the operation.

To import a KML/KMZ file, use the following procedure:

- 1. Choose the Edit\Import KML/KMZ file ... menu item.
- 2. Open the folder that contains the file with the pattern you want to open and select the necessary file from the list.



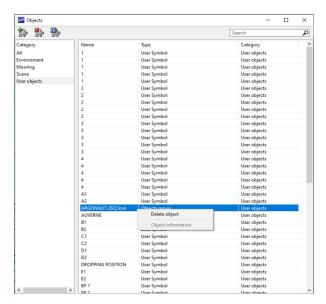
2. Double left-click on the file or click the Open button. The pattern appears on the chart.



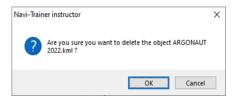
NOTE: You can add the necessary information on the chart and edit the pattern by using the User objects. For more information, see "Adding User Information to Chart" on page 198.

To delete the entire pattern from the chart, use the following procedure:

- 1. Choose the View\Panels\Objects ... menu item and select the User objects category.
- 2. Select the necessary pattern (Objects group) in the list.
- 3. Right-click on the Objects group. A context menu appears.



4. Choose the Delete object menu item. The screen will display the object deleting confirmation window.



5. Click the OK button. The pattern will disappear from the chart.

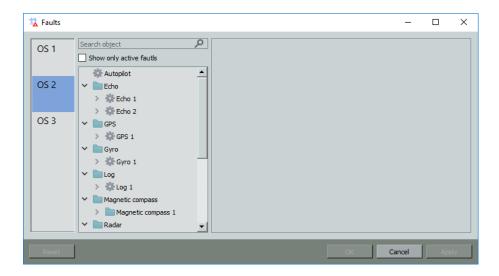
2.11 Faults Panel

All simulator faults can be set by using the integrated **Faults** panel (**View\Panels\Faults...** menu) which makes this process flexible and controllable. The **Faults** panel is also used in the e-Tutor for creating different fault scenarios.

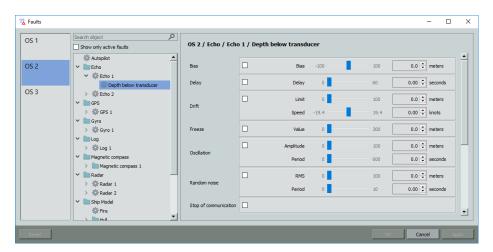
2.11.1 Setting Faults

To set the own ship/tug faults, use the following procedure:

- 1. Choose the View\Panels\Faults menu item or click the toolbar button. The Faults panel appears.
- 2. In the left column of the panel, select own ship/tug from the list of the ships.

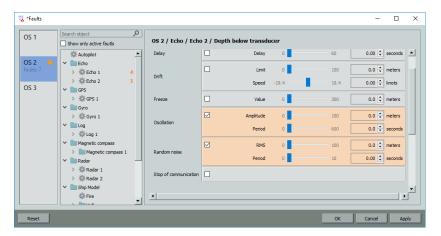


3. In the middle column of the panel, select the equipment category and sub-category (if available) where the fault should be activated. Or enter the name of equipment category (sub-category) in the search bar. The faults list appears in the right column of the panel.



NOTE: The set of faults depends on the ship model.

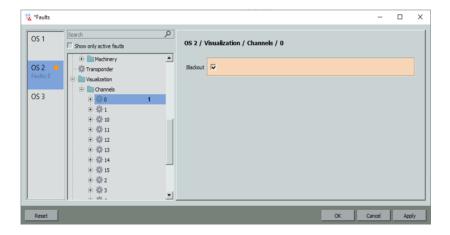
- 4. Set the faults values:
 - o If available, set values in the entry fields or move the slider to the desired position.
 - Select the checkbox next to the fault name to activate the fault. The activated faults are highlighted in the right column of the panel.



Total number of the activated ship faults appears in the left column of the panel under the own ship name.



5. Also, you can activate the fault of a visualization channel – displaying of a black screen. To do this, select the **Blackout** checkbox.



- 6. To show only activated ship faults, select the **Show only active faults** checkbox.
- 7. Click Apply to accept changes and continue editing.
- 8. Click **OK** to accept changes and close the **Faults** panel.
- 9. Click Reset to reset unaccepted changes and continue editing.
- 10. Click Cancel to close the Faults panel without accepting the changes.

2.11.2 Example of Fault Scenario

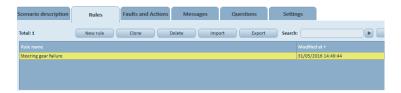
Let's create a scenario with steering gear failure. In the scenario, we set rudder jamming that occurs under the following conditions:

- ship speed of more than 15 knots;
- rudder angle of more than 25 degrees and less than -25 degrees.

NOTE: For more information about e-Tutor product, refer to e-Tutor 5000 manual.

To create a scenario, use the following procedure:

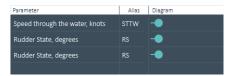
- 1. Create the exercise in the NTPRO Navi-Trainer Instructor application. Add the own ship to the exercise, for example, Cruise Ship Royal Princess.
- 2. Click the Toolbar button to open the e-Tutor Scenario Editor panel.
- 3. On the Rules tab, click the New rule button and enter the rule name, for example, "Steering gear failure".



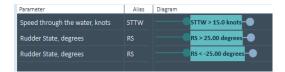
- Click the Add Parameter button.
- 5. In the Select parameter window, add the required parameters:
 - a. Select **Speed through the water** parameter: *root>OS 1>Sensors>Log>1>Speed through the water*.
 - b. Select two identical **Rudder State** parameters one by one: *root>OS* 1>Ship Model>Machinery>Steering>Port>Rudder State.



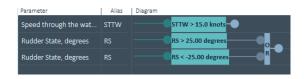
c. Click the Add and close button. The parameters appear in the Parameter list of the rule diagram.



- 6. Select parameter connectors and use the and buttons to add the following comparison blocks:
 - Speed through the water (STTW) > 15 knots;
 - Rudder State (RS) > 25 degrees;
 - Rudder State (RS) < -25 degrees.



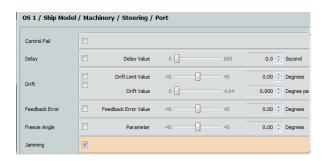
7. Select connectors of the Rudder State parameters and click button. The OR block will be added.



8. Select connector of the **Speed through the water** parameter and connector of the logical **OR** block and click the button. The logical **AND** block will be added.



- 9. Add a fault to the scenario:
 - a. Open the Faults and Actions tab and click the New button. The Faults panel appears.
 - b. In the Faults panel, select the OS1/Faults/Ship Model/ Machinery/Steering/Port category and select the Jamming checkbox. Click OK.



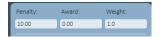
c. If necessary, enter a new name of the fault in the Name field, for example "Rudder jamming".



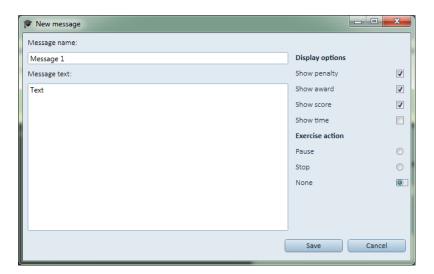
- 10. Open the Rules tab. On the Preconditions and results panel, specify the following settings, for example:
 - a. In the Initiation drop-down list, select the Circular rule triggering type.



b. In the **Penalty** field, enter 10 penalty points. In the **Weight** field, enter the value of 1.0.



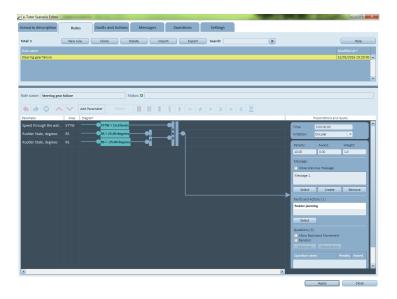
c. In the **Message** group box, click the **Create** button. The **New message** dialog box appears. Enter the message name and text, select **Exercise action** as **None** and click the **Save** button.



d. In the Faults and Actions group box, click the Select button to add the fault you have created. Select the Rudder jamming fault's checkbox and click OK.



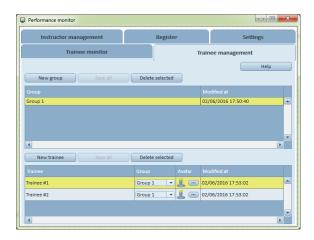
11. Click the Apply button to save the scenario.



12. Click the Close button to close the e-Tutor Scenario Editor panel.

To monitor the exercise, use the following procedure:

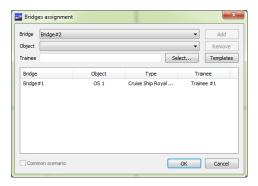
- 1. Click the toolbar button to open the Performance monitor panel.
- 2. Add the group of trainees:
 - a. Open the Trainee management tab.
 - b. Click the New group button and enter the group name. Click the Save all button.
 - c. Click the New trainee button and add the trainee to the group. Click the Save all button.



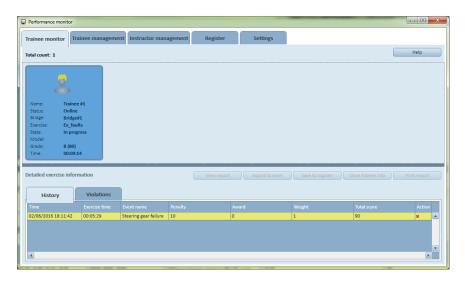
- 3. In the main Instructor window, click the toolbar button to assign the exercise to the bridge. The **Bridges** assignment window appears.
 - a. Select a bridge and object from the **Bridge** and **Object** drop-down lists.
 - b. Click the Select... button. In the e-Tutor Trainees dialog box, select a group and Trainee and click OK.



c. Click the Add and OK buttons.



- 4. Click the toolbar button to start the exercise.
- 5. In the **Performance monitor** panel, open the **Trainee monitor** tab. If the rule has been actuated, the event appears on the **History** tab.



2.12 Setting Custom Alarms

The custom alarms can be created using the Custom alarms panel (menu View\Panels\Custom alarms...) and activated on the own ship using the Faults panel.

NOTE: Also, the custom alarms can be created using the alarm.xml setup file which should be copied from the \Support\GeneralAlarm\alarms.xml folder of the NTI 6 installation package to the simulator \Nti\Data folder. The Instructor can edit the set of alarms contained in the xml-file.

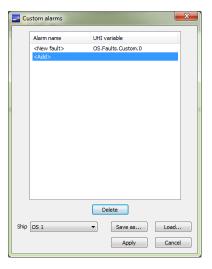
2.12.1 Creating Custom Alarms

To create custom alarms, use the following procedure:

1. Before the simulator session start, choose the View\Panels\Custom alarms... menu item, or click the toolbar button. The Custom alarms panel appears.



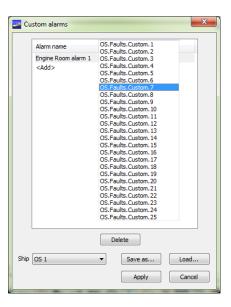
2. Double-click <Add>. The <New fault> table row appears.



3. To set the alarm name, double-click <New fault> field, and type the name of the alarm.



4. To change the UHI-variable value, double-click on the value in the **UHI variable** column, and select a new value from the drop-down list.



- 5. Repeat the previous steps (2-4), to specify other custom alarms.
- 6. To delete an alarm from the list, click it, then on the **Custom alarms** panel, click the **Delete** button or press the <Delete> keyboard key.
- 7. In the Ship drop-down list, select the own ship which custom alarms are specified for.

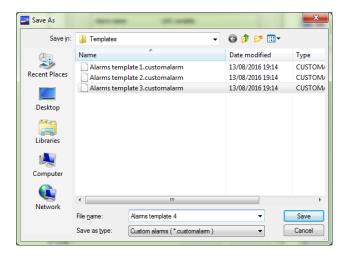


8. Click the Apply button to apply all settings.

2.12.2 Template of Custom Alarms

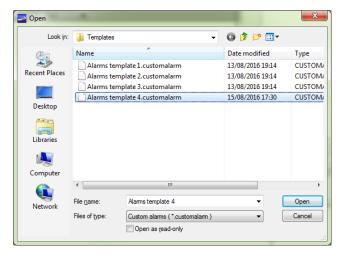
To save the set of custom alarms as a template, use the following procedure:

- 1. Before the simulator session start, open the **Custom alarms** panel (menu **View\Panels\Custom alarms...**) and create custom alarms (see "Creating Custom Alarms" on page 139).
- 2. Click the Save as... button.
- 3. In the Save As window, enter a file name, and click the Save button.



To load the template of custom alarms:

- Before the simulator session start, open the Custom alarms panel (menu View\Panels\Custom alarms...).
- 2. Click the Load... button on the Custom alarms panel.
- 3. Select the template file in the Open window, and click the Open button.



- 4. From the Ship drop-down list of the Custom alarms panel, select the vessel for which the template is defined.
- 5. Click the Apply button to apply the template's settings.

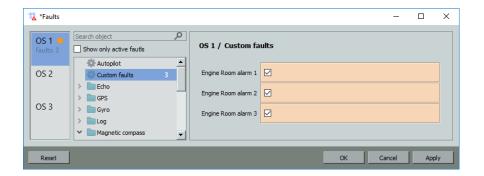
2.12.3 Activating Custom Alarms

To activate the custom alarms, use the following procedure:

- 1. Open the Faults panel (menu View\Panels\Faults and Actions...).
- 2. In the left column of the panel, select an own ship from the list of the ships.
- 3. In the middle column of the panel, select the *OS...\Custom faults* category or enter the "custom faults" in the search bar. The **Custom faults** category appears.

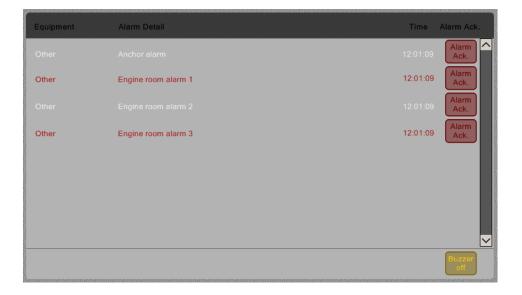


4. Click the **Custom faults** category. In the right column of the panel, select a checkbox next to the custom alarm's name to activate the alarm.



NOTE: To deactivate the custom alarm, clear the checkbox next to the alarm's name.

5. Click the **Apply** button to apply the settings. Click **OK** to apply the settings and close the **Faults** panel. On the Conning Display, the custom alarms are actuated on the **Alarms** page of the **Alarms** panel.



2.13 Setting Synchronization with External GPS **Source**

To set the initial position of a ship in accordance with an external source of GPS data, use the following procedure:

1. Before starting the exercise, choose the View\Panels\GPS Synchronization menu item, or click the toolbar button. The GPS Synchronization panel appears on the screen:





- 2. In the Ship drop-down list, select a ship (target ship).
- 3. In the COM field, enter the number of the COM-port connected to the source of GPS data.
- 4. Click the Set button. The current data from the external GPS source is displayed in the GPS data fields.
- 5. Click the Synchronize button. The position of the ship is set in accordance with the current data received from the external GPS source.

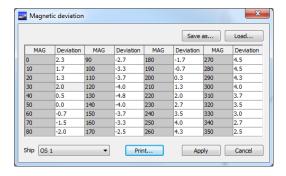
2.14 Magnetic Deviation

Magnetic deviation is the magnetic compass error caused by magnetized iron within the structure of the ship.

2.14.1 Setting Magnetic Deviation

Instructor can set magnetic deviation values for each model of the own ship. To do this, use the following procedure:

1. Choose the View\Panels\Magnetic deviation menu or click the **toolbar button.



- 2. From the Ship drop-down list, select the ship for which you need to change deviation values.
- 3. Enter changes in the deviation table.
- 4. Click the Apply button. The magnetic deviation table will be assigned to the selected ship.

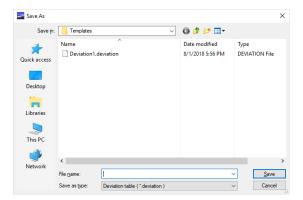
To print diagram and table of the magnetic deviation, use the following procedure:

- Open the Magnetic deviation panel (View\Panels\Magnetic deviation menu).
- 2. Click the Print... button.
- 3. In the Print dialog box, choose the printer name and number of copies and click the OK button.

2.14.2 Magnetic Deviation Template

The Instructor can save the magnetic deviation table as a template. To do this, use the following procedure:

- 1. Open the Magnetic deviation panel (View\Panels\Magnetic deviation menu).
- 2. Set the magnetic deviation values in the table (see "Setting Magnetic Deviation" on page 144).
- 3. Click the Save as... button in the Magnetic deviation window. The Save as window will appear.

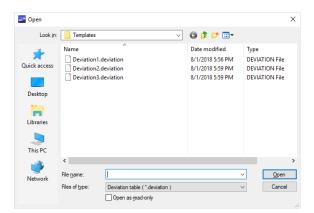


4. In the Save as window set the template name and click the Save button.

By default, the magnetic deviation template is saved to the NTPRO_6/Nti/Templates folder.

To load the magnetic deviation template, make the following steps:

- 1. From the Ship drop-down list of the Magnetic deviation panel, select the vessel for which the template is defined.
- 2. Click the Load... button. The Open window will appear.



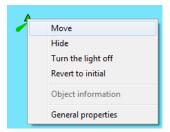
- 3. In the Open window, select the required template and click the Open button.
- 4. Click the **Apply** button on the **Magnetic deviation** panel. The magnetic deviation table will be assigned to the selected ship.

2.15 Aid-to-Navigation Objects (Buoys)

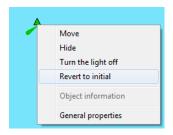
Objects of this type are available in the database for the sailing area, but their position and status can be changed in the course of creating an exercise.

To do this, use the following procedure:

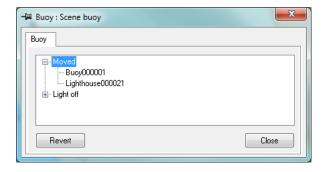
- 1. Open the buoy's context menu and choose the necessary command:
 - Move to move the object to another position on the chart;
 - Hide/Show to hide or show a buoy on the chart;
 - Turn the light on/off to turn on/off the navigational light on the buoy.



2. To return the object to the initial state, choose the **Revert to initial** menu item. All the set object properties will be cancelled.



3. To view the object properties, choose the **Properties** menu item. The **Scene buoy** window will display the set object properties:



4. To cancel a property, select it from the list and click the Revert button.

NOTE: All the operations with buoys are reflected on the radar and visualization screen, but are not passed to Navi-Sailor electronic chart system.

2.16 Land Objects

The simulator features a capability to set various types of 3D objects. The objects can be used as visual landmarks and radar targets.

2.16.1 Setting Land Objects

To set the objects, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select the required object type from the list from the Land object category.
- 3. Click Next. The screen will display the General window for the input of the object's spatial orientation.
- 4. Enter the orientation angle and click **Finish**. The window will close, whereas the cursor will be switched to the mode of setting the object on the exercise chart.
- 5. Click on the chart.

The Land object category contains special objects such as: Tidal stream signal station (7 types) and Traffic control signals station intended for displaying tidal current direction and ship traffic direction.

2.16.2 Tidal Stream Signal Station

To set Tidal stream signal station object on the exercise chart, perform the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- From the Land object category, select the Tidal stream signal station object and click Next. The General window appears.



- 3. In the Orientation field, set the station's orientation on the chart.
- 4. From the drop-down lists select a flashing symbol which will be displayed on Tidal Stream Signal Station: E (east), W (west), digit (from 0 to 13), Up Arrow, Down Arrow.
- 5. Click the Finish button and click on the chart.

NOTE: The displayed symbols can be changed in the course of the exercise on the object properties panel.

2.16.3 Traffic Control Signal Station

To set a Traffic Control Signals Station object on the exercise chart, perform the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Land object category select the Traffic Control Signals Station object and click the Next button. The General window appears:



- 3. In the Orientation field, set the station orientation on the chart.
- 4. Select one of the following flashing symbols, which will be displayed on Traffic Control Signals: H, T, HT.
- 5. To select a symbol, check the checkbox next to the required symbol by clicking the left mouse button.
- 6. Click the Finish button and click on the chart.

NOTE: The displayed symbols can be changed in the course of the exercise on the object properties panel.

2.17 Floating Objects

Floating objects are designed for increasing the display realism during the exercise.

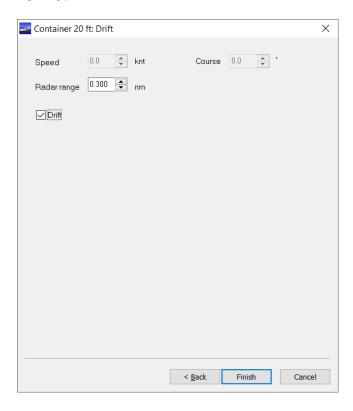
There are following types of floating objects:

- Buoys: Buoy Cardinal, Buoy Isolate Danger, Buoy Lateral, Buoy Preferred Channel To Port\Starboard, Buoy Safe Water, Buoy Special, Small buoy 1, Small buoy 2; Small mooring buoy, Lobster trap buoy, Lobster trap buoy line.
- Above-water navigational dangers: Container 20 ft, Container 40 ft, Floating mine, Trunk, Log vertical, Drug bale, Drug bale zone.

2.17.1 Setting Floating Objects

To set a floating object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- From the Floating object category, select the necessary object and click the Next button. The screen will display the Drift dialogue box for setting the object's parameters. The set of parameters can differ depending on the object type.



NOTE: Setting such floating objects as Lobster trap buoy line, Drug bale zone and Floating mine is described below.

- 3. Leave the **Drift** checkbox selected if you want the object to drift, or clear the **Drift** checkbox and enter the object's course and speed in the **Course** and **Speed** fields.
- 4. For the Container 20 ft, Container 40 ft, Floating mine and Trunk objects, you can set the radar range in the Radar range field.
- 5. Click the Finish button.

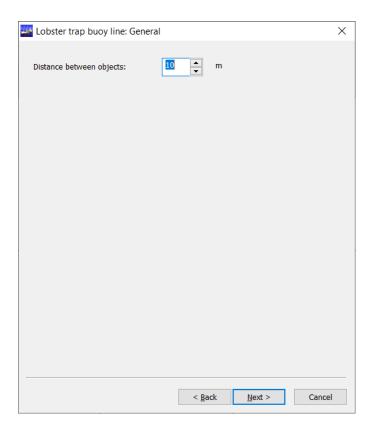
- Or if you create such objects like Buoy Cardinal/Lateral and Small buoy 1, 2: click the Next button, select the visual state of the buoy, and click the Finish button.
- 7. Click on the chart.

NOTE: If the ship collides with a floating object, a relevant entry appears on the Events panel.

2.17.1.1 Setting Lobster Trap Buoy Line

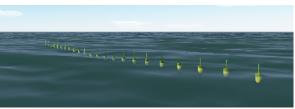
To set a Lobster trap buoy line object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Floating object category, select the Lobster trap buoy line object and click Next.
- In the Distance between objects field, set the distance between neighboring lobster trap buoys (1-100 meters). Click Next.



- 4. Leave the **Drift** checkbox selected if you want the object to drift, or clear the **Drift** checkbox and enter the object's course and speed in the **Course** and **Speed** fields.
- 5. Click the Finish button.
- 6. Left-click on the chart to start the line plotting. Move the cursor and left click. To complete line plotting, right-click. The lobster trap buoy line appears on the chart.

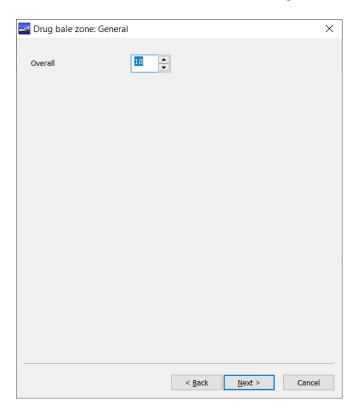




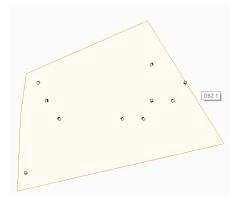
2.17.1.2 Setting Drug Bale Zone

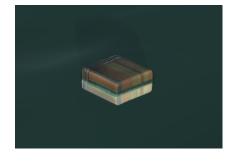
To set a Drug bale zone object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Floating object category, select the Drug bale zone object and click Next.
- 3. In the Overall field, set the set the number of drug bales within a zone (2-100). Click Next.



- 4. Leave the **Drift** checkbox selected if you want the object to drift, or clear the **Drift** checkbox and enter the object's course and speed in the **Course** and **Speed** fields.
- 5. Click the Finish button.
- 6. Click on the chart to fix the first point of the zone.
- 7. Move the cursor and click on the chart to fix other zone points on the chart.
- 8. Right-click to complete the zone creating. The last zone point will be connected with the first point, and a polygon delimiting the area appears on the chart.





2.17.1.3 Setting Floating Mine

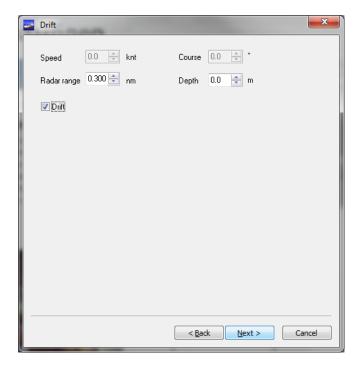
The NTPRO simulates the following operations with the floating mine:

- · explosion of a mine when colliding with a ship;
- explosion of a mine (instant);
- · explosion of a mine by a timer.

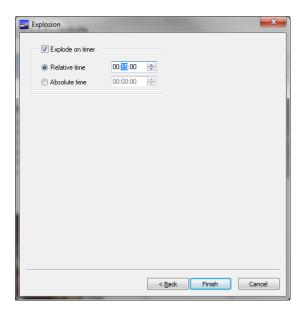


To use a Floating mine object:

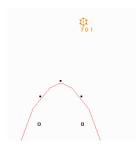
- 1. Set the own ship on the chart.
- 2. Choose the Edit\New object... menu item or click the toolbar button. From the Floating object category, select the Floating mine object. Click the Next button.
- 3. The drift of a mine is selected by default. If necessary, clear the **Drift** checkbox and set the floating mine properties: **Speed**, **Course**, **Radar range**, **Depth**.



- 4. Click the Next button.
- 5. If necessary, set explosion of a mine at a certain time. To do this: select the **Explode on timer** checkbox, select the type of time (**Absolute** or **Relative**) and enter the mine explosion time.



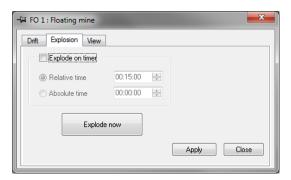
6. Click the Finish button and set the object on the chart.



7. To simulate the pitching of the vessel on waves induced by a mine explosion, select the **Bow waves** checkbox on the **Environment Settings** panel (**Edit\Environment settings...** menu item).

After starting the exercise, you can perform the following actions:

- 1. Open the floating mine properties panel.
- 2. On the Drift tab, set the mine properties: Speed, Course, Radar range, Depth. Or select the Drift checkbox.
- 3. On the **Explosion** tab, click the **Explode now** for instant mine self-destruction. Or select **Explode on timer** checkbox, set time of a mine explosion and click the **Apply** button.



Mine explosion is displayed on the visualization screen (see examples below):

• explosion of a mine when colliding with a ship:



NOTE: Upon collision with a floating mine, there will be no ship destruction if there is no special license.

• mine self-destruction:



2.17.2 Designation of Floating Objects on Chart

Table 9. Floating objects symbols

FLOATING OBJECT	SIMBOL
Buoy Cardinal North_ BI/Y	*
Buoy Cardinal East_ BI/Y/BI	\$
Buoy Cardinal South_ Y/BI	*
Buoy Cardinal West_Y/BI/Y	X
Buoy Isolate Danger	8

Table 9. Floating objects symbols (continued)

FLOATING OBJECT	SIMBOL
Buoy Lateral Green	_
Buoy Lateral Red	
Buoy Preferred Channel To Port	A
Buoy Preferred Channel To Starboard	
Buoy Safe Water	•
Buoy Special	×
Small buoy 1	
Small buoy 2	
Small mooring buoy	0
Lobster trap buoy	⑤
Container 20 ft	
Container 40 ft	
Floating mine	
Trunk	
Drug bale	
Log vertical	\bigcirc

2.18 Creature Objects

The Creature category contains a set of animated objects: Cow, Fireman, Life saver, Policeman, Traffic inspector, Whale, Whale shoal.

All the objects of this category can be seen both, visually and on the radar.

2.18.1 Whales

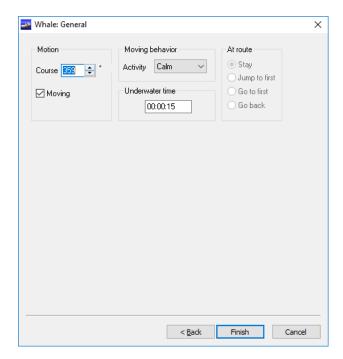
The Whale (single whale) and Whale shoal (five whales) are 3D animated objects that can move along the route, and simulate (visually) whale behavior: breaching, spyhopping, lobtailing, sailing and logging.

Objects are visible on radars, and when the object collides with a vessel, a "crash" event is generated on the **Event** panel.

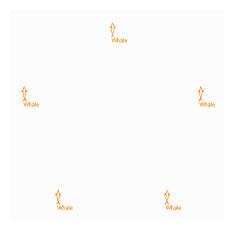


To set the object, use the following procedure:

- 1. Choose the Edit\New Object... menu item, or click the toolbar button.
- 2. Select Whale or Whale shoal object from the Creature category. Click the Next button.
- 3. In the Course field, set the object course.

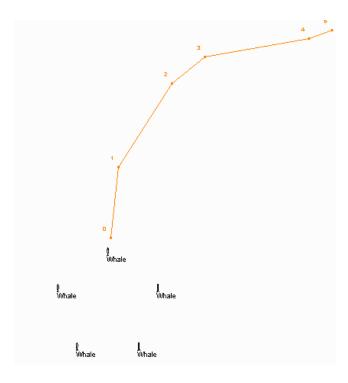


- 4. Set the object motion mode. To do this, select or clear the **Moving** checkbox:
 - III the object is moving at a speed of about four knots. In this case, all five types of behavior (breaching, spyhopping, lobtailing, sailing and logging) are animated.
 - the object is moving in the same place. In this case, only logging type of behavior is animated.
- 5. From the **Activity** drop-down list, select the type of object behavior (if the **Moving** checkbox is selected):
 - o Calm logging type of behavior is animated;
 - Active breaching, spyhopping, lobtailing, sailing and logging types of behavior are animated.
- 6. In the Underwater time field, set the duration of the whales' staying under the water.
- 7. Click the Finish button and set the object on the chart.

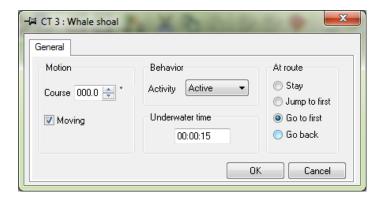


To move the Whale or Whale shoal object along the route:

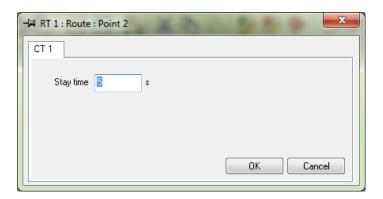
- 1. Right-click on the object and choose the **Begine route** context menu item.
- 2. Plot the route on the chart.



- 3. To set properties in the end waypoint of the route: right-click on the object and choose the **General properties** menu item. In the **At route** group box, select the required option and click **OK**:
 - Stay to stay in the end waypoint;
 - Jump to first to jump to the first waypoint;
 - Go to first to return to the first waypoint by the shortest route;
 - o Go back to go back along the same route.



- 4. To set properties in any waypoint of the route:
 - a. Right-click on any waypoint of the route and choose the **Properties** menu item.
 - b. In the Stay time field, set the duration of stop at this waypoint.



c. Click the OK button.

When the vessel collides with Whale object, the object disappears from the chart and the radar screen. On the Events panel, the following event of the Crash category appears: The ship has contacted with another object "Whale".



2.19 Radar Echo Objects

The objects of this type are intended for increasing the realism of inland radar effects.

The Radar echo objects may be used by the Instructor to flexibly simulate the radar object echo for the trainee scenarios in addition to the existent area radar effects.

There are the following radar echo objects:

- Solid radar object this object provides a simple radar echo effect.
 - The object can be used, for example, in a river confluence to make a ship on a parallel course invisible on the radar, if there are buildings on the ground that are transparent for the radar signal.
 - In that case, the Instructor can place a Solid radar object onto a building on the chart and set the object's height to make the ship invisible.
- · Bridge radar object this object increases the realism of the echo effect produced from a bridge.
 - Generally, a bridge is the most difficult object to display on the radar simulator, and the reason is that the top of the bridge can produce different reflections on the radar image depending on the bridge design.
 - The object can be used, for example, for the bridges having a complicated design. In that case, the Instructor should place it over a bridge. The effect can be customized via the object's panel.
- Voltage line radar object this object creates a specific radar echo effect including a false target which could be
 treated as some navigational obstacle (buoy, boat, etc) and, optionally, a high-voltage line itself which can be set
 visible or not. It simulates the real-life effect produced by high-voltage lines on the inland waterways.
 - The Instructor can use this object to get trainees familiar with the effect while using the simulator. This effect can be customized via the object's panel.

These objects are shown only on the instructor chart and on the radar (if a **Radar\ARPA display** task is run in the configurator), but not shown on the visualization.

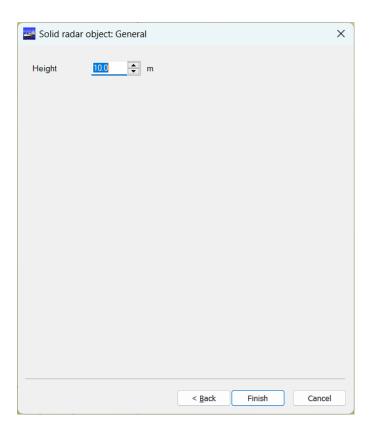
The Instructor can add them any time - before and after the exercise start.

NOTE: The distances at which the Bridge radar echo and Voltage line false target effects appear and disappear from the radar are set by the parameters in the Radar Image Ini file (see Navi-Trainer Professional 6. Technical Description and Installation Manual. Version 6.0).

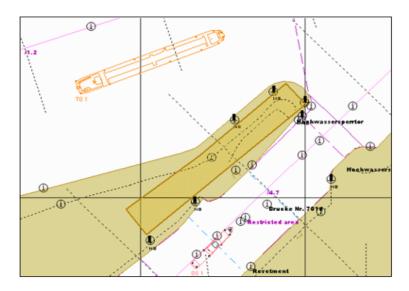
2.19.1 Setting Solid Radar Object

To set a Solid radar object on the exercise chart, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Radar echo category, select the Solid radar object and click Next.
- 3. In the **Height** field, set the object's height on the chart (0-999 meters). The echo height will be calculated relatively according to the lowest point of the area under the object. Click **Next**.

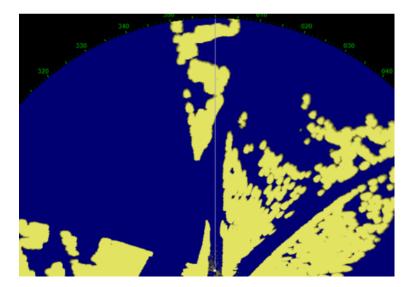


- 4. Click the Finish button.
- 5. Click on the chart to fix the first object point.
- 6. Move the cursor and click on the chart to fix other object points on the chart.
- 7. Right-click to complete the object creating. The last point will be connected with the first point, and a polygon delimiting the area appears on the chart.



NOTE: This picture is given as an example, you may use the object in many other scenarios, not only in a river confluence.

When the own ship moves close enough to the Solid object to be in line with it, this radar echo effect appears. For example, the object can shadow the ship on a parallel course so that it becomes invisible on the radar.



To adjust the properties of a Solid radar object during the exercise, use the following procedure:

- 1. Right click on the solid radar object on the chart and choose the **General properties** context menu item.
- 2. Change the height of this object as required and click the Apply button.

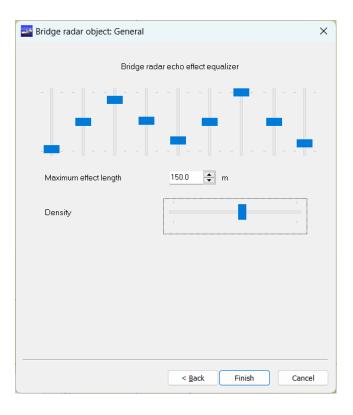


As a result, you can make some object (e.g., a ship on a parallel course behind this solid radar object) invisible or visible on the radar.

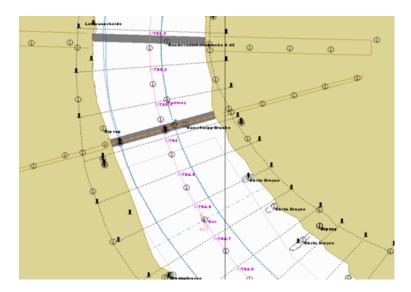
2.19.2 Setting Bridge Radar Object

To set a Bridge radar object on the exercise chart, use the following procedure:

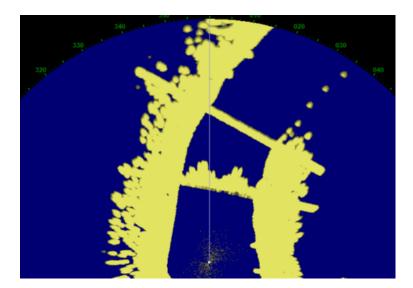
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- From the Radar echo category, select the Bridge radar object and click Next. The Bridge radar object setup window appears with default settings.
- 3. Set the following parameters:
 - Bridge radar echo effect equalizer move the equalizer's sliders to set the form according to which you want
 the effect to be drawn on the radar image. The equalizer controls the length of the stray field along each
 segment of the bridge from the starting point to the end point of the object. The value is from 0.0 to 1.0 times
 the height;
 - o Maximum effect length set the total maximum length for the stray field;
 - Density move the slider left or right to set the number of particles drawn in the effect.



- 4. Click the Finish button.
- 5. Click on the chart on one side of the bridge where it joints with the shore to fix the first object point.
- 6. Move the cursor along the bridge and click on its opposite side to complete the object creating. The last point will be connected with the first point, and a line over the bridge appears on the chart.

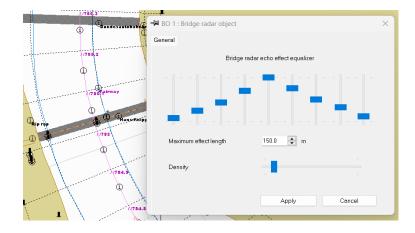


When the own ship moves close enough to the bridge, this radar echo effect appears on the radar.

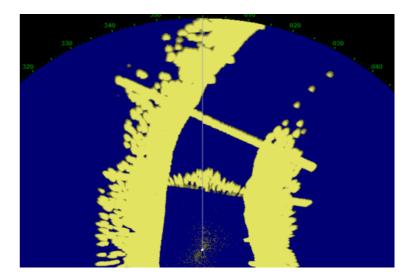


To adjust the properties of a Bridge radar object during the exercise, use the following procedure:

- 1. Right click on the bridge radar object on the chart and choose the **General properties** context menu item.
- 2. Change the required settings. For example, you can set the echo form with only one peak and reduce its density.



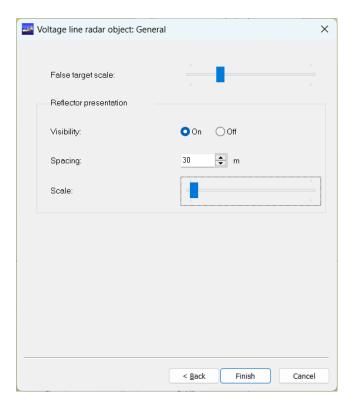
3. Click the Apply button. The radar image will change accordingly.



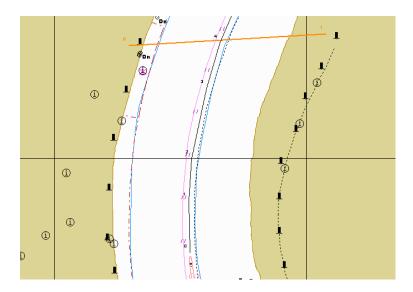
2.19.3 Setting Voltage Line Radar Object

To set a Voltage line radar object on the exercise chart, use the following procedure:

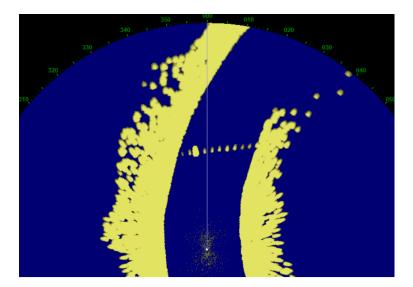
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Radar echo category, select the Voltage line radar object and click Next. The Voltage line radar object setup window appears with default settings.
- 3. Set the following parameters for the false target and plotted line (reflectors) presentation:
 - False target scale move the slider right or left to scale the false target up or down;
 - Visibility select the required checkbox to make the line plotted on the chart visible on the radar or not. By
 default the Off checkbox is selected, it means that the line will not be visible on the radar. To make it visible,
 select On;
 - Spacing set the interval between the line's reflectors (in meters);
 - Scale move the slider right or left to make these reflectors thicker or thinner.



- 4. Click the Finish button.
- 5. Click on the chart on one bank of the waterway to fix the first line point.
- Move the cursor to the opposite bank of the waterway at the angle you want the voltage line to go and click to complete the line creating. The last point will be connected with the first point, and a line over the waterway appears on the chart.



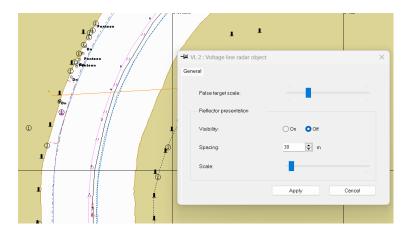
If the reflectors are set visible, the voltage line itself will be displayed on the radar. When the own ship moves close enough to the voltage line, the false target echo effect appears on the radar.



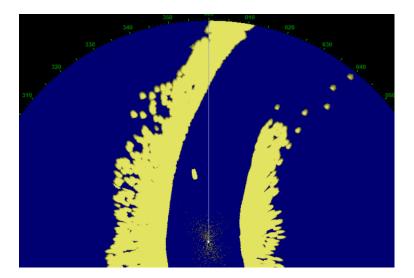
NOTE: This false target appears in place where the radar beam hits the plotted line in a 90 degrees angle. As an own ship moves and the place of the radar beam hit moves, the false target also moves (except the case when the high-voltage line is plotted in a direct line to the own ship's heading).

To adjust the properties of a Voltage line radar object during the exercise, use the following procedure:

- 1. Right click on the voltage line radar object on the chart and choose the **General properties** context menu item.
- 2. Change the required settings. For example, you can set the **Visibility** property to **Off** to make the plotted line (reflectors) invisible on the radar.



3. Click the Apply button. The radar image will change accordingly.



2.20 E-Tutor Objects

The "e-Tutor 5000" evaluation and assessment system provides the Instructor with a set of tools for the creation of simulator scenarios with complex object behaviour and fault occurrences.

NOTE: For more information about the e-Tutor product, refer to Evaluation and Assessment System (e-Tutor) 5000 manual.

The e-Tutor category contains the following objects:

- Alarm zone to monitor violations within a zone;
- Dynamic ruler to monitor the distance between an own ship and another object.
- ERBL to monitor electronic range and bearing line.
- Safety line to monitor the distance from an own ship to the safety line (see on page 167).

2.20.1 Using Safety Lines

To use the safety lines you can perform the following actions:

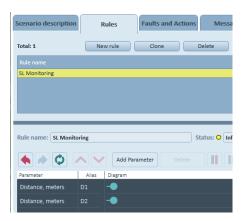
- 1. Create a new exercise and add an own ship to the exercise, for example, Cruise Ship Royal Princess.
- 2. From the e-Tutor category, select the Safety line object, click Finish and plot the line on the chart.
- 3. Repeat the previous step to add the second safety line to the chart.



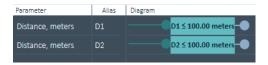
- 4. On the Rules tab, create a new rule:
 - a. Click the New rule button and enter the rule name, for example, "SL Monitoring".
 - b. Click the **Add parameters** button and select **Distance** parameter for SL1 and SL2: *root>OS 1>Geometric zones> SL1 (SL2)>Distance*. Click the **Add and close** button.



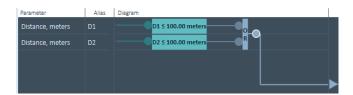
5. If necessary, rename the **Distance** parameters alias in the **Alias** column: D1 and D2.



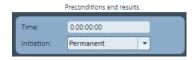
- 6. Select the connector of each parameter and click the button. The comparison blocks will be added:
 - o **Distance D1** ≤ 100 meters;
 - o Distance D2 ≤ 100 meters.



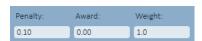
7. Select D1 and D2 connectors and click the button. The **OR** block will be added.



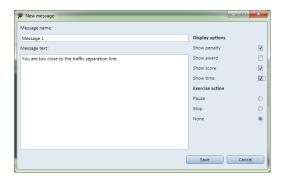
- 8. On the Preconditions and results panel:
 - a. In the Initiation drop-down list, select the Permanent type.



b. In the Penalty and Weight fields, enter the required values.



c. If necessary, in the **Message** group box, click the **Create** button. The **New message** dialog box appears. Enter a message name and text, select **Exercise action** as **None** and click the **Save** button.



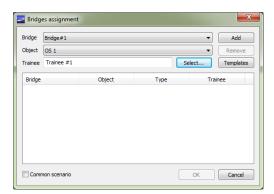
9. Click the Apply button to save the rule.



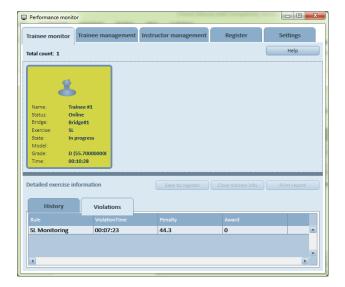
10. Click the Close button to close the e-Tutor Scenario Editor panel.

To monitor the exercise:

- 1. Click the toolbar button to open e-Tutor **Performance monitor** panel.
- 2. Add the group of trainees on the Trainee management tab.
- 3. In the main Instructor window, assign the exercise to the bridge : select bridge, object and Trainee, click the Add and OK button.



- 4. Click the toolbar button to start the exercise.
- 5. In the **Performance monitor** panel, open the **Violation** tab. If the rule has been violated, the violation time and penalty appear in the table.



2.21 AIS AtoN

AIS AtoN is a device or system that is designed and operated to enhance the safe and efficient navigation of vessels and/or vessel traffic.

There are three types of AIS AtoN:

- Real an AIS station located on a real AtoN (Aids to Navigation);
- Synthetic an AIS station located at a distance from a real AtoN and transmitting messages (message 21) about the location and status of AtoN;
- Virtual an AIS station transmitting messages (message 21) about the location and status of a non-existing (virtual) AtoN.

The AIS AtoN object, independently of the selected type, can be placed at any point on the chart or connected to the following exercise objects:

- Small buoy 1, 2 (Floating objects category);
- Oil platform, Oil platform 1, 2 (Mooring category).

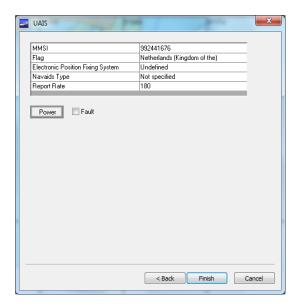
After running a simulator session, all data about the AtoN is displayed on the monitor of the Conning Display MKD imitator and is also displayed on the monitor of the radar. Upon deleting of AtoN Station object from the exercise, the respective data disappears from the monitors of the MKD imitator and the radar in 400 sec.

To set the object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Scene category, select the AtoN Station object.
- 3. In the Name field, type the name of the object (if necessary).
- 4. Click the Next button. The AtoN window appears on the screen:
- 5. In the AtoN AIS station group box, select the type of the object: Real, Synthetic, Virtual.
- If the Real type is selected, set the dimensions of the AtoN Station in the Bow, Stern, Port and Stbd fields of the Dimensions group box.



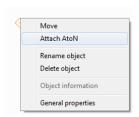
7. Click the **Next** button. The **UAIS** window appears.



- 8. Set the following parameters:
 - o MMSI number;
 - Flag (selected from the drop-down list);
 - Electronic Position Fixing System (selected from the drop-down list);
 - Navaids Type (selected from the drop-down list);
 - o Report Rate.
- 9. If it is necessary, click the Power button to switch on the device (by default, the device is switched on).
- 10. If it is necessary to set a device fault, select the Fault checkbox.
- 11. Click the Finish button and click on the chart.

To attach the AtoN Station object to another object:

- 1. Open the context menu of the AtoN object.
- 2. Choose the Attach AtoN menu item.



3. Click the object to connect it to the AtoN station.

2.22 Smart Targets

Smart targets are the target ships which can automatically perform maneuvers similar to real ships to avoid collisions with other vessels and chart objects during the exercise scenarios.

Smart targets are used for creating the exercises simulating the behavior of vessels in various dangerous situations.

The Instructor can use any 3 DoF or 6 DoF target ship model of the collection as a target ship with smart behaviour (smart target) in any area.

This section describes the collision avoidance maneuvers of vessels in the following dangerous scenarios:

- · with other vessels:
 - Ships assigned on separate routes (see "Maneuvers of Ships Assigned on Separate Routes" on page 179);
 - Ships assigned on the same route (see "Maneuvers of Ships Assigned on Same Route" on page 180);
 - Ships moving on their courses (see "Maneuvers of Ships Moving on Course" on page 182);
 - Ships in high traffic area making cooperative maneuvers (see "Cooperative Maneuvers of Ships in High Traffic Areas" on page 183).
- · with chart objects:
 - Buoys (see "Avoiding Collisions with Buoys" on page 184);
 - No-go zone (see "Avoiding No-Go Zones" on page 186);
 - Depth zone (see "Avoiding Depth Zones" on page 189).

Smart targets can maneuver and avoid the following objects:

- Other vessels (both smart and non-smart);
- Built-in and user-defined buoys;
- Zones in which the navigation of vessels is prohibited for different reasons (i.e. No-go zones);
- Zones where a risk of grounding is detected (i.e. Depth zones).

There may be two variants of maneuvering - course maneuvers and speed maneuvers. For more information, see "Variants of Maneuvering" on page 174.

The decision to make a maneuver is taken based on the following data:

- TCPA if TCPA is less or equal to the TCPA limit, the situation is clarified as dangerous;
- Stability of this dangerous situation it should be stable and not disappear during the analyzed period;
- Distance to the dangerous object.

For any type of maneuver, it is possible to view the maneuver events (see "Monitoring Maneuver Events" on page 178).

By default, the Smart Targets feature is disabled.

You can enable/disable and set Smart Targets in the following ways:

- Globally: for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- Individually: for the selected target ship (see "Setting Smart Targets Individually" on page 177).

NOTE: The individual target ship's settings are prioritized over the global ones.

2.22.1 Variants of Maneuvering

Smart targets can automatically perform the following maneuvers upon a dangerous situation:

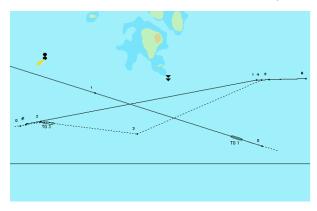
- Course maneuver: in this case, a ship changes its course to make the collision avoidance maneuver (see "Course Maneuvers" on page 174).
- <u>Speed maneuver:</u> in this case, a ship reduces its speed giving a way to another vessel and thus avoiding the collision (see "Speed Maneuvers" on page 175).

The choice of the optimal evasive maneuver is based on the minimum penalty costs and depends on the navigation conditions.

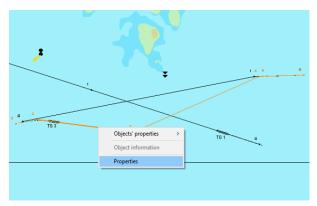
NOTE: The maneuver is assigned to the target ship in the pair based on COLREG 72 rules, if the navigation situation allows this ship to perform it safely, otherwise it is assigned to another ship with a safe trajectory.

2.22.1.1 Course Maneuvers

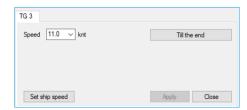
A course maneuver is shown on the chart as a separate object.



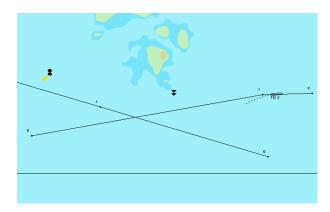
To view the maneuver properties: right-click on the maneuver route, open the context menu and choose the **Properties** item.



You can see the target ship that performs the maneuver, its speed and the maneuver properties: the segment and the final point of the maneuver route (in this case, it is till the end point on the original route).



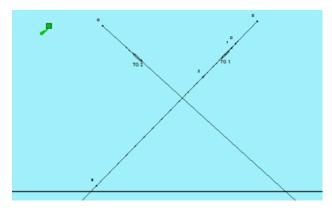
When the maneuver is finished, it disappears from the chart.



2.22.1.2 Speed Maneuvers

In general, a speed maneuver is an optimal variant of the collision avoidance maneuver, if the distance and TCPA at the start of the maneuver allow to avoid the collision by only reducing the speed.

A speed maneuver is not shown on the chart as a separate object, but the additional points appear on the original route, indicating that the vessel's speed will be reduced to give a way for another vessel.



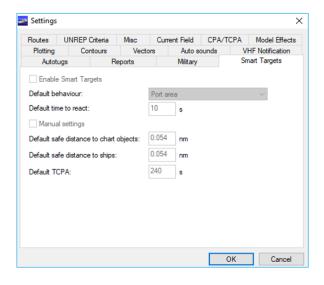
To monitor how the speed is reduced due to the maneuver, open the Ship info panel (see "Ship Information Panel" on page 231). You can see how the maneuvering ships are reducing the speed.

Also, it is possible to monitor the maneuver events (see "Monitoring Maneuver Events" on page 178).

2.22.2 Setting Smart Targets Globally

To enable/disable and set the Smart Targets feature globally, use the following procedure:

- 1. Choose the View\Settings... menu item or click the toolbar button, or press <Ctrl>+<Alt>+<S> key combination.
- 2. Open the Smart Targets tab on the Settings panel.



To enable the Smart Targets feature for all the target ships in the exercise, select the Enable Smart Targets checkbox.

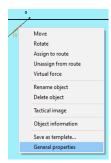
NOTE: To disable the Smart Targets feature globally (for all the target ships in the exercise), clear the **Enable Smart Targets** checkbox and click the **OK** button.

- 4. Set the following parameters (if necessary):
 - Default behaviour behaviour of ships depending on the traffic intensity:
 - Open water the lowest intensity, so that the vessels will maneuver far away from each other;
 - Port area (default value) the middle intensity, so that the vessels will maneuver closer to each other;
 - High traffic area the highest intensity, so that it is only possible to maneuver nearby with each other.
 - Default time to react time for taking the decision whether to wait for the situation to be changed so that the
 danger will disappear, or to make the offered maneuver. If you increase this value, the maneuver will appear on
 the chart earlier.
- 5. To modify the other default settings, select the Manual settings checkbox, and then set the following parameters:
 - o Default safe distance to chart objects minimum safe distance between the vessels and the chart objects;
 - o Default safe distance to ships minimum safe distance to other vessels;
 - Default TCPA TCPA limit for all the vessels at which the situation will be clarified as dangerous.
- 6. To apply the settings, click the **OK** button. These settings will be applied to all 3 DoF and 6 DoF targets in the exercise except the own ship and tugs.

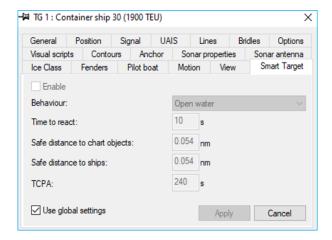
2.22.3 Setting Smart Targets Individually

To enable/disable and set the Smart Target feature only for the selected target ship, use the following procedure:

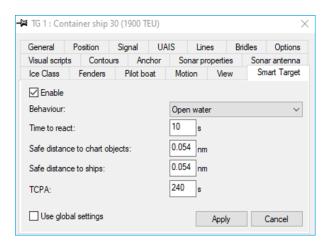
- 1. Choose the target ship on the chart which you want to be a smart or non-smart target.
- 2. Open the context menu of this target ship and choose the General properties item.



3. Open the Smart Target tab on the properties panel.



4. To enable the Smart Targets feature for this target ship, clear the **Use global settings** checkbox, and then select the **Enable** checkbox.



NOTE: To disable the Smart Target feature for the selected target ship, clear the **Enable** checkbox and click the **Apply** button.

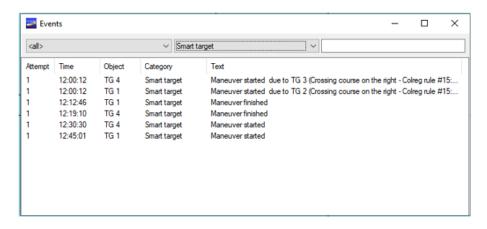
- 5. Set the following parameters (if necessary):
 - o Behaviour behaviour of the ship depending on the traffic intensity:
 - Open water (default value) the lowest intensity, so that the vessel will maneuver far away from other vessels;
 - Port area the middle intensity, so that the vessel will maneuver closer to other vessels
 - High traffic area the highest intensity, so that it is only possible to maneuver nearby with other vessels.
 - Time to react time for taking the decision whether to wait for the situation to be changed so that the danger will
 disappear, or to make the offered maneuver. If you increase this value, the maneuver will appear on the chart
 earlier.
 - Safe distance to chart objects minimum safe distance between the vessel and the chart object;
 - Safe distance to ships minimum safe distance to other vessels;
 - TCPA TCPA limit for this particular vessel at which the situation will be clarified as dangerous.
- 6. To save the settings, click the **Apply** button. These settings will be applied to this particular ship (3 DoF or 6 DoF target).

2.22.4 Monitoring Maneuver Events

Information on the maneuvers performed by smart targets in different dangerous situations is displayed on the **Events** panel.

To view messages, use the following procedure:

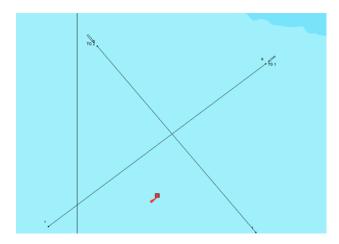
- 1. Choose the View\Panel\Events... menu item. The Events panel appears.
- 2. Select the Smart target category from the right drop-down list.



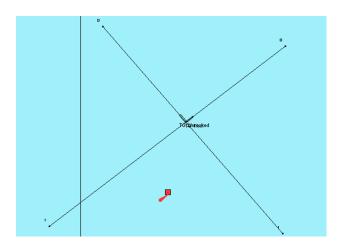
2.22.5 Maneuvers of Ships Assigned on Separate Routes

To create an exercise simulating the collision avoidance maneuvers of target ships assigned on separate routes, use the following procedure:

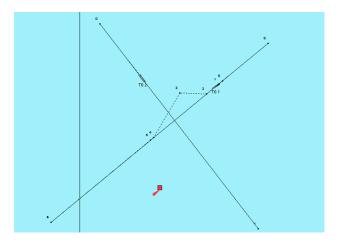
- 1. Select an area (see "Selecting Training Area" on page 28).
- 2. Add an own ship to the exercise (see "Adding Own Ship" on page 42).
- 3. Add 3 DoF and/or 6 DoF target ships to the exercise (see "Adding Target Ship" on page 44).
- 4. Move and rotate the target ships on the chart so that their courses are crossing and there may be a possible collision between them (see "Manipulations with Objects" on page 46).
- 5. Plot the routes for these target ships and then assign each of them on their own route (see "Route Planning" on page 120).



6. Pre-play the exercise to make sure that there will be a collision between these target ships. If not, move them on the routes to the new points and distances between them, and then pre-play the exercise again and again until the collision occurs (see "Pre-Playing Exercise" on page 201).



- 7. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- 8. Pre-play the exercise again. The collision was avoided due to the maneuver made by one of the target ships based on the rules of COLREG 72. In this example, TG1 started the maneuver due to TG2 crossing its course on the right.

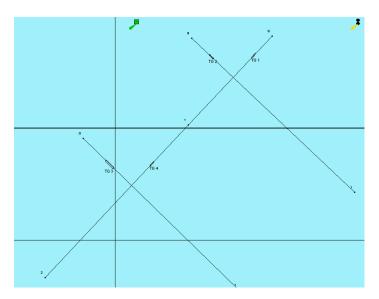


9. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

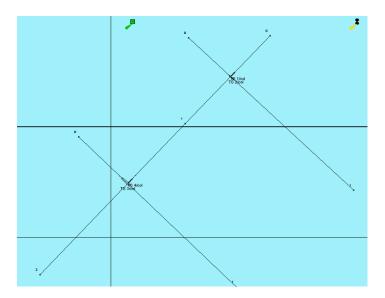
2.22.6 Maneuvers of Ships Assigned on Same Route

To create an exercise simulating the collision avoidance maneuvers of target ships assigned on the same route, use the following procedure:

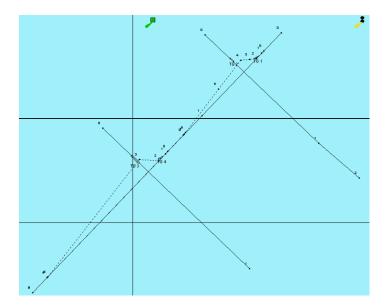
- 1. Follow steps 1-3 of the above exercise (see "Maneuvers of Ships Assigned on Separate Routes" on page 179). In this sample exercise, there will be 4 target ships, with two of them assigned to the same route.
- 2. Move and rotate the target ships on the chart so that there may be collisions with the ships crossing the same route of several target ships (see "Manipulations with Objects" on page 46).
- 3. Plot the routes for the target ships and then assign them on the routes so that several ships (in this sample exercise, there are two ships) should be assigned on the same route (see "Route Planning" on page 120).



4. Pre-play the exercise to make sure that the expected collisions occur. If not, move the ships on the routes to the new points and distances between them, and then pre-play the exercise again and again until the collisions occur (see "Pre-Playing Exercise" on page 201).



- 5. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- 6. Pre-play the exercise again. The collisions were avoided due to the maneuvers made by the appropriate target ships based on the rules of COLREG 72. In this example, TG1 and TG4 started the maneuvers due to TG2 and TG3 crossing their course on the right.



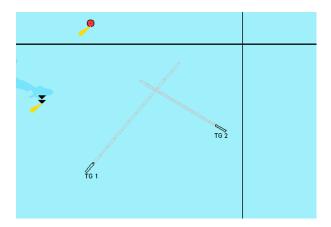
7. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

2.22.7 Maneuvers of Ships Moving on Course

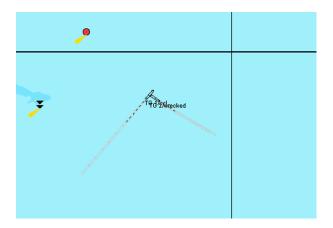
Smart targets that are moving on their course being not assigned to routes also make maneuvers upon a dangerous situation.

To create an exercise simulating the collision avoidance maneuvers of target ships moving on the course, use the following procedure:

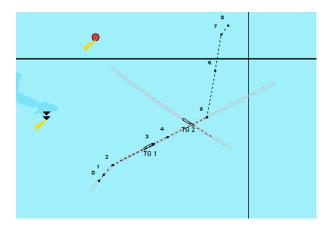
- 1. Select an area (see "Selecting Training Area" on page 28).
- 2. Add an own ship to the exercise (see "Adding Own Ship" on page 42).
- 3. Add 3 DoF and/or 6 DoF target ships to the exercise (see "Adding Target Ship" on page 44).
- 4. Move and rotate the target ships on the chart so that their courses are crossing and there may be a possible collision between them (see "Manipulations with Objects" on page 46).



 Pre-play the exercise to make sure that there will be a collision between these target ships. If not, move them to the new positions, and then pre-play the exercise again and again until the collision occurs (see "Pre-Playing Exercise" on page 201).



- 6. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- Pre-play the exercise again. The collision was avoided due to the maneuver made by one of the target ships based on the rules of COLREG 72. In this example, TG1 started the maneuver due to TG2 crossing its course on the right.



NOTE: The end point of such maneuver will be on the vessel's initial course.

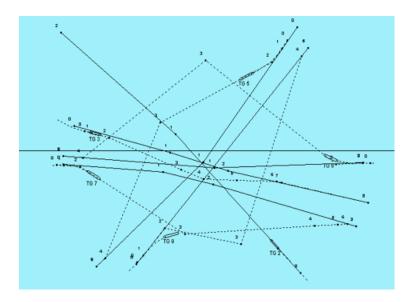
8. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

2.22.8 Cooperative Maneuvers of Ships in High Traffic Areas

Cooperative maneuvers are a set of coordinated maneuvers of a group of ships to optimally avoid dangers. Such maneuvers make it possible to act together in heavy traffic situations and avoid dangerous collisions during maneuvers while other vessels also make maneuvers.

In this case, situations are possible when a vessel with a safe trajectory must perform a maneuver to minimize risks for other vessels. Cooperative maneuvers reduce the number of cases when it is not possible to find an avoidance maneuver.

The example of cooperative maneuvers is shown below:



To create an exercise simulating cooperative maneuvers of ships in a high traffic area, see "Maneuvers of Ships Assigned on Separate Routes" on page 179. Follow the same steps, but add more than two target ships and simulate multiple collisions on step 6.

2.22.9 Avoiding Collisions with Buoys

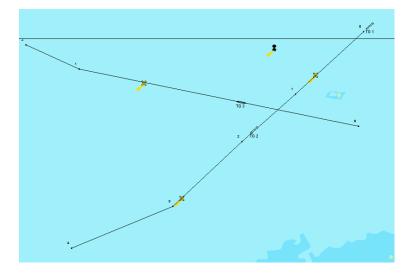
Smart targets are able to make maneuvers to avoid collisions with built-in buoys and user-defined ones that the Instructor may add to the chart during the exercise.

Also, it is possible to hide a built-in buoy during the exercise or move and place it on the route of any smart target to monitor its behavior in different scenarios.

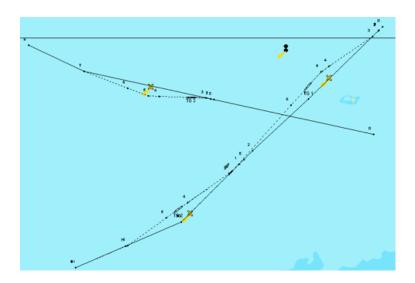
NOTE: If the Smart Targets feature is disabled, target ships move through all the built-in or user-defined buoys not recognizing them as dangerous objects. There are no collisions, but smart targets make this scenario more realistic.

To create an exercise simulating the maneuvers of target ships to avoid collisions with buoys, use the following procedure:

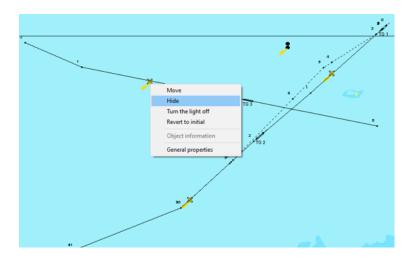
- 1. Select an area (see "Selecting Training Area" on page 28).
- 2. Add an own ship to the exercise (see "Adding Own Ship" on page 42).
- 3. Add 3 DoF and/or 6 DoF target ships to the exercise (see "Adding Target Ship" on page 44).
- Move and rotate the target ships on the chart so that their courses are crossing one or more built-in buoys (see "Manipulations with Objects" on page 46).
- Plot the routes for these target ships and then assign each of them on the same and/or separate routes (see "Route Planning" on page 120).



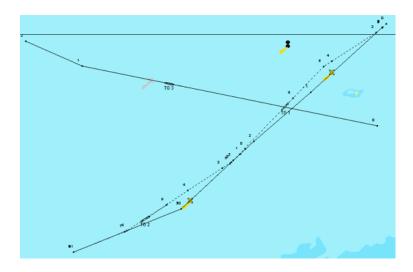
- 6. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- 7. Pre-play the exercise (see "Pre-Playing Exercise" on page 201). You can see the automatic maneuvers made by all the target ships to avoid collisions with built-in buoys.



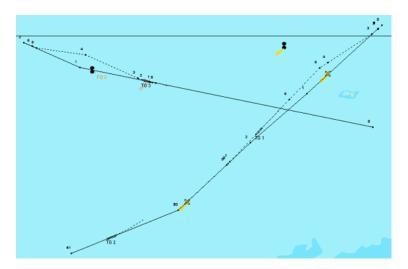
- 8. To check the smart targets' behaviour with hidden buoys, do the following:
 - a. Pre-play the exercise again (if necessary) and choose the buoy on the chart which you want to hide.
 - b. Open its context menu and choose the **Hide** item.



You can see the target ship moving on its original route, without any maneuver. In this sample exercise, it is TG3.



- 9. To add a newly defined buoy to the chart during the exercise, do the following:
 - a. Add a new buoy to the chart as a Floating object (see "Setting Floating Objects" on page 149).
 - Move and place the buoy on the route of the required target ship.
 You can see the maneuver made by the target ship (in this sample exercise, it is TG3) to avoid the collision with the buoy.



10. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

2.22.10 Avoiding No-Go Zones

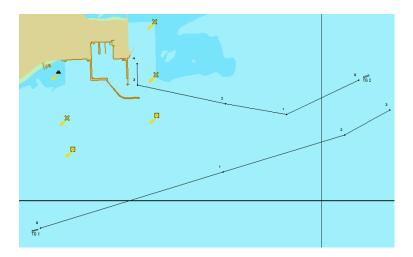
No-go zones are designed for simulating the specific dangerous zones (for example, oil spill areas) that may occur during the navigation and must be avoided by vessels.

The Instructor can define such zones before or during the exercise to monitor the behaviour of smart targets.

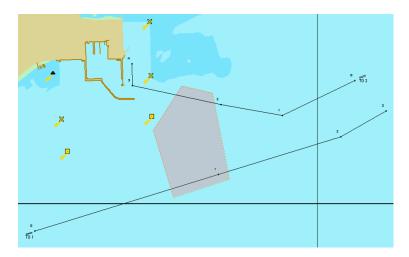
NOTE:If the Smart Targets feature is disabled, target ships move through No-go zones not recognizing them as dangerous objects. There are no collisions, but smart targets make this scenario more realistic.

To create an exercise simulating the maneuvers of target ships to avoid no-go zones, use the following procedure:

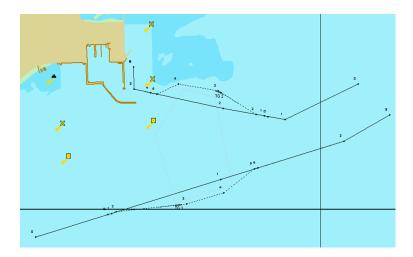
- 1. Select an area (see "Selecting Training Area" on page 28).
- 2. Add an own ship to the exercise (see "Adding Own Ship" on page 42).
- Add 3 DoF and/or 6 DoF target ships to the exercise (see "Adding Target Ship" on page 44).
- 4. Move and rotate the target ships on the chart so that their courses will cross the area where you will define one or more No-go zones (see "Manipulations with Objects" on page 46).
- Plot the routes for these target ships and then assign each of them on the same and/or separate routes (see "Route Planning" on page 120).



- 6. To add a no-go zone to the chart, do the following:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the Scene category, select the No-go zone object and click the Finish button.
 - c. Left click on the chart, plot the required no-go zone so that the routes of the existing target ships will go through it, and then right-click. The last zone point will be connected with the first point, and a polygon delimiting the area appears on the chart.



- 6. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).
- 7. Pre-play the exercise (see "Pre-Playing Exercise" on page 201). You can see the automatic maneuvers made by the target ships to avoid the no-go zone.

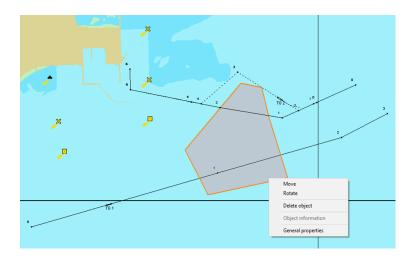


NOTE: If a no-go zone was defined before the exercise start, it is slightly visible on the chart.

- 9. To make a no-go zone clearly visible on the chart, do the following:
 - a. Open the no-go zone's context menu and choose the General properties item.
 - b. Set the required parameters (see "Editing Composite Condition Zone" on page 85).



- 10. To change a no-go zone's contours before or during the exercise, see "Editing Composite Condition Zone" on page N.
- 11. To set a no-go zone object on the chart before or during the exercise, open its context menu and select the required command:



12. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the Smart Target tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

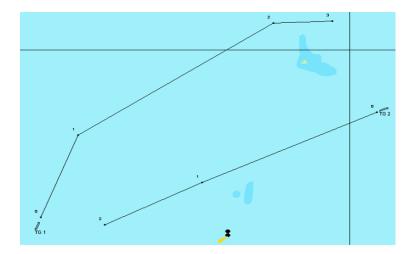
2.22.11 Avoiding Depth Zones

Composite condition zones for defining the Depth (Depth zones) are designed for simulating how vessels can avoid grounding.

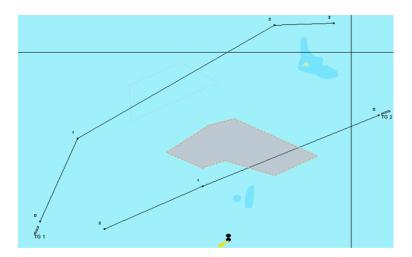
The Instructor can define such zones before or during the exercise to monitor the behaviour of smart targets when there is a risk of grounding in a certain zone.

To create an exercise simulating the maneuvers of target ships to avoid grounding in depth zones, use the following procedure:

- 1. Select an area (see "Selecting Training Area" on page 28).
- 2. Add an own ship to the exercise (see "Adding Own Ship" on page 42).
- 3. Add 3 DoF and/or 6 DoF target ships to the exercise (see "Adding Target Ship" on page 44).
- 4. Move and rotate the target ships on the chart so that their courses will cross the area where you will define one or more Depth zones (see "Manipulations with Objects" on page 46).
- Plot the routes for these target ships and then assign each of them on the same and/or separate routes (see "Route Planning" on page 120).

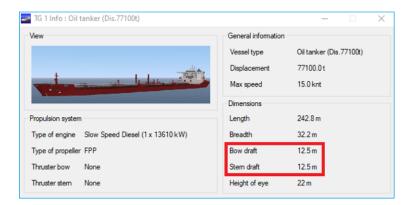


6. Add one or more depth zones to the chart (see "Creating Composite Condition Zone" on page 84).

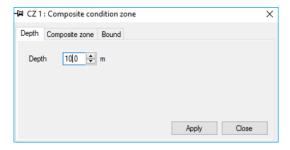


7. View the bow and stern drafts of the vessels participating in the exercise. To do this:

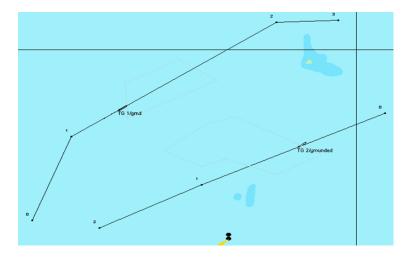
a. Open the target ship's context menu and choose the Object information item. You can see the required values in the Dimensions section.



- b. Remember the maximum between them if they differ.
- c. Repeat this operation for the other target ship (if any).
- 8. Set the depth value for the composite condition zone(s) less than the draft of the related target(s). For example, if the maximum draft of a target ship is 12.5 m, then you should set the Depth properties equal to 10 m. To do this:
 - a. Open the depth zone's context menu and choose the General properties item.
 - b. Set the required value and click the Apply button.

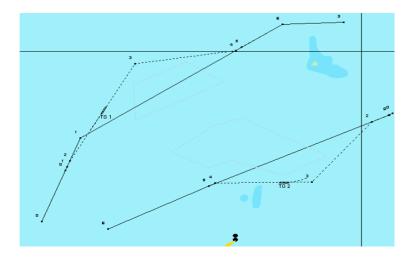


9. Pre-play the exercise to make sure that the expected groundings occur (see "Pre-Playing Exercise" on page 201).



10. Enable the Smart Targets feature globally for all the target ships in the exercise (see "Setting Smart Targets Globally" on page 176).

11. Pre-play the exercise again. You can see the automatic maneuvers made by the target ships to avoid the depth zones.



12. To disable the Smart Target feature for a particular ship which you don't want to perform the maneuvers, use the **Smart Target** tab of the target ship properties panel (see "Setting Smart Targets Individually" on page 177).

2.23 Using Objects Templates

Objects templates are used for placing an object or a group of objects with the set properties in any place on the chart or in another training area.

2.23.1 Template of Single Object

Instructor can create a single object template and then use it in different exercises.

NOTE: This option is available for objects in the following categories: Ship, Ship target, Tug, Tug target, Anchor handling, Barge, Creature, Fishing, Floating Object, Helicopter, Land object, Transport.

To create a template of a single object, use the following procedure:

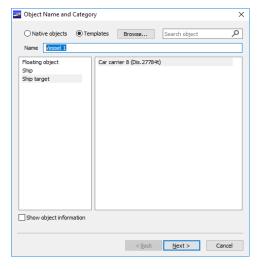
 Open the object context menu and choose the Save as template... item. The Save As Template window appears.



- 2. Enter the name of template in the Template name field.
- 3. To change the folder for saving the template, click the Change... button.
- Click the OK button. The template will be created and saved by default to the \NTPRO_6\Nti\Templates folder.

To set the object template in the selected place, use the following procedure:

- 1. Choose the Edit\New Object... menu item or click the toolbar button. The Object Name and Category window appears.
- 2. Select the **Templates** option button.
- 3. In the left-hand part of the window, select an object category.
- 4. In the right-hand part of the window, select an object template.
- 5. In the **Name** field, enter the new object name (if the name is entered incorrectly, it is highlighted in red and the further object setting will be impossible).

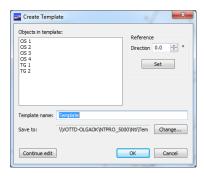


- 6. Click the Next button.
- 7. In the **General** window, set the object's initial parameters if necessary.
- 8. Click the Finish button and click on the chart.

2.23.2 Template of Group of Objects

To create a template of a group of objects, use the following procedure:

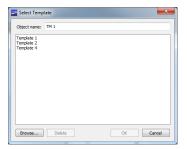
- 1. Choose the Edit\Create Template menu item. The template symbol appears next to the cursor on the screen.
- 2. Click the objects which should be added to the template.
- 3. Right-click to complete the adding of objects to the template. The Create Template dialogue box appears:



- 4. In the **Direction** field, set the object group direction. Or click the **Set** button, and set the direction on a chart with the cursor.
- 5. In the **Template name** field, enter the template name.
- 6. To change the folder for saving the template, click the Change... button and select the required folder...
- 7. If it necessary to continue editing the template (to delete or add objects), click the **Continue edit** button. The template symbol appears next to the cursor on the screen. Then repeat steps 2,3 of this procedure.
- 8. Click the **OK** button. The template will be created and saved, by default, to the *NTPRO_6\Nti\Templates* folder.

To insert the template in the selected place on the chart, use the following procedure:

1. Choose the **Edit\Insert Template...** menu item. The screen will display the **Select Template** window with a list of available templates.



NOTE: To delete a template from the list, click it and click the **Delete** button.

2. Select a template from the list or click the Browse... button to select a template file from another folder.

- 3. Enter a template name and click the **OK** button.
- 4. In the General window, set the template parameters:
 - Speed speed of the template's objects;
 - o Orientation object orientation relative to the template setting point.



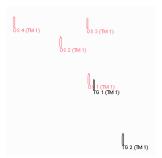
5. Click the **Finish** button. The square shape of the template will be shown on the screen:



6. By moving the cursor, select the template setting point on the chart and click the left mouse button. The template will be set in the selected point.



7. To ungroup the template, position the cursor on the template boundary and choose the **Ungroup** context menu item. Template objects will be ungrouped. The template name will be added to the name of each object.

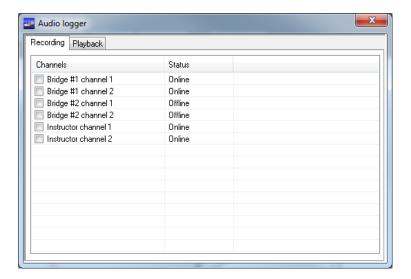


2.24 Setup of Exchange Recording

Recording of trainee exchange during an exercise is carried out by the Audio Logger module which must be added to the configuration of the Instructor workplace or bridge. The Audio Logger accepts data from the sound card (with a microphone connected or handset) and generates audio log-files. Logging process can be controlled by the Instructor from the **Audio logger** panel.

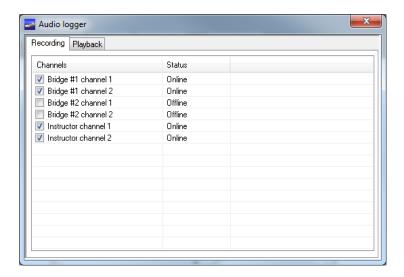
To switch on the recording of exchange, use the following procedure:

- 1. Choose the View\Panels\Audio Logger... menu item, or click the toolbar button.
- 2. On the Audio logger panel, open the Recording tab.



NOTE: The **Channel** column contains full list of audio channels. The **Status** column contains audio channel status. The channel available for recording will have the **Online** status.

3. Select channels which should be recorded.



NOTE: For the exchange playback, see "Playback of Audio Log Files" on page 278.

2.25 Setup of Video Log File Recording

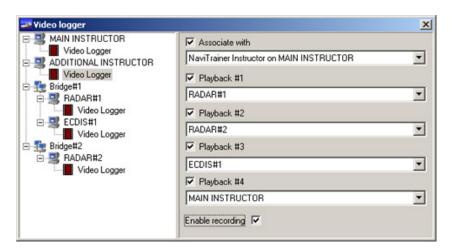
Video log files are recorded in the course of an exercise by the VideoLogger module included in the instructor workplace or bridge configuration. The use of the VideoLogger software module is recommended for logging instructor input at his/her instructor workstation and for recording the trainee performance on the Radar/ARPA, NavAids, Fishing station, and ECDIS computers. The recording is controlled from the instructor workplace on the Video logger panel.

NOTE: It is not advisable to use the VideoLogger software module on the computers where Navi-Trainer Model and Visual Channel tasks are run.

Visual Channel and Conning Display tasks are played back during the debriefing from the main log file data, so these require no video log files.

To turn on the recording of video files, perform the following procedure:

- 1. Choose the View\Panels\Video logger... menu item, or click the toolbar button. The Video logger panel contains a full list of Video Logger tasks as per the current simulator configuration.
- 2. In the left-hand part of **Video logger** panel, select, one by one, Video Logger tasks required for the future recording, selecting the **Enable recording** checkbox in its right-hand part.



NOTE: The selecting of the **Enable recording** checkbox in the course of the exercise will not result in the recording of video log files.

At the moments when video recording is started and stopped, screens of the pertinent computers will black out for a second, which is an indirect indication of Video Logger task response.

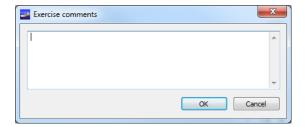
To play back the log file recorded during the exercise, see "Playback of Video Log Files" on page 279.

2.26 Creating Comments on Exercise

At the exercise preparation stage, the Instructor can add own comments which will subsequently be shown in the **Open** window when this exercise or log file is opened.

To create and edit comments on an exercise, use the following procedure:

1. Choose the Edit\Comments... menu item or click the toolbar button. The Exercise comments window appears.



2. Enter comments and click the OK button.

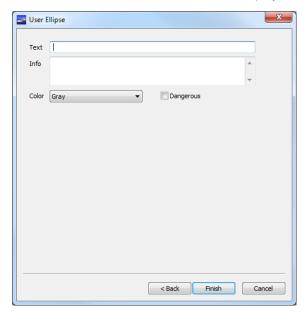
NOTE: Comments can also be added or edited in the Comments field of the Save As dialogue box.

2.27 Adding User Information to Chart

The Instructor can add the necessary information to the exercise chart by using the User objects.

To add instructor information to the chart, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. In the User objects category, select the necessary object from the list:
 - User Ellipse to show an ellipse on the chart;
 - User Line to show a line on the chart;
 - User Symbol to show a symbol on the chart;
 - User Text to show a text on the chart;
 - User Zone to show a zone on the chart.
- 3. In the Name field, enter the object name if necessary.
- 4. Click the Next button. The screen will display a window for the input of the object properties.



- 5. Set the object properties:
 - Text the text will be displayed on the chart and/or in the pop-up window, which appears when the cursor is positioned on the object border;
 - Info additional information on the object;
 - o Color colour of the text (symbol, line, ellipse, zone);
 - Dangerous to make a note in the Chart Info window that the object is dangerous.
- 6. Click the Finish button.
- 7. To set the User Symbol or User Text object, click the required point on the chart.
- 8. To set the User Line object:
 - a. Click the line's start point on the chart.
 - b. Move the cursor and click the line's second point.
 - c. To complete the line setting, right-click.

- 9. To set an User Ellipse object:
 - a. Click the required point on the chart.
 - b. By moving the cursor, set the object dimensions and click.
- 10. To set a User Zone object:
 - a. Click the zone's first node position.
 - b. Move the cursor and click the position of the zone's second node .
 - c. To complete the zone setting, right-click.



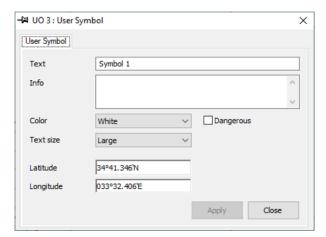
After the user object has been set on the chart, its parameters can be shown in the Chart Info window:

- 1. Choose the Chart\Chart Info menu item or click the toolbar button.
- 2. Click the contour of the user object. The Chart Info window appears.



To change the position of a User Symbol or a User Text object:

- 1. Select the User Symbol or User Text, which coordinates on the chart you want to change.
- 2. Choose the General properties item in the context menu. The following window appears:

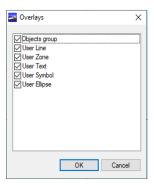


- 3. Change the latitude and/or longitude.
- 4. Click Apply. You can see that the position on the chart changes accordingly.



To turn on/off the display of user objects on the chart:

- 1. Choose the Chart\Overlays\User Info item.
- 2. Select/clear the checkboxes: Objects group, User Symbol, User Text, User Line, User Ellipse and User Zone.



3. Click OK.

2.28 Pre-Playing Exercise

An exercise can be pre-played at the Instructor workplace. The exercise pre-playing is identical to its running on the bridge except that the control of the own ships should be performed by the Instructor.

Table 10. Tools for control of exercise pre-playing

PURPOSE	MENU	HOT KEYS	BUTTON
To start the exercise pre-playing	File\Control\Start	<ctrl>+<2></ctrl>	
To suspend (to continue) exercise pre-playing	File\Control\Pause	<ctrl>+<3></ctrl>	
To stop the exercise pre-playing	File\Control\Stop	<ctrl>+<4></ctrl>	

The exercise can be pre-played on the slow, real, or fast time scale. The time scale can be selected on the toolbar (20:1, 10:1, 5:1, 2:1, 1:1, 1:2, 1:5, 1:10, 1:20).



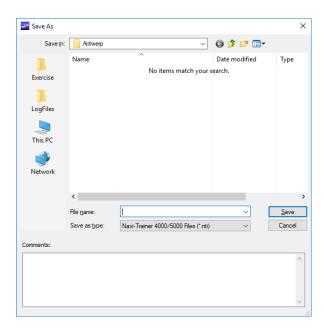
NOTE: It is not advisable to use a high acceleration rate in pre-playing exercises containing a large number of objects.

2.29 Saving and Loading Exercise

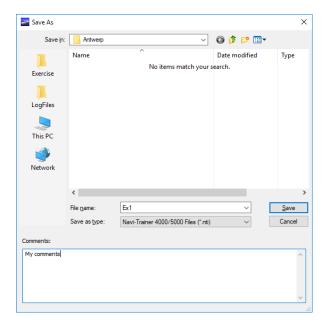
2.29.1 Saving Exercise

To save an exercise, use the following procedure:

1. Choose the File\Save or File\Save As... menu item. The Save As dialogue box appears.



- 2. Select the folder where the exercise will be saved. By default, the program offers to save the exercise in the current area folder in the NTPRO_6\Exercise folder.
- 3. In the File name field, enter the exercise name.
- 4. In the Comments field, enter or edit comments for the exercise as required.

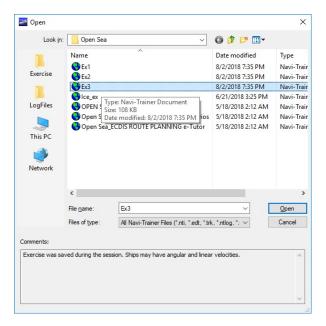


5. Click the Save button.

2.29.2 Loading Exercise

To load an exercise, use the following procedure:

- 1. Choose the File\Open... menu item or press the <Ctrl>+<O> key combination, or click the toolbar button.
- 2. Open the *Exercise* folder and select the necessary area folder. The window will display the list of exercise files with *nti extension. The tabular form will specify: file name, size, type, date when modified. The bottom part of the window will show comments for the selected exercise if these were added before by the Instructor.



NOTES:

- In the Open window, you can perform standard operations on files (deleting, renaming, copying, etc.). To do this, right-click on the exercise name and select the necessary item from the context menu.
- To sort files, click the right mouse button on an empty field in the table, and select the necessary item from the context menu, or click the mouse on the table column heading.
- 3. To load an exercise, select the exercise file in the list and click the **Open** button. The **Open** dialogue box will close down. The exercise window will appear, with a sign in its left top corner.
- 4. If the exercise contains objects missing in the objects base of the current simulator version, the **Select object** dialog box will appear. Select an object to use instead of the missing one and click the **OK** button.

To load exercise files which have been recently run, choose the File\Recent Files menu item.

3. Simulator Session

3.1	Simula	itor Session Management	207
	3.1.1	Assignment of Exercises to Bridges	207
	3.1.2	Starting Exercise	210
	3.1.3	Pausing Exercise	210
	3.1.4	Rewinding Exercise	210
	3.1.5	Stopping Exercise	211
	3.1.6	Switching Between Bridges	211
	3.1.7	Saving Current State of Exercise	211
	3.1.8	Session Overloading	211
	3.1.9	Pending Tasks	212
3.2	Contro	ol of Ships	214
	3.2.1	Control of Own Ship and Target Ship (6-DoF Model)	214
	3.2.2	Control of Target Ship (Simplified Model)	214
3.3	Contro	ol of Autopilot	217
3.4	Giving	Sound Signals	219
	3.4.1	Giving Sound Signals by Target Ships	219
	3.4.2	Giving Siren Signals	220
3.5	Motion	Parameters	221
3.6	Contro	ol of Navigation Signals	222
	3.6.1	Selecting Signals Preset Combination	222
	3.6.2	Setting Lights	223
	3.6.3	Setting Shapes	224
	3.6.4	Change of Lights and Shapes	225
3.7	Flashir	ng Light Signalling	226
3.8	UAIS N	Message Exchange	227
	3.8.1	Sending AIS Messages	227
	3.8.2	Monitoring of UAIS Messages	228
3.9	Using '	Virtual Force	229
	3.9.1	Setting Virtual Force	229
	3.9.2	Changing Virtual Force	230
	3.9.3	Setting Virtual Force Properties	230
3.10	Ship Ir	nformation Panel	231
3.11	Events	s Panel	232
3.12	Monito	oring of Dangerous Targets	234
	3.12.1	CPA/TCPA Monitoring Panel	234
	3.12.2	General CPA/TCPA Settings	234
	3.12.3	Graphic Presentation of CPA/TCPA Data	235
3.13	Ship D	liagram	237
	3.13.1	Control of Ship Diagram	237
	3.13.2	Diagram Parameter Properties	241
	3.13.3	Diagram Settings Template	243
	3.13.4	Export of Diagram Data to CSV-Format	243

	3.13.5 Diagram Printout	244
3.14	Instructor Visualization	245
	3.14.1 Adding Camera to Exercise	245
	3.14.2 Control of Instructor Visualization	248
3.15	Visualization Screen Capture	253
3.16	Loading Visual Profile on Bridge	255
3.17	Visual Settings	256
3.18	Setting Radar Position	258
3.19	Loading ECDIS Data	259
3.20	Transmitting NAVTEX Messages	260
3.21	Control of Auxiliary Machinery	263
3.22	Control of Remote Sensors	265
3.23	Creating Log Scripts	267

3.1 Simulator Session Management

3.1.1 Assignment of Exercises to Bridges

To distribute exercises among the simulator bridges, the Instructor should assign the exercise objects (own ship/tug) to the bridges.

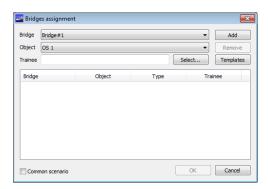
NOTE: The Helicopter and VTMS station objects can be assigned to the bridge if the necessary settings are specified in the Configuration Editor.

There are following types of the training scenario:

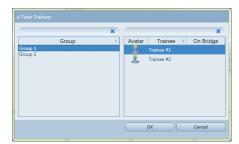
- Joint voyage whereby several bridges are created and assigned with several ships sailing along different routes
 in a single exercise. In this case, the trainees will be sailing in the same training area and will be able to see each
 other in the process;
- Separated voyage whereby own ships simultaneously sail separately in different exercises;
- Common scenario whereby identical exercises are loaded on the bridges. In this case, own ships sail separately
 in the same exercise along the same route simultaneously.

To assign exercise object(s) to the simulator bridge, use the following procedure:

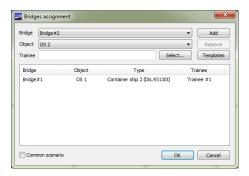
1. Click the Assign toolbar button , or press the <Ctrl>+<1> key combination, or choose the File\Control\Assign... menu item. The Bridges assignment dialogue box appears.



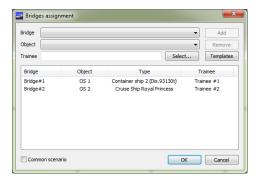
- 2. In the Bridge drop-down list, select the bridge name.
- 3. In the Object drop-down list, select the object name.
- 4. If it is necessary, enter the trainee name in the Trainee field. Or select a trainee by using the "e-Tutor" application:
 - a. Click the Select... button. The e-Tutor Trainees window appears.
 - Select the Group and Trainee and click the OK button. For more information about "e-Tutor" application, see manual E-Tutor 5000. User Manual.



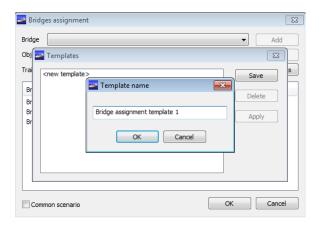
5. Click the **Add** button. In the task distribution list, a new table line appears. It contains the bridge name, the name of the object assigned to it, object type and trainee data.



6. If there is more than one bridge in the simulator and more than one object, repeat the procedure described above for each pair.

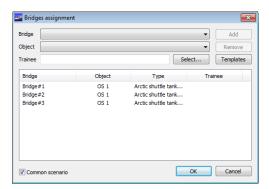


- If the assignment is required to be changed, select the line in the table and click Remove button. The line will
 disappear, and the bridge name and object name will appear in the Bridge drop-down list and the Object dropdown list respectively.
- 8. To use the template of assignment to several bridges:
 - a. Click the Templates button.
 - b. Click the Save button in the Templates dialog box.
 - c. Enter the template name and click the OK button.



- d. To apply the template in the exercise, open the **Bridges assignment** window. Click the **Templates** button. Select the required template and click the **Apply** button.
- e. To delete the template from the **Templates** dialog box, click the **Delete** button.

9. To choose a common training scenario, select the **Common scenario** checkbox. In this case, it is possible to assign the same object to several bridges.



 After the tasks distribution has been completed, click the OK button in the Bridges assignment window. The loading procedure may take considerable time. In the process of loading, the Start toolbar button remains disabled.

WARNING: If the Common Scenario is selected, the following functionality is not supported in the simulator:

- Ice functionality
- Towing and mooring operations including inland operations with articulated tug-barges and tug-barge trains
- Automatic tugs
- Application of the tug's virtual force
- Underway replenishment operations
- Fishing operations
- Giving of distress signals
- Operations with routes
- Lowering and hoisting lifeboats
- Helicopter take-off and landing from/onto the helicopter deck
- Inflatable fenders
- Bow wave and wind shading effects
- Oil spillage
- All scenarios in which the ship is mechanically interacting with the environment, e.g., anchor handling, ro-ro
 operations, wheelhouse/mast raising and lowering
- Own ship collision speed, i.e., any contact of another ship's hull with the own ship results in a collision
- Hydrodynamic interaction between own ship and other ships
- Joint operation with other simulators.

3.1.2 Starting Exercise

To start an exercise on the simulator bridge, click the **Start** button on the toolbar , or press the **Ctrl>+<2>** key combination, or choose the **File\Control\Start** menu item.

The time scale can be selected from the toolbar drop-down list:



3.1.3 Pausing Exercise

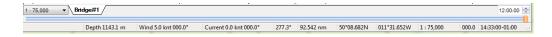
To suspend an exercise, click the **Pause** button on the toolbar , or press the <Ctrl>+<3> key combination, or choose the **File\Control\Pause** menu item.

To resume the exercise, click the **Pause** button again, or press the <Ctrl>+<3> key combination, or choose the **File\Control\Pause** menu item.

3.1.4 Rewinding Exercise

To rewind the exercise during the simulator session, use the following procedure:

1. Click the **Rewind** toolbar button, or press the <Ctrl>+<5> key combination, or choose the **File\Control\Rewind** menu item. The rewind time bar appears.



2. To set a time point from which you want to replay the exercise, move the slider on the time bar, or enter the required time in the time spinbox 12.200 .



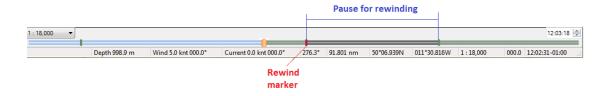
- 3. Click the rewind toolbar button again. The rewind time bar disappears.
- 4. Click the start button to start the exercise from the selected time point.

WARNING: After the exercise rewinding, the following settings are not saved:

- Fault settings (except for radar faults)
- Autopilot settings
- Monitoring object parameters in the diagram window and Research console
- E-Tutor assessment scores
- Settings for joint operation with other simulators.

The results of multiple replays are stored in the NTPRO log file with *.extlog extension.

During the log file playback, the red time marker shows the start of the exercise rewinding on the log time bar (see "Log Playback" on page 272).



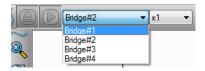
NOTE: Pause periods are displayed on the log time bar if the **Show log pauses** option is enabled on the **Misc** tab of the **Settings** panel (see "Miscellaneous Settings" on page 58).

3.1.5 Stopping Exercise

To stop the exercise, click the **Stop** button on the toolbar , or press the <Ctrl>+<4> key combination, or choose the **File\Control\Stop** menu item.

3.1.6 Switching Between Bridges

To switch between the bridges in one or more exercises, open the list of bridges on the toolbar and select the bridge.



Additionally, to switch between the bridges in one exercise, click the tab with bridge name in the left bottom corner under the chart.



The own ship assigned to the selected bridge and placed in the centre of the training area appears in the chart.

3.1.7 Saving Current State of Exercise

In the course of an exercise, the current state of exercise can be saved as an initial state of new exercise. To do this, choose the File\Save As... menu item. In the Save As dialog box, enter the file name and click the Save button.

3.1.8 Session Overloading

The simulator does not impose any limitations on the number of objects entered in an exercise. It should be noted, however, that with too many objects, there may be an overloading of the session. In this case, the screen may display the following warning:



If such warning is displayed, you should stop the exercise and analyze what caused the overload. The overloading may be due to the following reasons:

Table 11. Possible causes of session overloading

CAUSE	RECOMMENDATIONS
The computer running the "Navi-Trainer Model" program does not meet the specification	Install the computer according to the specification
The computer running the "Navi-Trainer Model" program is engaged in other tasks	It is not advisable to run other tasks on the computer intended for running the Navi-Trainer Model program
Large number of targets in an exercise or a large number of exercises (sessions) running simultaneously	Set a smaller number of targets ships in all the simultaneously running exercises
A large number of target ships in the exercise are operating in the 6-DoF	It is recommended that a simplified model of the target ship should be used if you are not planning mechanical or hydrodynamic interaction between the own ship and this target
Too large calculation time acceleration has been set	After the overloading of session the time scale will automatically change to 1:1

It should be remembered that the most resource consuming model components (from the viewpoint of calculations) are the following:

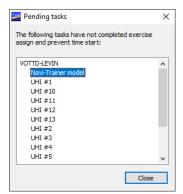
- Calculations of the ship hull interaction with the environment limits (inclined bottom, shoals, channel);
- · Calculations of the ship hull mechanical interaction with another ship or tug;
- Calculations of forces from the mooring and tow lines;
- Motion of automatic tugs on the route (proceeding to a work point with the circumventing of obstacles);
- Motion of high-speed vessels (of Combat Boat type) on the waves.

Although the developer does not provide any specific recommendations as to the reduction of the model overload, the recommendations listed above should be taken into account in the generation of an exercise.

3.1.9 Pending Tasks

Pending tasks are tasks that are initializing for a long time after assignment exercises to bridges and do not allow the Instructor to start simulator sessions.

The **Pending task** window appears on the Instructor workplace a certain time after the exercise assignment to the bridge. The window contains the list of all pending tasks on the hosts.



If the task is finally initialized, it disappears from the list of pending tasks.

The Instructor can wait until the pending task is fully initialized, or stop this task in the Router.

NOTE: In the Router, pending tasks are marked with a \triangle sign.

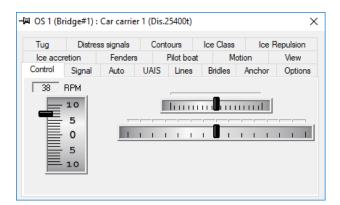
To close the **Pending tasks** window, click \times or click the **Close** button.

The Pending tasks window will reappear only after the next exercise assignment to the bridge.

3.2 Control of Ships

3.2.1 Control of Own Ship and Target Ship (6-DoF Model)

Control of the own ship and target ship (6-DoF model) can be executed from the **Control** tab of the properties panel. The tab contains propulsion and steering controls, typical for the selected ship type:



If another object is selected on the exercise chart, the panel will automatically display the selected object's controls.

For the panel to be always linked with one selected object, click the 💆 "pin" in the top left corner of the window.

NOTE: To control several objects simultaneously, you can open as many panels as there are objects, and fix each in the same way.

3.2.2 Control of Target Ship (Simplified Model)

Setting New Course

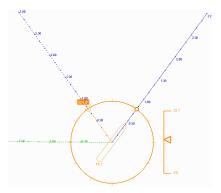
To set a new course of the target ship, use the following procedure:

- 1. Click the target ship on the chart.
- 2. Click the "course ball" on and move it around the circle. The new course is displayed on the "orange bar" near the "course ball" position.



3. Click to confirm the new course. The course will be changed over time.

If the absolute/relative speed vectors are shown on the chart, the additional absolute/relative vectors are displayed as dotted lines until the target ship course reaches the set value.



NOTE: To show absolute vectors on the chart, click toolbar button (View\Vectors\Absolute menu). To show relative speed vectors on the chart, click the toolbar button (View\Vectors\Relative menu).

4. To immediately change a course, press and hold the <Shift> key, then follow steps 2, 3.

Setting New Speed

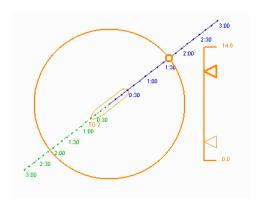
To set the target's speed, use the following procedure:

- 1. Click a target ship on the chart.
- 2. Click the "speed throttle" and move it along the scale. The new speed is displayed on the "orange bar" near the "speed throttle" position.



3. Click to confirm the new speed. The speed will be changed over time.

If the absolute/relative speed vectors are shown on the chart, new speed vectors length is adjusted in accordance with a new set speed.

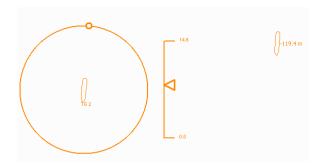


4. To immediately change the speed, press and hold the <Shift> key, then follow steps 2, 3.

Setting New Position

To set the target's position:

- 1. Click a target ship on the chart.
- 2. Press and hold the <Shift> key, click the ship contour and drag it to a new position.

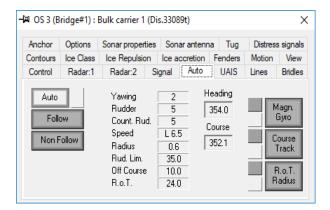


3. Click to confirm the new position. The target ship position will change immediately.

3.3 Control of Autopilot

To control the own ship/tug autopilot, use the following procedure.

- 1. Open the Auto tab of the ship properties panel.
- 2. Make sure that the own ship's speed is above 4 knots.
- 3. Click the Auto button to switch the autopilot on. The indicator to the right of the Auto button glows



- 4. Click the Magn./Gyro button to select the heading device type.
- 5. Click the Course/Track button to select the autopilot mode:
 - Course control mode.
 - Track control mode (the own ship should be assigned to a route).
- 6. Click the R.o.T./Radius button to select the vessel turn mode (Rate of Turn or Turn Radius).

Digital indicators display the autopilot parameters and settings:

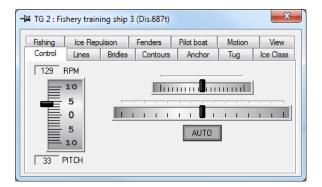
- Yawing coefficient of rudder angle change rate in the ship yaw;
- Rudder coefficient of correspondence between the rudder angle and deviation from course value;
- Count. Rud. ship steadying coefficient as the set course is steered to;
- Speed ship speed;
- Radius ship turn radius as the course is altered;
- Rud. Lim. maximum permissible rudder turn position in degrees;
- o Off Course error limit in the deviation from the course;
- R.o.T. ship's rate of turn with a change of course in °/min;
- Heading current heading value in degrees;
- Course set ship course in degrees.

NOTE: Autopilot settings can be made from the ship bridge only.

- 2. To switch the ship steering from the Auto to the Manual mode:
 - a. Click the **Follow** button to switch on the Follow mode. The Trainee can steer the vessel by using the Follow Up Helm.
 - b. Click the **Non Follow** button to switch on the Non-Follow mode. The Trainee can steer the vessel by using the Non-Follow up tiller.

To switch on/off the Track Control mode for the target ship (6-DoF model), use the following procedure:

1. Open the Control tab of the target ship properties panel.



2. Click the AUTO button to switch on/off the Track Control mode of the target ship autopilot.

NOTE: The autopilot of a target ship (6-DoF model) operates only in the Track Control mode if the target ship is assigned to a route. The Track Control mode will be automatically switched off after the target reaches the last route waypoint.

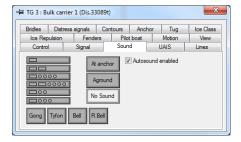
3.4 Giving Sound Signals

3.4.1 Giving Sound Signals by Target Ships

During the simulator session, sound signals are given by a target ship/tug automatically in the poor visibility conditions (fog) regardless of the time of the day.

The **Sound** tab of the target ship properties panel contains sound signals' indicator buttons which glow when signals are automatically given.

To disable automatic signalling at the moment, click the **No Sound** button. The automatic signalling will be disabled until the necessary conditions are fulfilled, e.g., a poor visibility zone.

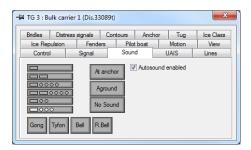


To turn off the automatic signalling, clear the Autosound enabled checkbox.

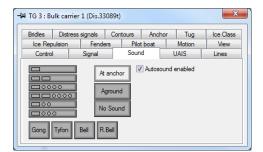
NOTE: It is possible to disable/enable the automatic sound signals for the selected target ships on the **Sound** signals of the **Settings** panel (see on page 57).

To give sound signal manually:

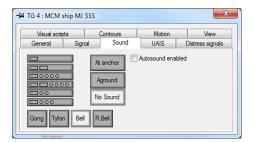
1. To give a sound signal in the poor visibility conditions, click the appropriate button in the left-hand buttons column. The signal will be given by the tyfon.



2. Click the **At anchor** or **Aground** button to select signals given by the ship when anchored or aground. The signal will be given by the gong or bell.



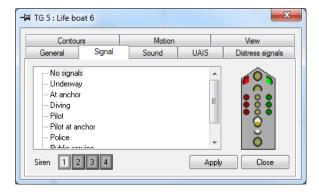
- 3. Click and hold the button:
 - Gong to give a gong signal;
 - Tyfon to give a tyfon signal;
 - Bell to give a bell signal;
 - R. Bell to give a repeating bell signal.



3.4.2 Giving Siren Signals

To give a siren signal, use the following procedure.

- 1. Open the Signal tab of the ship properties.
- 2. To turn on the siren signal, click one of the Siren buttons: 1, 2, 3, 4 (the option is available for certain objects).

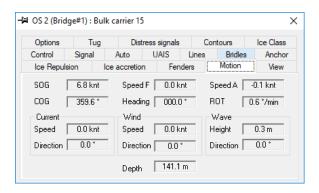


3. To turn off the siren signal, click the corresponding Siren button again.

NOTE: The trainee can give the siren signals on the Signals page of the Conning display.

3.5 Motion Parameters

The **Motion** tab of the own ship/tug (target ship/tug, barge) properties panel displays the following current ship motion parameters:



- SOG speed over ground;
- COG course over ground;
- Speed F, Speed A fore and aft speed;
- · Heading;
- ROT rate of turn;
- · Current current speed and direction;
- Wind wind speed and direction;
- · Wave wave height and direction;
- · Depth.

3.6 Control of Navigation Signals

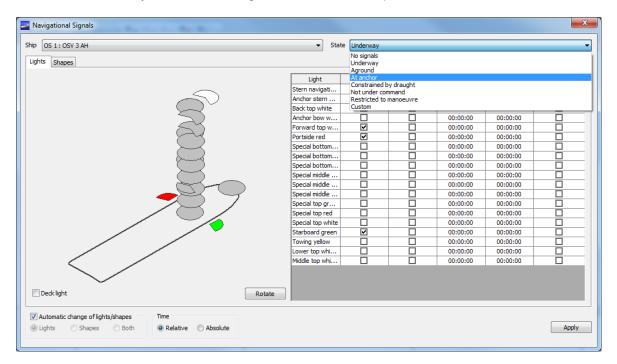
The Navigational signals panel is used to control navigation signals (lights and shapes).

The set of available lights/shapes depends on the vessel model.

3.6.1 Selecting Signals Preset Combination

To select preset combination of lights and shapes, use the following procedure:

- 1. Choose the View\Panels\Navigational Signals... menu item, or click the toolbar button.
- 2. Select an own or target ship from the Ship drop-down list of the Navigational Signals panel.
- 3. Select the necessary combination of signals from the State drop-down list.



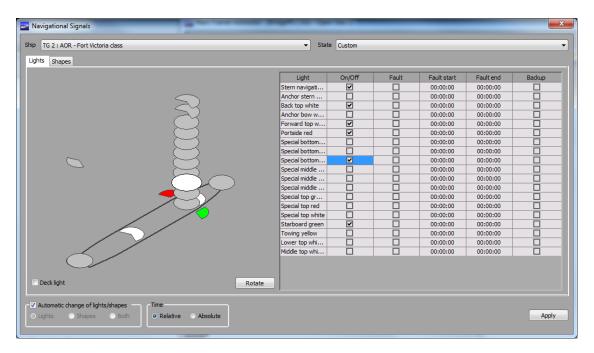
4. Click the Apply button.

NOTE: The combination of lights and shapes can also be set from the **Signals** page of the ship properties panel (see "Navigational Signals Settings" on page 112).

3.6.2 Setting Lights

To set lights, use the following procedure:

- 1. Choose the View\Panels\Navigational Signals... menu item, or click the toolbar button.
- 2. Select an own or target ship from the Ship drop-down list of the Navigational Signals panel.
- 3. Open the Lights tab.
- 4. To turn on/off the lights, check/clear the necessary lights checkboxes in the table's **On/Off** column and click the **Apply** button.

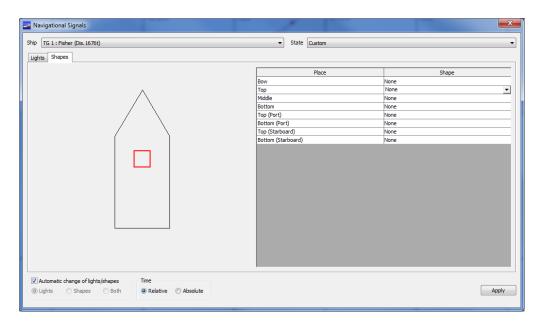


- 5. To set a light fault, check the required light in the Fault column and click the Apply button.
- 6. To set the light fault duration:
 - a. Select the time type (Relative or Absolute) in the Time group box.
 - b. Enter the fault start time in the Fault start column.
 - c. Enter the fault end time in the Fault end column.
 - d. Click the Apply button.
- 7. To turn on a backup light, check the required light in the table's **Backup** column and click the **Apply** button.
- 8. To turn on deck lights, check the Deck light checkbox and click the Apply button.

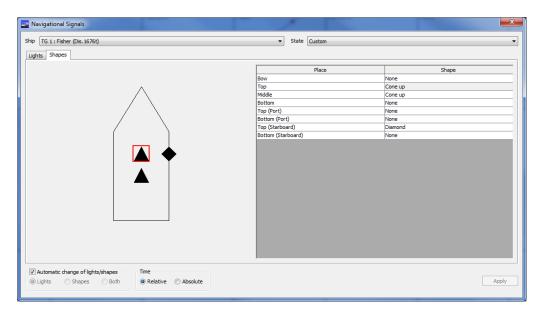
3.6.3 Setting Shapes

To set shapes, use the following procedure:

- 1. Choose the View\Panels\Navigational Signals... menu item, or click the toolbar button.
- 2. Select an own or target ship from the Ship drop-down list of the Navigational Signals panel.
- 3. Open the Shapes tab.
- 4. Click on the Shape column next to the required shape place. On mimic the red square shows the selected place:



- 5. Select the necessary shape from the drop-down list (Sphere, Cone up, Diamond, Cone down, Cylinder). Click the **Apply** button.
- 6. Repeat steps 4 and 5 to set other shapes.



3.6.4 Change of Lights and Shapes

By default, the navigational shapes are automatically replaced with navigational lights at night or in low visibility (fog).

The Instructor can disable the automatic change of navigation signals and specify which navigation signal will be displayed regardless of the daytime and visibility conditions. To do this, use the following procedure:

- 1. Choose the View\Panels\Navigational Signals... menu item, or click the toolbar button.
- 2. Clear the Automatic change of lights/shapes checkbox.
- 3. Select the navigational signal type: Lights/Shapes/Both.
- 4. Click the Apply button.

3.7 Flashing Light Signalling

For a target ship to give flashing light signals to the own ship, use the following procedure:

- 1. Start the exercise.
- 2. Choose the View\Panels\Flash Light....menu, or click the lettoolbar button.
- 3. In the Source drop-down list, select the target ship signal source.
- 4. In the Recipient drop-down list, select the own ship signal recipient:



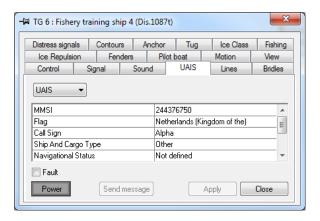
- 5. In the Manual mode (the **Manual** option is selected), to give a flashing light signal, click and hold the **Flash** button. The indicator glows as it is held.
- 6. In the Text mode (the Text option is selected), do the following:
 - a. Type an alphanumeric message in the Message field;
 - b. Set the signal speed (word/min) in the Speed field;
 - c. Click the Send button. The flashing light signal will be given by using the international Morse code.

3.8 UAIS Message Exchange

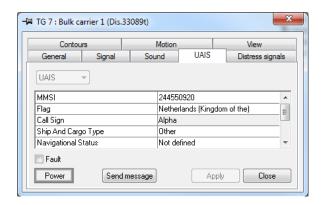
3.8.1 Sending AIS Messages

The Instructor can send any text message on behalf of any target ship. To do this, use the following procedure:

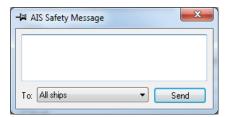
1. Open the target ship properties panel and select the UAIS tab.



- 2. Select the AIS type from the drop-down list: UAIS or IAIS (Inland AIS).
- 3. Change the UAIS or IAIS parameters in the table as required.
- 4. To turn on the transponder, click the Power and Apply buttons. The Send message button will be activated.



5. Click the **Send message** button. The **AIS Safety Message** window appears.

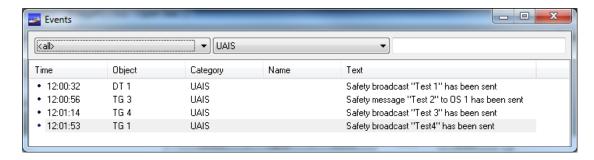


- 6. Type any message text.
- 7. From the To: drop-down list, select the destination.
- 8. Click the Send button.

3.8.2 Monitoring of UAIS Messages

To monitor all UAIS messages sent during the exercise, use the following procedure:

- 1. Choose the View\Panel\Events... menu item. The Events panel appears.
- 2. Select the UAIS category from the right drop-down list.



3.9 Using Virtual Force

The Instructor can set virtual horizontal force affecting the own ship/tug or target ship (6 DoF model). The force can be applied to any point of the ship hull in any direction horizontally.

NOTE: Virtual forces are not applied to the fleet formation.

3.9.1 Setting Virtual Force

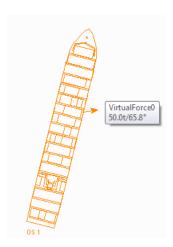
To set a virtual force, use the following procedure:

- 1. Open the object context menu and choose the Virtual force item.
- 2. In the **New virtual force** window, enter a virtual force name as required:



NOTE: Before the simulator session start, the Instructor can rename the virtual force. To do this, open the context menu of the force's vector and choose the **Rename force** item.

- 3. Click the OK button.
- 4. Click on the ship hull in the force application point. The force application point will be fixed on the ship hull.
- 5. By moving the cursor, set the force value and direction, and click the left mouse button. The force vector will be fixed on the chart:



6. Repeat steps 1-5 to create other virtual force vectors.

3.9.2 Changing Virtual Force

To change the virtual force parameters graphically, use the following procedure.

- 1. Open the context menu of the force vector and choose the **Change force** item.
- 2. By moving the cursor, set the force value and direction, and click. The force vector will be fixed on the chart.

To change the application point and parameters of a virtual force, use the following procedure:

- 1. Open the context menu of the force vector and choose the **Move force** item.
- 2. Click on the ship hull in the force application point. The force application point will be fixed on the ship hull. The mouse cursor will assume the shape of a force vector.

NOTE: Right-click to set a virtual force in a new point without changing its parameters.

3. By moving the cursor, set the force value and direction, and click. The force vector will be fixed on the chart.

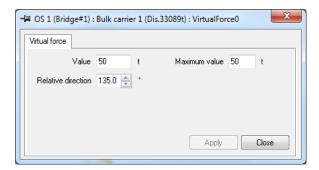
To remove a virtual force, use the following procedure.

- 1. Open the context menu of the force vector.
- 2. Choose the Remove force menu item.

3.9.3 Setting Virtual Force Properties

To set the virtual force properties, use the following procedure:

1. Open the context menu of the force vector and choose the **Properties** item. The **Virtual force** tab appears.



- 2. Set the following virtual force properties:
 - Value current value of the force vector, t;
 - Relative direction relative direction of the force vector, deg;
 - o Maximum value maximum value of the force vector, t.
- 3. Click the Apply button.

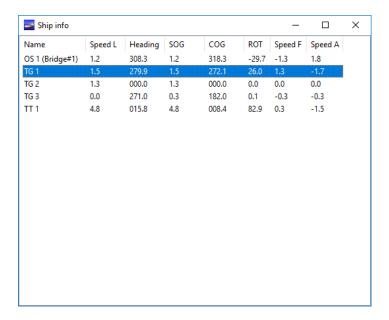
3.10 Ship Information Panel

During the simulator session, the Ship info panel displays the following information on ships (target ships):

- Name -vessel and bridge names.
- Speed L longitudinal speed;
- · Heading heading;
- SOG speed over ground;
- · COG course over ground;
- ROT rate of turn;
- Speed F fore transverse speed;
- Speed A aft transverse speed.

To open the panel, use the following procedure:

1. Choose the View\Panels\Ship info... menu item, or click the toolbar butto



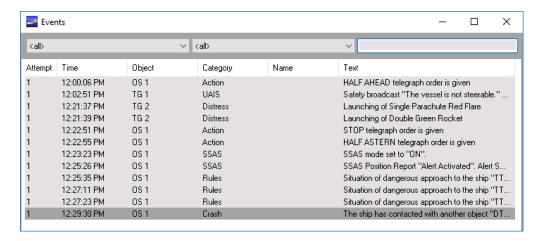
2. Make a double left-click to select an object from the list. The object will be placed in the centre and highlighted in orange on the exercise chart.

NOTE: Information on ships can be sorted by any of parameters. To sort, click on the required table heading.

3.11 Events Panel

The Events panel displays the list of events, which take place during the simulator session.

To show the **Events** panel, choose the **View\Panels\Events**...menu item, or click the toolbar button.



The table of the **Events** panel contains:

- Attempt the exercise attempt number as a result of rewinding (if the value is 1, the rewinding of the exercise was not used).
- Time event time;
- Object exercise object, with which the event is associated;
- Category event category;
- · Name event name;
- Text tag text.

The events may be sorted in the table:

- To sort events by the object, select the ship name from the left drop-down list.
- 2. To sort events by the category, select the event category from the middle drop-down list.
- 3. To sort events by the text comment, enter the "key word" in the right text box.

There are the following events categories:

- Action messages on the Trainee performance on the bridge are formed when the autopilot, lights, or sound signals are turned on/off, anchor is dropped or heaved, etc.;
- Crash messages on accidents are formed in the case of ship collision or grounding;
- Distress messages on the Trainee sending a distress signal;
- Intercom messages on the Trainee actions during the work with the Intercom;
- Ice Navigation messages on the Trainee performance in conducting navigational operations in ice conditions;
- Lock message on the Trainee breaking the lock passing rules;
- · Lines messages during the work with mooring lines;
- Rules messages on sailing rule violation are formed for dangerous targets as calculated CPA/TCPA values
 exceed their limits;
- SSAS messages are formed when the Trainee works with the Ship Security Alert System;

- UAIS messages on the trainee steps in the operation with UAIS are formed when text messages are sent, and the station static and dynamic data is changed;
- Instructor the Instructor generated events;
- VHF messages on the Trainee actions during the work with the VHF communication.

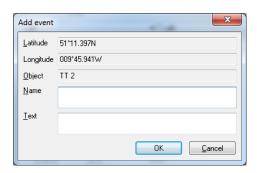
The events with comments may be added by the Instructor. An instructor event can be associated with any own and target ship.

To add the instructor event, use the following procedure:

1. Right click on the **Events** panel. The **Add event** menu appears on the panel.



- 2. Choose the Add event... menu item.
- 3. Click any object on the chart. The screen will display Add event containing the object position and name.



4. Type the event name in the **Name** field, and any text comment in the **Text** field and click the **OK** button. The event will be added to the list of events.

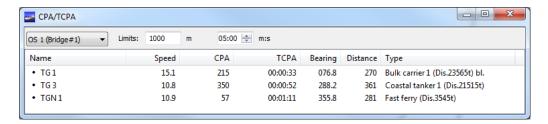
NOTE: For focusing the chart on a geographic point where the event occurred, double left-click on the event on the **Events** panel.

3.12 Monitoring of Dangerous Targets

3.12.1 CPA/TCPA Monitoring Panel

To monitor the CPA/TCPA data, use the following procedure:

1. Choose the View\Panels\CPA/TCPA... menu item, or click the toolbar button. The CPA/TCPA panel appears. The panel contains the list of dangerous targets by the CPA/TCPA criterion.



- 2. Select a reference ship from the drop-down list of vessels.
- 3. Set the CPA/TCPA limit values in the Limits fields:
 - a. Set the distance at which the dangerous targets will be monitored 1000 m.
 - b. Set the monitoring time 05:00 🖨 m:s.

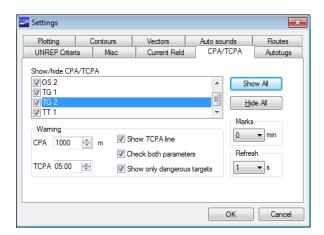
NOTE: If the CPA/TCPA values are less than the limit values, the dangerous target parameters will be displayed in the table below.

To select an object on the chart, make a double left-click on the object from the list in the table.

3.12.2 General CPA/TCPA Settings

To adjust general CPA/TCPA settings, use the following procedure:

1. Choose the View\Settings... menu item and open the CPA/TCPA tab of the Settings panel.



- 2. Select checkboxes of the objects which the CPA/TCPA data should be displayed for.
- 3. To show CPA/TCPA data for all the objects, click the Show All button. To hide it, click the Hide All button.
- 4. Set the limit values for the CPA and TCPA:
 - CPA distance to the closest point of approach;

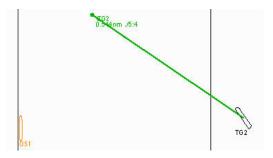
- TCPA time to the closest point of approach.
- 5. To display the CPA/TCPA data for dangerous targets only, select the **Show only dangerous targets** checkbox (the checkbox is selected by default). The CPA/TCPA values which are less than or equal to the set limit values will be shown on the chart.
- 6. Select the **Check both parameters** checkbox to classify the target as dangerous if the values of both CPA and TCPA are less than or equal to the set limit values.
- 7. Select the **Show TCPA line** checkbox to show the TCPA line on the chart (it is selected by default). If the checkbox is cleared, the CPA point will be displayed on the chart only.
- 8. From the **Marks** drop-down list, select an interval between the time markpoints on the TCPA line (0 min by default).
- 9. From the **Refresh** drop-down list, select the time interval for updating the CPA/TCPA data graphic display (1 sec., by default). The CPA/TCPA graphic information will be updated at the selected intervals.

NOTE: For more information, see "Graphic Presentation of CPA/TCPA Data" on page 235.

3.12.3 Graphic Presentation of CPA/TCPA Data

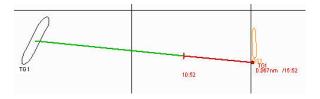
During the exercise, the CPA/TCPA data can be presented graphically in three ways:

 If the current CPA/TCPA values are greater than the limit CPA/TCPA values, the CPA point and the TCPA line will be marked in green:

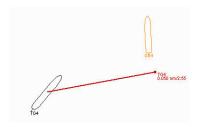


NOTE: The limit CPA/TCPA values are set on the CPA/TCPA tab of the Settings panel (see "General CPA/TCPA Settings" on page 234).

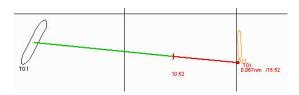
- The TCPA line will be divided in two parts (green and red) in the following ways:
 - Green part of TCPA line shows time before the dangerous TCPA. In this case the current CPA/TCPA values are greater than the limit CPA/TCPA values
 - Red part of TCPA line shows the dangerous TCPA. In this case the current CPA/TCPA values are less than the limit CPA/TCPA values.



• If the current CPA/TCPA values are less than or equal to the limit CPA/TCPA values, the CPA point and the TCPA line are shown in red:



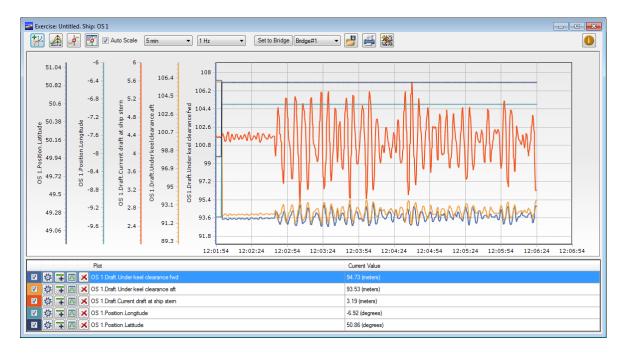
To turn on the graphic presentation of the CPA/TCPA data, choose the View\Vectors\Relative menu item, or click the toolbar button). The chart will graphically display the CPA/TCPA data as per the settings made. The current CPA/TCPA values are shown next to the CPA point.



NOTE: To set the relative speed vector display parameters, open the **Vectors** tab of the **Settings** panel (see "Speed Vectors Settings" on page 55).

3.13 Ship Diagram

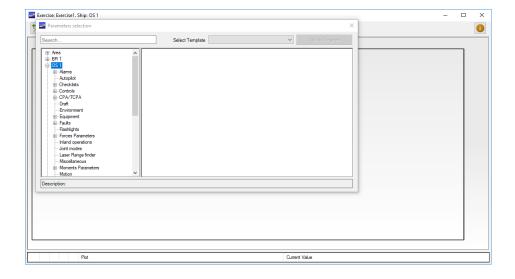
During the simulator session and the log file playback, the Instructor can monitor the change of the vessel motion parameters in the Ship Diagram window.



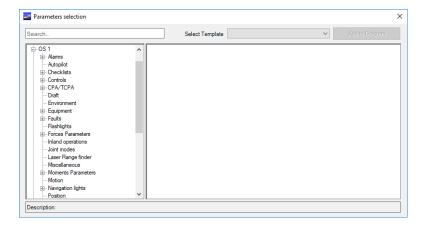
3.13.1 Control of Ship Diagram

To control the ship diagram, use the following procedure:

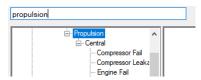
1. Open the context menu of any own ship and choose the **Ship diagram** item. The **Parameters selection** window appears. The left part of the window contains the list of exercise objects, exercise area and time parameters.



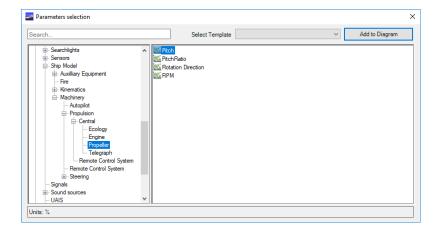
- 2. In the Parameters selection window, select the parameters for monitoring:
 - Select the required ship (target ship) from the list of objects. The list of parameters groups opens (it depends on the ship model).



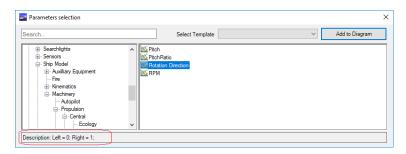
b. Select a group and subgroup of parameters from the list. To quickly find a parameter, enter the name of the parameter or group of parameters in the search bar.



The list of parameters opens in the right part of the Parameters selection window.



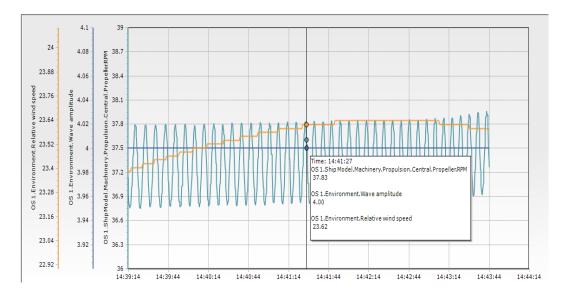
c. Left click on the parameter name. The parameter description appears at the bottom of the window.



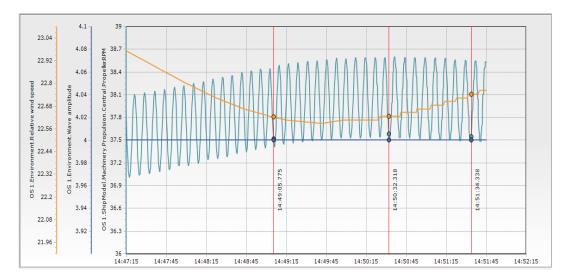
- d. Double left-click on the parameter name or click the **Add to Diagram** button to add the parameter for monitoring in the diagram window.
- e. After the parameters have been added for monitoring, close the Parameters selection window. The coloured curves of parameter values will be shown on the plot of the diagram window. The horizontal axis shows the time, the vertical axis shows the parameter values which are displayed in the same colour as the corresponding curves. The parameters information table appears below the plot.



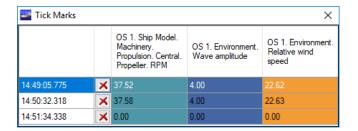
- 3. To add new parameters for monitoring, click the button. The Parameters selection window appears. Repeat step № 2.
- 4. To read off parameter values at the specified time, click the button and move the vertical line along the time axis. The prompt window shows the parameter values at the selected time.



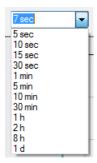
5. To add a tick-mark to the diagram, click the button. If it is necessary, drag the tick-mark on the time axis.



To view information on tick-marks, click the button.



- 6. If it is necessary, click the **Auto Scale** checkbox to automatically scale the diagram window when the parameter exceeds existing limits.
- 7. From the drop-down lists, select:
 - the time interval history of changing parameter values on a plot.



o the updating rate.



- 8. To send information on monitored parameters to the simulator bridge (option is available after the exercise start):
 - a. Select the bridge name from the bridge drop-down list.
 - b. Click the **Set to Bridge** button . Information on monitored parameters will be displayed on the **Research** console of the bridge.

NOTE: The Trainee can add tick marks to the plot, show/hide parameter curves on the plot and read off parameters values at the specified time.

- 9. To export diagram data to the CSV format, click the button (see "Export of Diagram Data to CSV-Format" on page 243).
- 10. To print diagram data, click the button (see "Diagram Printout" on page 244).
- 11. To create a ship diagram settings template, click the button (see "Diagram Settings Template" on page 243).
- 12. To hide/show the parameters information table in the bottom part of the ship diagram window, click the button.
- 13. To set parameters properties, see "Diagram Parameter Properties" on page 241.

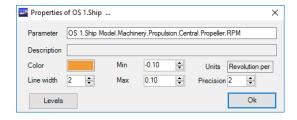
3.13.2 Diagram Parameter Properties

To change settings of parameters displayed in the ship diagram window, use the following procedure:

1. Select the required parameter in the table in the bottom part of the ship diagram window.

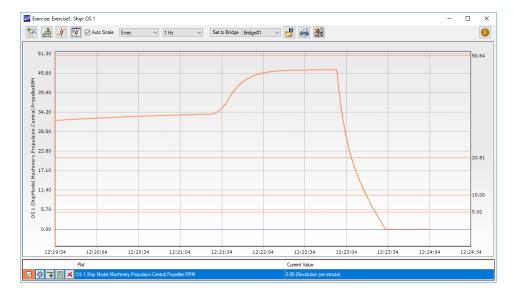


- 2. To show/hide the parameter curve on the plot, select/remove the ✓ checkbox.
- 3. To set the parameters properties:
 - a. Click the button. The parameter properties window appears.

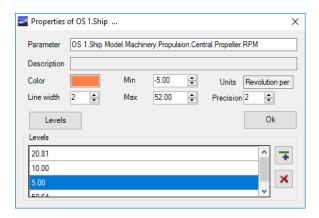


- b. In the Parameter field, enter the parameter name which will be shown in the diagram window (if required).
- c. In the **Min** and **Max** fields, set the parameter minimum and maximum values (scale range of parameter values).
- d. In the Color field, set the colour of the parameter's curve in the ship diagram window. To do this, click on the Color field and select the required colour in the Color dialogue box, click the OK button.
- e. In the Line width field, set the width of the parameter's curve in pixels.
- f. In the Precision field, specify decimal places.
- g. Click the OK button.

- 4. If you want to add a level on a plot to show constant value of the parameter, use one of the following ways:
 - Click the button and click on the plot.



o Open the parameter properties window . Click the **Levels** button. The **Levels** list box appears. To add a new level to the plot, click the . button, enter the required value and click the **OK** button.



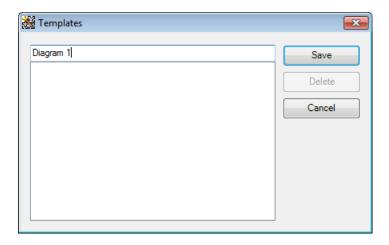
NOTE: If the user has previously added levels, the values of these levels are displayed in the **Levels** list box of the parameter properties window.

- 5. To remove the parameter's level: open the parameter properties window . Select the level in the **Levels** list box and click the button.
- 6. To display the ship diagram in a new window, click the <a> button.
- 7. To stop the parameter monitoring in the ship diagram window, click the **b**utton.

3.13.3 Diagram Settings Template

The Instructor has a capability to save specified settings of the ship diagram window. To do this, create the template:

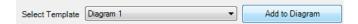
- 1. Open any own ship context menu and choose the **Ship diagram** item.
- Select required parameters in the Parameters selection window (see "Control of Ship Diagram" on page 237).Close the Parameters selection window.
- 3. Set the parameter properties (see "Diagram Parameter Properties" on page 241).
- 4. Click the button in the ship diagram window.
- 5. In the **Templates** dialog box, enter the template name and click the **Save** button.



To delete the diagram settings template, select it in the Templates window and click the Delete button.

To load the diagram settings template to the exercise, use the following procedure:

- 1. Open any own ship context menu and choose the **Ship diagram** item.
- In the Parameters selection window, open Select Template drop-down list, select the required template and click Add to Diagram.



3.13.4 Export of Diagram Data to CSV-Format

If at least one monitored parameter is selected in the ship diagram window, the data related to the parameter can be exported to the CSV-format. To do this, use the following procedure:

- 1. Click the button in the ship diagram window.
- 2. In the Export to CSV dialog box, set the following parameters:

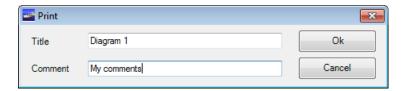


- From start of a time period;
- o To end of a time period;
- Step graduation step.
- 3. Click the Export button.
- 4. In the Save As window, select a folder, enter the name of the file and click the Save button.

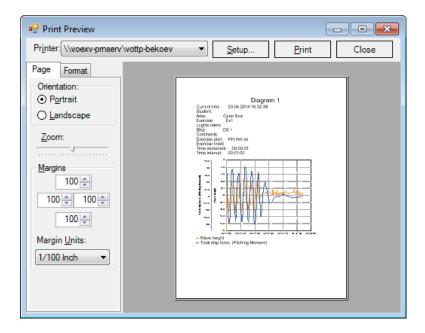
3.13.5 Diagram Printout

To print out the diagram data, use the following procedure:

- 1. Open the ship diagram window (see "Control of Ship Diagram" on page 237).
- 2. Click the button.
- 3. In the Title field, enter the heading (diagram name). In the Comment field, enter comments if required. Click Ok.



4. In the **Print Preview** window, adjust printing settings if necessary.



5. Click the Print button.

3.14 Instructor Visualization

The instructor visualization expands the instructor functional capabilities and is used for visual monitoring of the surroundings during the exercise.

The main capabilities of the instructor visualization are:

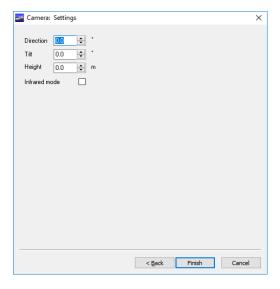
- Formation of visual monitors from several visual channels;
- Setting of an unlimited number of instructor visual cameras, both at the preparation stage and during the exercise conduct.

The instructor visual camera can be set in any place on the chart, or attached to mobile or stationary exercise objects (own ships, target ships, aircraft, helicopters, life rafts, VTMS stations, etc.).

3.14.1 Adding Camera to Exercise

To add an instructor visual camera to the exercise, use the following procedure:

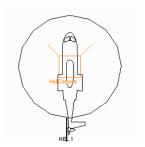
- 1. Select the Camera object in one of the following ways:
 - Choose the Edit\New object menu item and select the Camera object from the Scene category. If necessary, enter the camera name in the Name field. Click the Next button.
 - ∘ Choose the Edit\Add Camera menu item, or click the toolbar button .
- 2. In the Settings window, set the camera general properties:
 - a. In the Direction field, set the camera direction with respect to the true north.
 - b. In the Tilt field, set the camera tilt angle.
 - c. In the Height field, set the camera height.
 - d. To enable the infrared camera mode, select the Infrared Mode checkbox.



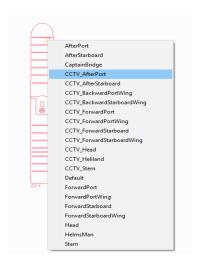
- 3. Click the Finish button.
- 4. Click the required point on the chart. The camera position is fixed in the selected place.



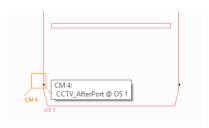
- 5. Or attach the camera to an object:
 - a. Click the required object. The camera will be attached to the default viewpoint of this object.



b. If the object has several viewpoints, choose the required viewpoint from the context menu.



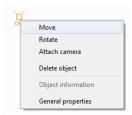
The camera will be attached to the selected viewpoint of this object.



NOTE: In the process of exercise fulfillment, the camera position and properties are recorded in a log file for debriefing.

To operate the camera, use the camera context menu:

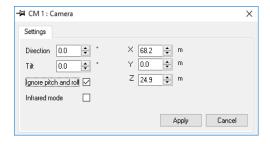
- Move to move the camera to another place on the chart (unless the camera is attached to the object).
- Rotate to change the camera direction (unless the camera is attached to the object).
- Rename rename the camera object (only before the exercise start).
- Attach to attach the camera to the object (only to the default object's viewpoint).
- Detach to dettach the camera from the object.
- Delete object to remove the camera from the chart.



NOTE: If an object which the camera was attached to is deleted, the camera is detached automatically, the camera's position remaining unchanged.

To set properties of the attached camera:

- 1. Choose the **General properties** item from the camera context menu.
- 2. Set the following parameters:
 - o Direction camera direction relative to the object direction;
 - o Tilt camera vertical tilt;
 - o X, Y, Z camera shift along the longitudinal/transverse/vertical axis relative to the object centre.
- 3. To prevent the camera from pitching and rolling with the object, select the Ignore pitch and roll checkbox.
- 4. To enable the infrared camera mode, select the Infrared mode checkbox.



5. Click the Apply button.

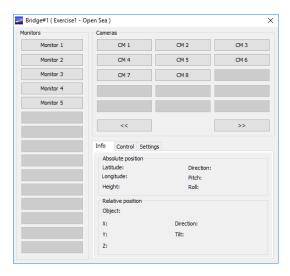
3.14.2 Control of Instructor Visualization

Control of Instructor visualization is executed from a special panel. To open the panel, choose the View\Panels\Camera control... menu item.

3.14.2.1 Assigning Cameras to Monitors

To assign camera to the monitor, use the following procedure:

1. Choose the View\Panels\Camera control... menu item or click the toolbar button. The window for controlling the Instructor's visualization appears.



- 2. Click the button with the monitor name in the Monitors group.
- Click the button with the camera name in the Cameras group. To move through the camera buttons list (forward and backwards), use the arrow buttons.

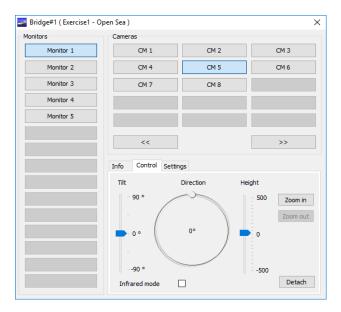


NOTE: If the monitor is selected, any successive clicking on the camera button will result in this camera reassignment to this monitor. The same camera can be assigned to several monitors.

3.14.2.2 Control of Cameras

To control the camera, use the following procedure:

- 1. Choose the View\Panels\Camera control... menu item, or click the toolbar button.
- 2. Click the button with the camera name in the Cameras group.
- 3. Select the Control tab.



- 4. Move the scroll boxes to set the following camera parameters:
 - Tilt camera tilt angle (from -90 to 90 degrees);
 - o Direction camera direction with respect to the true north;
 - Height camera height (from -500 to 5000 m).
- 5. To zoom in the picture on the visualization, click the Zoom in button.
- 6. To zoom out the picture on the visualization, click the **Zoom out** button.
- 7. To revert the picture on the visualization to the camera default position, click the **Revert position** button.
- 8. To enable the infrared mode, select the Infrared mode checkbox.
- To detach the camera from the object, click the **Detach** button (the button is not enabled unless the camera is attached to an object).

The Info tab shows the following information about the location of the selected camera:



- Absolute Position group box:
 - Latitude and Longitude;
 - Height camera elevation above the sea level;
 - o Direction camera direction (turn angle) relative to the true north;
 - o Pitch camera's vertical inclination angle (the trim or pitch angle taken into account);
 - o Roll camera's transverse inclination angle (the heel angle taken into account).
- Relative position group box:
 - X longitudinal shift relative to the object centre;
 - Y transverse shift relative to the object centre;
 - Z vertical shift relative to the object centre;
 - o Direction camera direction (turn angle) relative to the object direction;
 - o Tilt camera's vertical inclination angle.

NOTE: Relative parameters are not displayed unless the camera is attached to an object.

3.14.2.3 Camera Settings

To adjust the camera settings, use the following procedure:

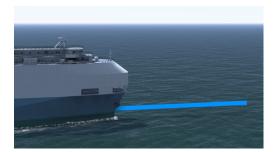
- 1. Choose the View\Panels\Camera control... menu item, or click the toolbar button.
- 2. Click the button with the camera name in the Cameras group.
- 3. Select the Settings tab.



4. To display the ship names (above the ships) on the Instructor visualization screen, select the **Ship names** checkbox.



- 5. To display absolute speed vectors on the Instructor visualization screen:
 - a. Select the Speed vectors checkbox;
 - b. Select the colour from the Color drop-down list;
 - c. Set the length of the speed vector by using the scroll box (shorter or longer).



- 6. To display routes on the Instructor visualization screen:
 - a. Select the Routes checkbox;
 - b. Select the colour of routes from the Color drop-down list.
- 7. To display the composite condition zones borders:
 - a. Select the Local zone borders checkbox;
 - b. Select the colour of borders in the Color drop-down list.
- 8. To switch off the display of weather conditions (layers of clouds, low visibility, rough sea, ship roll) on the Instructor visualization screen, select the **Suppress weather** checkbox.
- 9. To prevent the camera from pitching and rolling with the object, select the **Ignore pitch and roll** checkbox.
- 10. To display User objects on the Instructor visualization screen, select the User objects checkbox.
- 11. To limit the usage of the "3Dconnexion SpaceMouse compact" controller, clear the Allow 3D controller box when not necessary. The box is checked by default enabling the controller operation.

3.14.2.4 Advanced IR Camera Settings

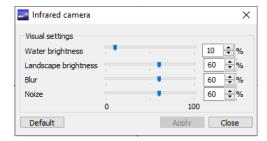
The simulator features a capability to perform exercises in night time and low-visibility conditions by using the advanced infrared camera.

For operation with the camera, the simulator bridge configuration is required with a separate visualization task: Visualization channel 2 (see *Navi-Trainer 6 Professional. Technical Description and Installation Manual. Version 6.0*).

Using this advanced IR camera, you can focus your view on a single direction of interest around the own ship and then detect and track the required object (e.g., a man overboard) in full darkness or low light.

The camera environment visual settings can be adjusted via the Infrared camera panel of the Instructor workspace.

To open the panel, choose the View\Panels\Infrared camera... menu item, or click the toolbar button.



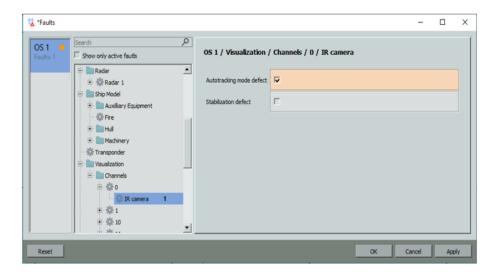
The Infrared camera panel contains the following controls:

- Water brightness to set the water brightness from 0 to 100% (default: 10%);
- Landscape brightness to set the landscape brightness from 0 to 100% (default: 60%);
- Blur to set the environment blur (default: 60%);
- Noize to set the noise level from 0 to 100% (default: 60%);
- Default to reset the camera to the default values;
- Apply and Close to apply the changes and close the panel.

Setting Camera Faults

To set the camera faults, use the following procedure:

- 1. Choose the View\Panels\Faults ... menu item. The Faults panel appears (see "Faults Panel" on page 132).
- 2. In the left column of the panel, select an own ship.
- 3. In the middle column of the panel, select the Visualization\Channels\0\IR camera fault category.
- 4. In the right column of the panel, select the checkbox next to the name of the fault you want to activate (for the details, see the table below).



5. Click OK to accept changes and close the Faults panel.

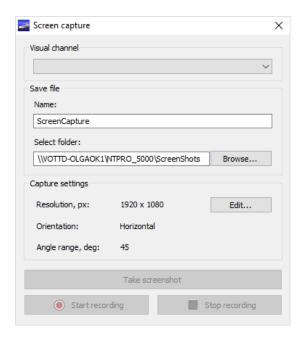
Table 12. Advanced IR camera faults

FAULT	DESCRIPTION
Autotracking mode defect	The lock mechanism is out of order. The locked target is dropped, if any. Clicking the Lock button has no impact on the lock state - it is not possible to lock a new target.
Stabilization defect	The vertical stabilization is out of order. It is nearly impossible to find and lock floating objects in hard waves (e.g. a man over board).

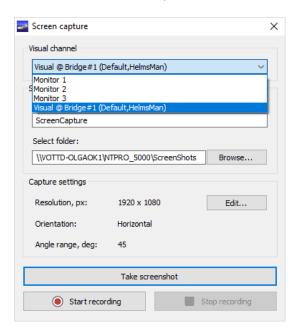
3.15 Visualization Screen Capture

To take screenshots or capture video of the visualization screen, use the following procedure:

1. Choose the View\Panels\Screen capture.. menu item, or click the window appears.



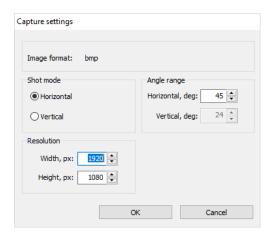
2. From the Visual channel drop-down list, select the required visual channel.



NOTE: Visual channel cannot be selected unless at least one visualization task is started.

3. In the Name field, enter the appropriate file name (if necessary).

- Click Browse... and choose the file save location. The default folder location where screenshots and videos are automatically stored is ...\NTPRO_6\ScreenShots.
- 5. To take a screenshot:
 - a. Click the Edit... button. The Capture settings window appears:



- b. Specify the following parameters:
 - Shot Mode: horizontal or vertical;
 - Angle Range viewing angle (from 1 to 360 degrees); depends on which shot mode was selected horizontal or vertical;
 - Resolution image resolution in pixels in width and height (max 20000 px).
- f. Click OK. The Capture settings window closes.
- g. Click the Take screenshot button. The image file in the BMP format is saved in the selected folder.
- 6. To record video:
 - a. When ready, click the Start recording button. The video capture starts.
 - b. When finished, click the **Stop recording** button. The video file in the MP4 format is saved in the selected folder.

3.16 Loading Visual Profile on Bridge

The **Visual Profile** panel (**View\Panels\Visual Profiles...** menu) allows the Visual Profile file to be selected and loaded on the bridge.

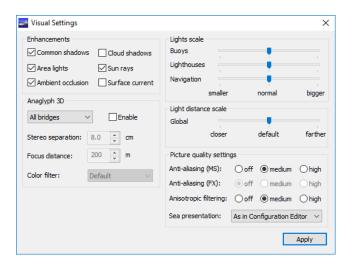
The Visual Profile is created and edited with the aid of the Visual System Tuning program (see *Navi-Trainer Professional 6. Technical Description and Installation Manual. Version 6.0*).

The Visual Profile contains information on a set of visual tasks for a group of channels, on the channel positions relative to each other, visual settings of each channel, information on the geometry adjustment, blends adjustment and colour adjustment.

3.17 Visual Settings

To set visualization parameters, use the following procedure:

1. Choose the View\Panels\Visual Settings... menu item or click the toolbar button. The Visual Settings panel appears.



- 2. In the Enchancements group box, set the following visual effects:
 - Common shadows to display shadows of objects in the exercise.
 - o Cloud shadows to display shadows of clouds.
 - Area lights to display light areas around the scene lights.
 - Sun rays to display sun rays passing through the clouds.
 - o Ambient occlusion effect.
 - Surface current to display ripple on the water surface.
- 3. In the Anaglyph 3D group box, select the Enable checkbox to switch on the Anaglyph 3D mode. From the drop-down list, select a bridge (or all bridges) where the Anaglyph 3D settings will be applied.



Adjust Anaglyph 3D mode settings:

- Stereo separation the difference in horizontal position between the right and left images. The greater the stereo separation, the greater the stereo effect. Too much separation, however, can be uncomfortable if your eyes are not adjusted to stereo.
- o Focus distance focus distance, m
- Color filter colour filter: default (red and blue) or trichrome.

- 4. In the Lights scale group box, move the sliders, to set the following effects:
 - o Buoys lights on the scene buoys;
 - Lighthouses lights on the scene lighthouses;
 - Navigation navigational lights on the ships.
- 5. In the **Light distance scale**, move the slider to set a scale of the navigational light visibility distance (default state is 6 nm).
- 6. In the Picture quality settings group box, set effects for the picture quality improvement:
 - Anti-aliasing smoothing the jagged or stair-stepped edges of the objects. You can choose the Multi-Sampled
 Anti-Aliasing (MS) and/or Fast Approximate Anti-Aliasing (FX). To switch off the effect, select the off radio
 button; to use the medium effect, select the medium radio button; to use the maximum effect, select the high
 radio button.
 - Anisotropic filtering filtering the colour noise of object surfaces. To switch off the effect, select the off radio button; to use the medium effect, select the medium radio button; to use the maximum effect, select the high radio button.
 - Sea presentation quality of the sea presentation: Standard standard quality; Simplified simplified quality (it does not require high processing power of graphics card); As in Configuration Editor the quality set in the Configuration Editor.

NOTE: Application of visualization optimizing effects requires considerable processing power of graphics card. The improvement of visualization by means of described effects is efficient for widescreen pictures produced by projectors. As for visualization applied to displays, these settings may not take such a notable effect.

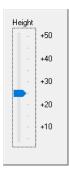
7. To apply all settings, click the Apply button.

3.18 Setting Radar Position

The simulator features a capability to perform exercise monitoring by using the radar. For the operation with the radar, the simulator configuration is required to include the appropriate applications: Radar/ARPA display.

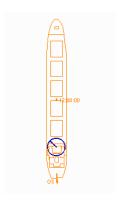
To set the radar position, use the following procedure:

- 1. Choose the Edit\Set Radar Position menu item, or click the toolbar button.
- 2. Click on the chart. The radar position will be fixed on the chart, and the screen will display the antenna elevation setting panel.



3. Set the radar elevation and click on the chart beyond the panel. The panel will disappear, whereas the radar screen will display the exercise scene observed from the set point.

The radar can be attached to any moving or stationary exercise object. To do this, at the time when you are setting the radar, click the object contour. In this case, the position of the radar will be determined by the position of the object in the exercise.



3.19 Loading ECDIS Data

There is a capability to transmit information from the trainee ECDIS to the Instructor ECDIS, which enables the Instructor:

- · to view any trainee's passage plan;
- · to view manual electronic corrections made by the Trainee;
- to overlay the trainee's manual electronic updates on the standard update prepared beforehand on the instructor workstation.

Before loading data on the Instructor's ECDIS, perform the following operations:

- 1. Install the Navi-Sailor 4000 program on the Instructor Workstation.
- 2. Set the @InstructorECSHost parameter in the Configuration Editor for the Navi Trainer Instructor task.
- 3. Start the ECS Interface task on Trainee's Bridges.

To load the ECDIS data, use the following procedure:

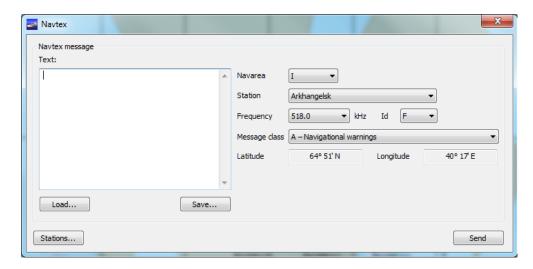
1. Click the toolbar button. The Load ECDIS data window appears:



- 2. Select the bridge which data is required to be received from.
- 3. Click the OK button.

3.20 Transmitting NAVTEX Messages

The Navtex panel is used to transmit NAVTEX messages to the own ship's ECDIS.



To open the panel, choose the View\Panels\Navtex... menu item, or click the toolbar button.

The panel contains the following controls and indicators:

- Text field for entering the text of the message by using the keyboard.
- Load... button for loading the message from the text file.
- . Save... button for saving the message in a text format.
- Navarea drop-down list for selecting the number of the navigational area.
- Station drop-down list for selecting the NAVTEX station in the navigational area.
- Frequency drop-down list for selecting the transmitting frequency of the NAVTEX message.
- Id field for indicating the identifier of the NAVTEX station.
- Message class drop-down list for selecting the identifier of the message transmitted to the NAVTEX system.
- Latitude/Longitude fields for indicating the coordinates of the NAVTEX station.
- Send button for transmitting Navtex-message to the own ship's ECDIS.
- Stations... button for calling the panel to edit NAVTEX stations in the database.

To transmit a NAVTEX message, use the following procedure:

- 1. Select the View\Panels\Navtex menu item, or click the toolbar button. The Navtex window appears.
- 2. Enter the message in the Text field.
- 3. Select the number of the navigational area from the Navarea drop-down list.
- 4. Select the NAVTEX-station from the Station drop-down list.
- 5. Select the transmitting frequency from the Frequency drop-down list.
- 6. Select the class of the message from the Message class drop-down list.
- 7. Click the Send button. The message in the NAVTEX format is transmitted to the ship's ECDIS.

To save the message in a text file:

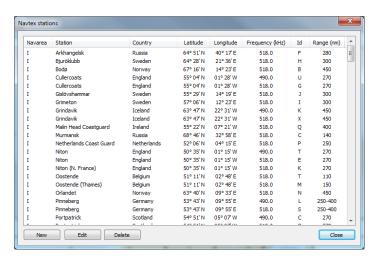
- 1. Click the Save... button in the Navtex window.
- 2. In the Save as window, select the folder to save the text file, type the file name and click the Save button.

To load messages from the text file:

- 1. Click the Load... button in the Navtex window.
- 2. In the Open window, select the text file with the message, click the Open button.

To add a new NAVTEX-station, use the following procedure:

1. In the Navtex window, click the Stations... button. The Navtex stations window with a list of all stations will appear.



2. In the Navtex stations window, click the New button. The Edit Info window appears.



- 3. In the Edit Info window, set the station parameters:
 - Navarea number of the navigational area (select from the drop-down list);
 - Station name;
 - Latitude/Longitude coordinates of the station;
 - Country;
 - Frequency transmitting frequency (kHz);
 - Id letter identifier of the station;
 - Range broadcasting range (nm).
- 4. Click the Save button. After the parameters have been saved, the new station is added to the list of stations.

To edit the station parameters:

- 1. In the Navtex stations window, select the required information from the list.
- 2. Click the Edit button.
- 3. Set new station parameter values.
- 4. Click the Save button.

To delete a station from the list:

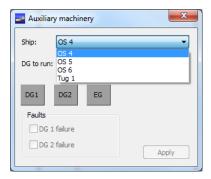
- 1. In the Navtex stations window, select the required station from the list.
- 2. Click the Delete button.

3.21 Control of Auxiliary Machinery

The Auxiliary machinery panel is designed to perform operations with an emergency power plant.

To run the diesel-generators, use the following procedure:

- 1. Before the exercise start, choose the View\Panels\Auxiliary machinery... menu item or click the button on the toolbar.
- 2. From the Ship drop-down list, select an own ship.



- 3. From the DG to run drop-down list, select the diesel generators (DG) which will be run on the simulator bridge:
 - o DG1 the first diesel generator;
 - DG2 the second diesel generator;
 - DG1 + DG2 the first and second diesel generators.

If the Trainee starts or stops the generators, the respective indicators will light up on the **Auxiliary machinery** Instructor's panel:

- DG1 DG 1 is running;
- DG2 DG 2 is running;
- EG emergency DG is running. The indicator lights up when both generators fail, or if the Trainee does not start
 another generator on time when one generator fails.

To set the generator fault, use the following procedure:

- 1. After the exercise start, choose the View\Panels\Auxiliary machinery... menu item or click the button on the toolbar.
- Select the DG 1 failure checkbox and/or the DG 2 failure checkbox and click the Apply button. At the same time, an alarm signal is triggered on the simulator bridge (Instrum\EPC page).



3. To cancel the generator fault, clear the checkboxes on the Auxiliary machinery panel and click the Apply button.

The following Trainee actions are recorded on the **Events** panel (**View\Panels\Events...** menu):

- start of diesel generator DG1, 2;
- stop of diesel generator DG1, 2;
- automatic stop of diesel generator DG 1, 2;
- automatic start of emergency diesel generator EG.

3.22 Control of Remote Sensors

The remote sensor can be set in any point on the chart and attached to any object. The remote sensor is used to monitor the following parameters:

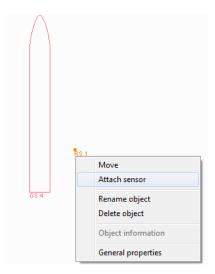
- · Latitude;
- Longitude;
- ROT (rate of turn);
- · Velocity;
- · Depth;
- · Current speed;
- Current direction;
- True wind speed;
- True wind direction.

To set the remote sensor, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Reference category, select the Remote sensor object and click the Finish button.
- 3. Left click on the chart.

To attach the remote sensor to any moving or stationary object:

1. Open the sensor's context menu and choose the Attach sensor menu item.

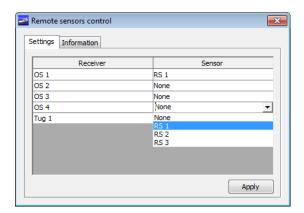


2. Click the required object. The sensor will observe the position relative to the object and move with the object.

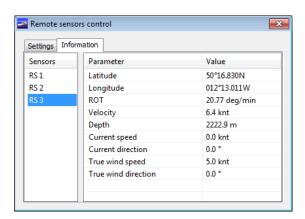
To detach the sensor form the object, open the context menu of the remote sensor and choose the **Detach sensor** menu item.

To control the remote sensors, use the following procedure:

- 1. Choose the View\Panels\Remote sensors... menu item or click the toolbar button. The Remote sensors control panel appears.
- 2. On the **Settings** tab, for each receiver (own ship) select the sensor which will transmit data to the **Remote Sensor** IBID panel. Click the **Apply** button.



3. To view the sensor parameters, open the **Information** tab.

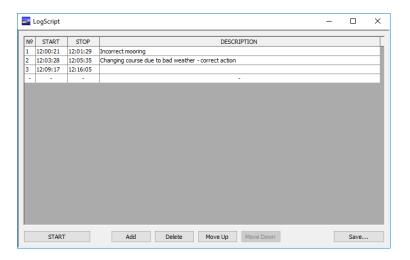


3.23 Creating Log Scripts

The log scripts can be created in different ways:

- by using the LogScript panel. In this case, the Instructor can record any fragments of the exercise and place them
 in the necessary order for the subsequent playback;
- by using any text editor. In this case, the Instructor can write a log script in the txt format. The extension of the log script file should be *.ntlogsc. The log-script can contain one or several log files; the log script and log files used in it should be located in one folder.

To open the LogScript panel, choose the View\Panels\LogScript... menu item, or click the loolbar button.

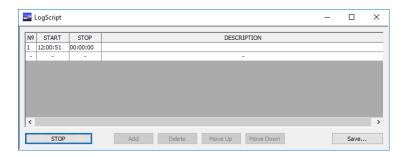


The LogScript panel contains the following information:

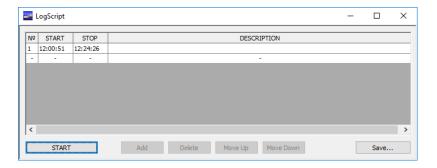
- № number of the exercise fragment (shows the fragment playback order);
- Start exercise fragment start time;
- Stop exercise fragment stop time;
- Description exercise fragment description (the text appears on the visualization screen during the log script playback).

To record exercise fragments to a log script, use the following procedure:

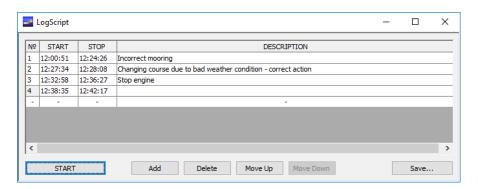
- 1. After the exercise start, choose the View\Panels\LogScript... menu item, or click the button.
- 2. To start recording of the first exercise fragment, click the **Start** button. The fragment number and start time will appear in the first row of the table.



3. To end the recording of the first fragment, click the **Stop** button. The first fragment stop time will be added to the row.



- 4. If necessary, add the description of this fragment in the **Description** column. When this log script fragment is played back, the screen will display the entered text.
- 5. Repeat steps 2-4 to record other fragments to the log script.



- 6. To add a fragment to the script, click the Add button. The new fragment's row appears.
- 7. To move a fragment up, select it in the table and click the Move Up button.
- 8. To move a fragment down, select it in the table and click the Move Down button.
- 9. To delete a fragment, select it in the table and click the **Delete** button.
- 10. To manually change the fragment start and/or stop time:
 - a. Click the required table cell.
 - b. Enter the necessary time (the stop time should not be less than the start time) and click.
- 11. To save the current log script: click the Save... button, enter the log script name in the File name field, and click the Save button. The log script file will be saved with the *.ntlogsc extension.

NOTE: To play back the log-script, see "Log Script Playback" on page 274.

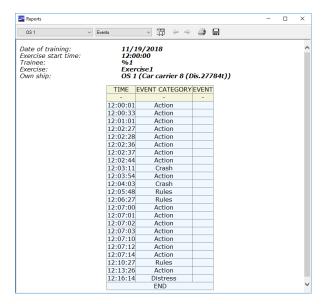
4. Debriefing

4.1	General	270		
4.2	Loading Log File	271		
4.3	Log Playback	272		
	Log Script Playback			
4.5	.5 Exercise Window Printout			
	6 Playback of Audio Log Files			
4.7	Playback of Video Log Files	279		
4.8	Exercise Report Generation			
	4.8.1 Report Generator Wizard	280		
	4.8.2 Report Setup Wizard	283		
	4.8.3 Report Settings	285		
	4.8.4 Reports Panel	285		

4.1 General

Debriefing presumes the playback of an exercise record (log) and generation of different reports for post processing and analysis.

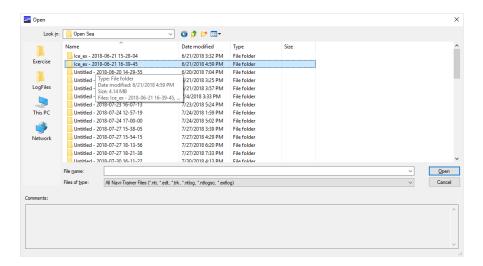
The exercise report is formed in accordance with the selected template. The report can be printed out or saved to a file in the Excel (*.csv) format.



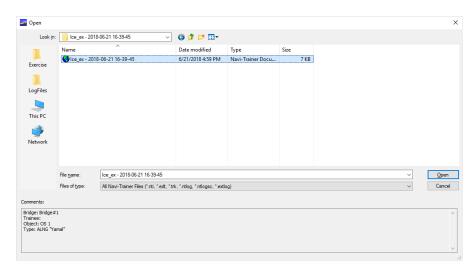
4.2 Loading Log File

To load a log file, use the following procedure:

- 1. Choose the File\Open... menu item, or click the toolbar button, or press the <Ctrl>+<O> key combination. The Open window appears. By default, all log files are stored in the NTPRO_6...\LogFiles folder.
- 2. Open the LogFiles folder.
- 3. Open the area folder. The list of log file folders appears. The folder name contains: exercise name, date and time.



- 4. Select the log files folder. The log file with . *extlog extension appears.
- 5. Click on the log file name to show comments in the bottom part of the window.



6. To load a log file, double click on its name or click the **Open** button. The **Open** dialogue box will close down. The log file window will appear, with a icon in the top left corner.

NOTE: In the Open window, you can perform standard operations on files (deleting, renaming, copying, etc.).

4.3 Log Playback

The log-file contains a recording of exercise from the time of the object assignment to the bridge until the time of stopping the exercise.

The procedure for running the log playback is similar to the exercise running procedure.

To play back a log on the Instructor workplace, click the toolbar button, or press the <Ctrl>+<2> key combination, or choose the File\Control\Start menu item.

To play back a log on the bridge, use the following procedure:

- 1. Click the toolbar button, or press the <Ctrl>+<1> key combination, or choose the File\Control\Assign menu item in main menu.
- 2. From the Bridge drop-down list, select the bridge name.
- 3. From the Object drop-down list, select the object name.
- 4. In the Trainee field, enter the Trainee name (optional).
- 5. Click the Add button.
- 6. After the distribution has been completed, click the **OK** button. The window will disappear, and the initial data loading procedure will start.
- 7. Click the toolbar button, or press the <Ctrl>+<2> key combination, or choose the File\Control\Start menu item.

To stop the log playback, click the toolbar button, or press the <Ctrl>+<4> key combination.

To change the log playback start time:

- 1. Click the toolbar button or press the <Ctrl>+<3> key combination.
- 2. Move the slider on the time-bar, or use the spin buttons, or enter start time in the spin box at the right-side.



3. Click the toolbar button again.

To change the time scale of the log playback, open the toolbar drop-down list and select the time scale: 1:20,1:10, 1:5, 1:2, 1:1, 2:1, 5:1, 10:1, 20:1.



The log's time bar contains time markers that divide the log's time bar into segments:

- Play marker (green) indicates starting of an active session period;
- · Pause marker (black) indicates starting of a pause period;
- · Rewind marker (red) indicates rewinding starting.



NOTE: To show exercise pause periods during the log playback, make sure the **Show log pauses** option is enabled on the **Misc** tab of the **Settings** panel (View\Settings\Misc).

For example, let's consider a log's time bar as shown in the figure below. In this case, the log's time bar is divided into eight segments.



The table below contains the description of log's time bar segments.

Table 13 . Description of log's time bar

SEGMENT №	DESCRIPTION
1	time after assignment an exercise to the bridge and before the exercise start
2, 4, 6, 8	active session periods (light green colour of the segment)
3, 7	pause (dark gray colour of the segment)
5	pause for rewinding (dark gray colour of the segment)

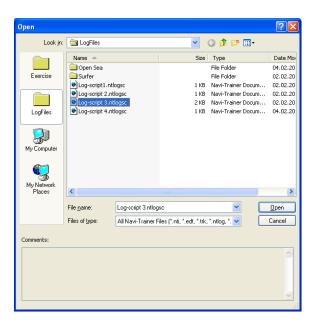
4.4 Log Script Playback

The Instructor can load the log script and play it back on the Instructor workplace or on the simulator bridge.

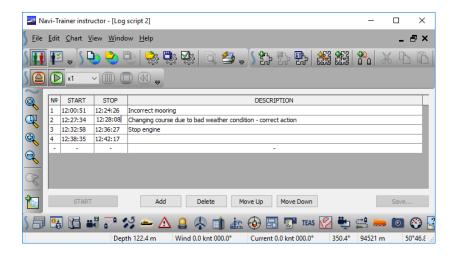
NOTE: To create a log script, see "Creating Log Scripts" on page 267.

To load a log script and start its playback, use the following procedure:

- 1. Choose the File\Open... menu item, or click the button.
- 2. In the Open window, select a folder in which the log script was saved (the default is the LogFiles folder).
- 3. Select the necessary file (the file should have the *.ntlogsc extension).

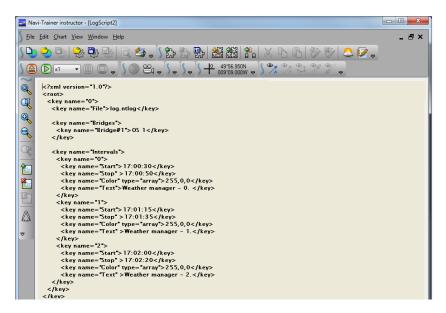


4. Click the Open button. The log script window appears.



NOTE: If necessary, edit the log script and save it by clicking the **Save...** button. The **Start/Stop** button is disabled during the editing. The log script editing procedure is similar to the log script editing during the simulator session (see "Creating Log Scripts" on page 267).

If the log script has been created in the text editor, the text of the log script is displayed:



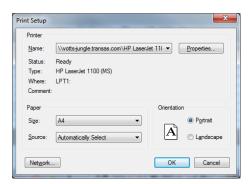
- 5. To start the log script playback on the Instructor workplace, click the toolbar button.
- 6. To start the log script playback on the bridge, click the and toolbar buttons.
- 7. To stop the log script playback, click the toolbar button.

4.5 Exercise Window Printout

During the log file playback, the Instructor can print out the exercise window at any time.

To set the page parameters, use the following procedure:

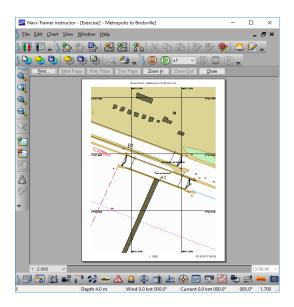
1. Choose the File\Page Setup... menu item. The Print Setup window appears.



- 2. In the **Printer** group, set the printer name and properties.
- 3. In the Paper group, select the page size and the type of feeding sheets to the printer.
- 4. In the **Orientation** group, select the page orientation.
- 5. Click the OK button.

To preview document, use the following procedure:

1. Choose the File\Print Preview menu item or click the late toolbar button.

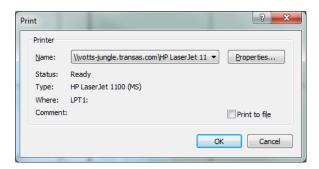


NOTE: The page bottom margin contains the current scale of the chart, date and the page printout time. The top margin shows the exercise name. For a log-file, the exercise creation time and date are also displayed in the top margin.

- 2. Click the Zoom In/Zoom Out button to zoom the image in/out.
- 3. To print out the page, click the **Print** button.

To print out the main exercise window, use the following procedure:

1. Choose the File\Print... menu item or press the <Ctrl>+<P> key combination. The Print window appears:



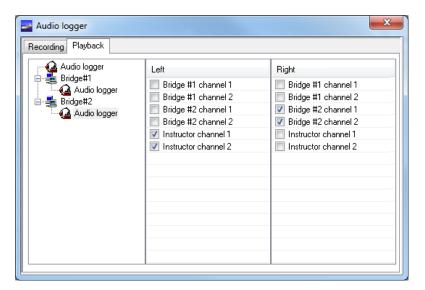
- 2. Set up the window printout settings as required.
- 3. Click the OK button.

For the prompt printout of the main exercise window, click the at toolbar button The document will be printed out on the selected printer with the current settings.

4.6 Playback of Audio Log Files

Exchange can be played back on any simulator configuration computer with a sound card and Audio logger application. To start the exchange playback, use the following procedure:

- 1. Choose the View\Panels\Audio logger... menu item or click the toolbar button.
- 2. Open the Playback tab of the Audio logger panel.
- 3. In the configuration tree, select the Audio Logger task which will be used for playback.
- 4. Select records for the right and left channels, by selecting necessary checkboxes.

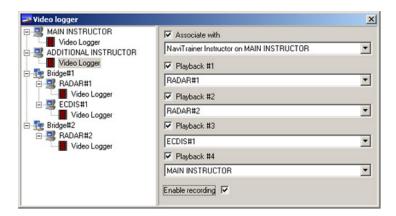


NOTE: The exchange records will be played during the exercise log playback. Left and right channels will reproduce the sum of selected records.

4.7 Playback of Video Log Files

Video log files can be played back on any computer included in the simulator configuration where the Video Logger task is run. To turn on playback of the log file recorded during the exercise, use the following procedure:

- 1. On any instructor station, load the log file.
- 2. Choose the View\Panels\Video logger... menu item, or click the toolbar button.
- 3. From the tree in the left-hand part of the panel, select the **Video Logger** task which will be used for playing back video recordings.
- 4. In the right-hand part of the panel, select the **Associate with** checkbox and from the drop-down menu select that instructor station where the log file is loaded.
- 5. Select the tapes which will be played back by the previously selected Video Logger task. To do this:
 - a. Select the tape number (Playback #) by use the checkbox.
 - b. Select from the drop-down list the name of the computer where the tape was recorded.



- 6. Start playing back the log file on the Instructor station.
- 7. On the computer which was selected switch to the Video Logger task by using the Taskbar or <Alt>+<Tab>key combination.
- 8. Click on the picture which you wish to zoom. The second click will return all the pictures on the screen.

The computer which was selected will play back all selected tapes, each in its part of the screen, or one of them on the entire screen.

NOTE: The maximum number of tapes played back on one screen, is 4. If less than 4 tapes are played back, it is allowed to add new ones without stopping playback of the main log file.

Playback will be performed in the course of playing back the main exercise log file responding to the changes of acceleration ratio, to the **Pause** command, and to leaps from one event to another (**Events** panel).

4.8 Exercise Report Generation

Exercise report generation is the export of different model parameters to the Excel format for post-processing and analysis.

The simulator has a capability to generate the following standard report types:

- Ship dynamics contains the main ship motion parameters;
- Events contains a list of events which took place during the simulator session;
- Forces contains forces and moments affecting the ship;
- Log contains an extended list of the ship motion parameters;
- Towing contains main parameters of tow lines;
- · Traffic contains the main parameters of the target motion;
- Tugs contains forces and moments from tugboats acting on the ship;
- Virtual forces the report (in a tabular form) contains components of external forces and moments affecting the ship.

Exercise report generation is performed by the Report Generator Wizard application. The Report Setup Wizard application is designed for creating report settings templates.

Also, the user can form a simple report from the log-file by using the Reports panel (menu View\Panels\Reports...).

4.8.1 Report Generator Wizard

The Report Generator Wizard application is designed for exercise and log-file report generation in the CSV-format.

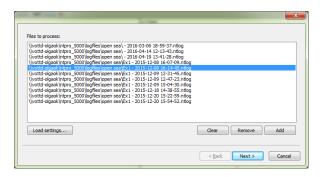
The Report Generator Wizard allows:

- a set of exercises to be started and played automatically without assigning to a bridge. The duration of playback can be defined for each exercise.
- forming a report for an exercise or a log file. The report contains a table or graphic display of log file data.
- selecting a report type: Events, Forces, Log, Ship dynamics, Training and other. An individual file is created for every ship and report type.
- · loading report settings templates:
 - o templates created in the Report Setup Wizard (see "Report Setup Wizard" on page 283).
 - o templates created in the Diagram window (see "Diagram Settings Template" on page 243).
- Saving a report in the CSV-format to any folder.

To use the Report Generator Wizard, perform the following steps:

- 1. Run the Navi-Trainer 6\Report Generator Wizard application.
- 2. Add log-files and an exercise for processing:
 - a. Click the Add button.
 - In the Open window, select the necessary exercise and/or log files. To select several files at once, hold the <Shift> button. Click the Open button.
 - c. To remove a file, select it and click Remove button. To remove all files, click the Clear button.

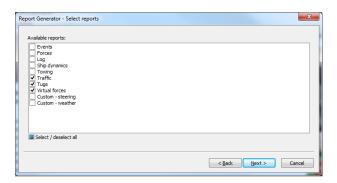
3. If necessary, click the Load settings... button to load a previously created report settings template.



- 4. Click the Next button.
- 5. Make the following report settings:
 - Time step time interval between the recordings of the report data.
 - Output directory folder where the reports will be saved. By default, the reports are saved in the NTPRO_ 6\Nti\Export folder. To select another folder, click the Browse button.
 - Maximum running time exercise running time. When the time expires, the exercise is automatically stopped.

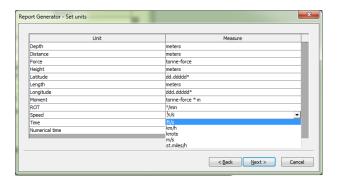


- 6. Click the Next button.
- 7. To choose the desired report type, select the checkbox next to its name. To choose all report types, select the Select\deselect all checkbox.

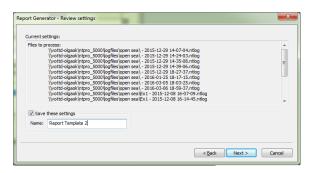


NOTE: The list of report types contains standard report types (Events, Forces, Log, Ship dynamics and others) and Custom report types added from the Diagram templates. Please note that parameters in the Diagram template may be specific for a particular ship; if another ship does not have these parameters, they will not be displayed in the output file.

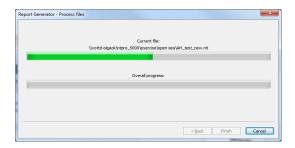
- 8. Click the Next button.
- 9. In the Measure column, select units of measure from the drop-down lists.



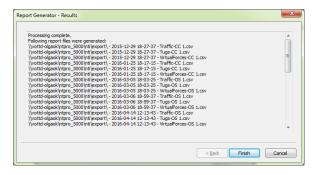
- 10. Click the Next button.
- 11. To save the report generator current settings as a template, select the **Save these settings** checkbox, then enter the name of the settings template in the **Name** field.



12. Click the Next button. The file processing will begin.



13. After the successful file processing, click the Finish button.



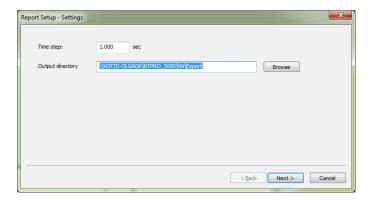
A report file in the CSV format will be created for every ship and selected report type. All report files will be saved in the selected folder.

4.8.2 Report Setup Wizard

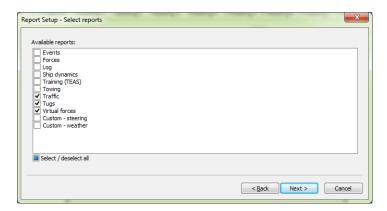
The Report Setup Wizard application is designed for creating report settings templates.

To create the report settings template, use the following procedure:

- 1. Run the Navi-Trainer 6\Report Setup Wizard application.
- 2. Make the following report settings:
 - Time step time interval between the report data recordings.
 - Output directory folder where the reports will be saved. By default, the reports are saved in the NTPRO_ 6\Nti\Export folder. To select another folder, click Browse.



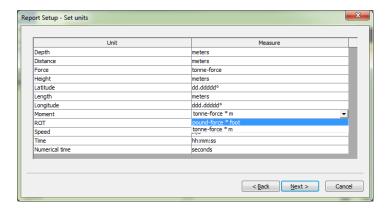
- 3. Click the Next button.
- 4. Select report types by using the checkboxes. To choose all report types, select the Select\deselect all checkbox.



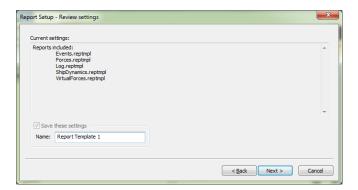
NOTE: The list of report types contains standard report types (Events, Forces, Log, Ship dynamics and others) and Custom report types added from the Diagram templates. Please note that parameters in the Diagram template may be specific for a particular ship; if another ship does not have these parameters, they will not be displayed in the output file.

5. Click the Next button.

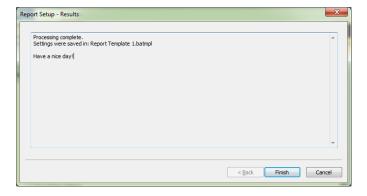
6. In the Measure column, select units of measure from the drop-down lists.



- 7. Click the Next button.
- 8. Enter a template name.



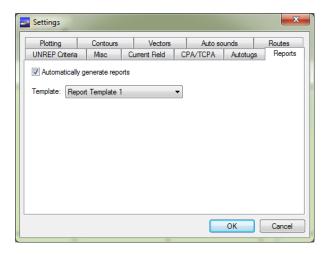
- 9. Click the Next button.
- 10. Click the Finish button. Settings have been saved in the *batmpl file.



4.8.3 Report Settings

To automatically generate a report after stopping an exercise, adjust the following settings:

- 1. Before the start of the exercise, choose the View\Settings menu item and open the Reports tab.
- Select the Automatically generate reports checkbox.
- 3. From the **Template** drop-down list, select the report settings template.



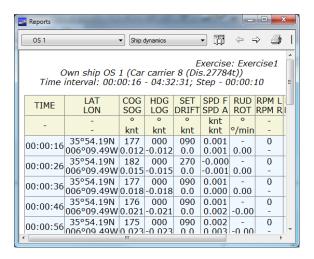
NOTE: The report settings template can be created in the Report Setup Wizard, Report Generator Wizard and on the Diagram panel.

4. Click **OK**. After stopping any exercise, the report file is automatically saved in the output folder that was specified in the report settings template.

4.8.4 Reports Panel

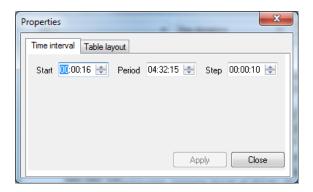
The Reports panel is used for generating simplified reports. To do this:

- 1. Open the exercise log file.
- 2. Choose the View\Panel\Reports menu item. The Reports panel appears.



- 3. From the list of ships, select the ship which you wish to generate the report for.
- Select a report type from the list of reports.

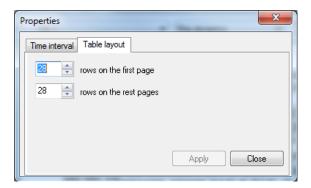
5. To open the report properties panel, click the panel. button. The screen will display the report parameters setting panel.



- 6. On the Time interval page set the following parameters:.
 - Start- report start time counted from the beginning of the exercise.
 - Period report duration.
 - Step report time step.
- 7. Click the Apply button.

NOTE: If the set parameters go beyond the set log file duration, the screen will display a pertinent warning.

8. On the **Table layout** page set the size of the table displayed on the report pages (number of rows on the first and and the rest of pages).



- 9. Click the Apply and Close buttons.
- 10. To view the report, use arrow buttons on the Reports panel.
- 11. To print out the report, click the **Print** button on the **Reports** panel.
- 12. To save a tabular report, click the **Save** button on the **Reports** panel. The report will be saved in the specified folder in the Excel format (comma separated values).

NOTE: The format allows the reports to be viewed in the form of electronic Microsoft Excel tables.

5. Special Operations

5.1	Mooring Operations		289
	5.1.1	Mooring Walls	289
	5.1.2	Fender Protection	289
	5.1.3	Operations with Tow/Mooring Lines	293
	5.1.4	Control of Mooring Winches	296
	5.1.5	Setting Mooring Objects	297
	5.1.6	Operations with Inflatable Fenders	304
5.2	Opera	ations with Anchors	306
	5.2.1	Setting Anchor Properties	306
	5.2.2	On-Chart Control of Anchors	307
	5.2.3	Anchor Control During Exercise	308
5.3	Towing Operations		
	5.3.1	Setting Bridle	310
	5.3.2	Creating Barge Tow	315
	5.3.3	Operations with Tow Winches	316
5.4	Towing in Inland Waterways		
	5.4.1	Articulated Tug Barge	322
	5.4.2	Pusher-Barge Train	324
	5.4.3	Breaking Connection	
	5.4.4	Attaching Remote Sensor to Barge	327
	5.4.5	Barge Navigational Signals	328
	5.4.6	Applying Assist Force	329
5.5	Opera	ations with Automatic Tugs	330
	5.5.1	Automatic Tug General Settings	331
	5.5.2	Tug Reserving and Releasing	331
	5.5.3	Basic Operations	333
	5.5.4	Automatic Tug Control Page	338
5.6	Pilot E	Boat Operations	339
	5.6.1	Pilot Boat Maneuvers	339
	5.6.2	Pilot Transfer	341
5.7	Operations in Ice Conditions		344
	5.7.1	Using Ice Object	345
	5.7.2	Using Ice Generator Object	351
	5.7.3	Control of Ice Resistance	361
	5.7.4	Setting Ice Accretion	
	5.7.5	Ice Channel Evaporation Visual Effect	362
	5.7.6	Operations with Icebergs	362
	5.7.7	Monitoring Trainee Performance	365
5.8	Fishin	ng Operations	
	5.8.1	Shoal Objects	366
	5.8.2	Pinnacle Objects	368
	5.8.3	Fishing Buoys	370

	5.8.4	Setting Trawl	. 371
	5.8.5	Setting Purse Seine	374
	5.8.6	Setting Long Line	376
	5.8.7	Watching Trainee Performance	378
5.9	Search and Rescue Operations		
	5.9.1	SAR Objects	. 379
	5.9.2	Flag Signalling	390
	5.9.3	Giving Distress Signals	. 393
	5.9.4	Use of Rescue Boat	. 394
	5.9.5	Life Raft Operations	. 397
	5.9.6	Using Lifesling for MOB Recovery	. 399
5.10	Opera	tions with Helicopters and Aircrafts	403
	5.10.1	Helicopter Initial Parameters	.403
	5.10.2	Helicopter Control	405
	5.10.3	Operations with Aircraft	. 409
5.11	Under	way Replenishment Operations	410
	5.11.1	Setting Underway Replenishment Criteria	.410
	5.11.2	Underway Replenishment Control Panel	. 411
	5.11.3	P&D Line and Span Wire Rendering/Heaving	.412
5.12	Joint S	Ship Maneuvering in Formations	415
	5.12.1	Creating Fleet Formation	.415
	5.12.2	Fleet Formation Management	417
5.13	Passir	ng of Locks	419
	5.13.1	Setting Lock Initial Properties	.419
	5.13.2	Control of Lock	.420
	5.13.3	Monitoring Fulfilment of Exercise with Locks	.424
5.14	Contro	ol of Semaphores	425
5.15	Ro-Ro	Operations	426
5.16	ECDIS	STraining	.430
	5.16.1	Ordering Charts, Chart Updates and Weather Forecasts	430
	5.16.2	Order Processing	. 433
5.17	FOS D	ata Import	435
5.18	Opera	tions with Submarine	442
5.19	Pirate	Attacks	.444
5.20	Opera	tions with Moveable Bridges	.445
5.21	GNSS	Control	.447
5.22	Contro	ol of Motion Platforms	.449
	5.22.1	Control of Motion Platforms via UHI Task	. 449
	5.22.2	Control of Motion Platforms via MPI Task	. 451

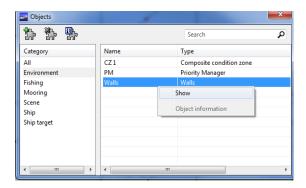
5.1 Mooring Operations

5.1.1 Mooring Walls

The actual edge of the mooring wall does not always coincide with the coastline on the exercise chart. To view actual borders of mooring walls, it is useful to switch on the display of mooring walls and piers on the exercise chart.

To show mooring walls on the chart, use the following procedure:

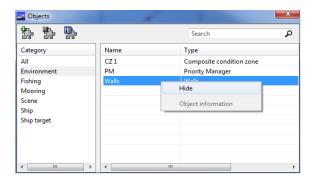
- 1. Choose the View\Panels\Objects... menu item.
- 2. On the Objects panel, select the Walls object from the Environment category.
- 3. Right-click on the object. A context menu appears.



4. Choose the Show menu item. Contours of mooring walls on the chart will be shown as a red line.



To hide mooring walls, choose the **Hide** menu item.



5.1.2 Fender Protection

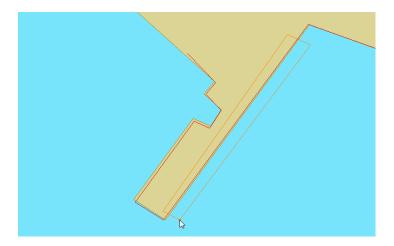
Fender zones are used for fender protection of mooring walls.

NOTE: Before setting a fender zone, the mooring walls should be shown on the chart (see "Mooring Walls" on page 289).

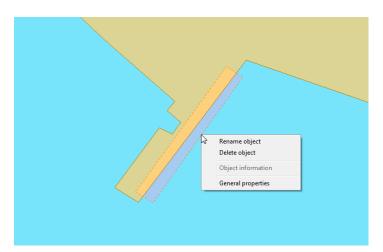
Fender Zone

To set a fender zone, use the following procedure:

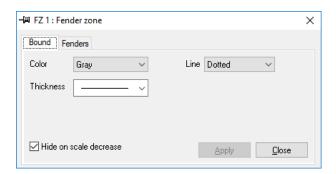
- 1. Before the exercise start, choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Fender zone object and click the Finish button.
- 3. Draw a fender zone along the mooring wall.
- 4. Right-click to complete drawing.



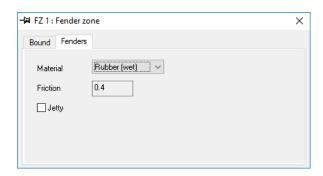
5. Right-click on the zone boundary and choose the **General properties** context menu item.



- 6. On the **Bound** tab, adjust the zone boundary presentation:
 - a. Set the colour, thickness and line type of the zone boundary.
 - b. If necessary, clear the **Hide on scale decrease** checkbox to show the zone contour when the chart scale is decreased.



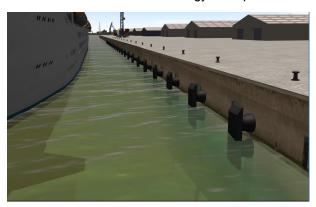
- c. Click the Apply button.
- 7. On the Fenders tab, select fender properties:
 - a. From the Material drop-down list, select fender material; the friction coefficient appears in the Friction field.



- b. If necessary, select the **Jetty** checkbox to disable the effect of the hydrodynamic interaction with mooring walls.
- c. Close the window.

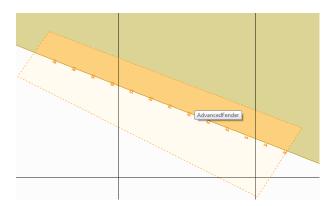
Advanced Fender Zone

The advanced fender zone contains mooring fenders simulated as separate objects attached to the mooring wall, with correct reaction force and energy absorption.

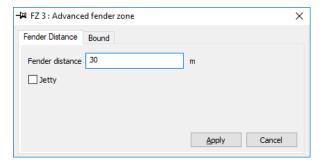


To set the Advanced fender zone, use the following procedure:

- 1. Before the exercise start, choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Advanced fender zone object and click the Finish button.
- 3. Draw a fender zone on the chart. The mooring wall should be within the boundary of the fender zone.



- 4. Right-click on zone boundary and choose the General properties context menu item.
- 5. On the Fender Distance tab, set the zone properties:
 - a. In the Fender distance field, set the distance between two adjacent fenders (from 4 to 100 meters).



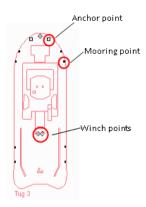
- b. If necessary, select the Jetty checkbox to disable the effect of hydrodynamic interaction with mooring walls.
- c. Click the Apply button.
- 6. On the Bound tab, adjust the zone boundary presentation:
 - a. Set the colour, thickness and line type of the zone boundary.
 - b. If necessary, clear the **Hide on scale decrease** checkbox to show the zone contour when the chart scale is decreased.
 - c. Click the Apply button.

5.1.3 Operations with Tow/Mooring Lines

5.1.3.1 Line Attachment Points

The following line attachment points are displayed on the ship contour:

- Mooring point point in the form of a small filled square on the ship contour. From the point you can give a line and attach it to another object's mooring point. The mooring point has the properties panel for setting the properties of the line and mooring winch (for more information, see "Setting Mooring Point Properties" on page 293).
- Anchor point point in the form of an empty square on the ship contour. From the point you can drop the anchor (for more information, see "On-Chart Control of Anchors" on page 307).
- Winch point point in the form of an empty rhombus on the ship contour (these points are available for some own/target tug objects). From the point you can give a line and attach it to any mooring point of another object. The mooring point has the properties panel for setting properties of the line and mooring winch (for more information, see "Setting Winch Point Properties" on page 316).



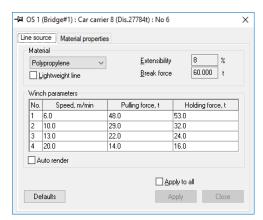
5.1.3.2 Setting Mooring Point Properties

To set properties of a mooring point, use the following procedure:

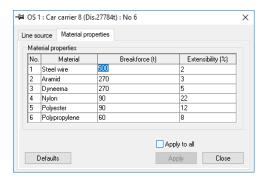
1. Before the exercise start, increase the chart scale so that the mooring points become visible on the object contour.



- 2. Right click on the mooring point.
- 3. Choose the Properties context menu item. The screen will display a window with open Line source tab.



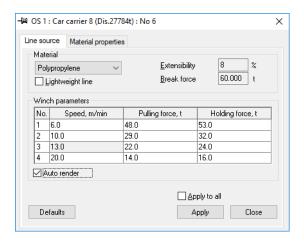
- 4. From the **Material** drop-down list, select the line material: steel wire, aramid, dyneema, nylon, polyester, polypropylene. The selected material type will determine the line extensibility, break force and winch parameters.
- 5. If necessary, set the weightless line by selecting the Lightweight line checkbox.
- 6. To change the line break force value:
 - a. Open the Material properties tab.



- b. In the **Breakforce** column, enter a new value for the required material type, press the **Enter** key and click the **Apply** button.
- To apply settings to all the mooring points of the ship, select the Apply to all checkbox and click the Apply button.
- d. To return to the initial settings, click the **Defaults** button.
- e. To continue editing of line properties, open the Line source tab.
- 7. Set the winch parameters in the Winch parameters table on the Line source tab:
 - a. In the Speed column, enter four winch speed values.
 - b. In the Pulling force column, enter the winch pulling force values for four winch speeds.
 - c. In the Holding force column, enter the holding force values for four winch speeds.

NOTE: The Pulling force value should be smaller than the Holding force, otherwise there is winch slipping. The Holding force value should be smaller than the Break force value.

d. If necessary, select the Auto render checkbox to automatically pay out the line at large loads.



- 8. Select the Apply to all checkbox if the specified properties should be applied to all the mooring points of the ship.
- 9. Click the Apply button to save new settings, or the Defaults button to return to the initial settings.

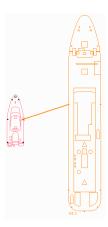
5.1.3.3 Handling Lines

To give and make fast the line, use the following procedure:

- 1. Increase the display scale so that mooring points become visible on the object contour.
- 2. Position the cursor on the object mooring point. A prompt containing the mooring point number appears:



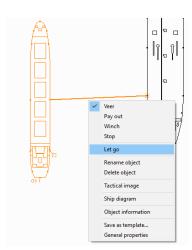
3. Click the object mooring point and then click the another object's mooring point. The line is made fast.



To give the line with free end, click the object mooring point and then click on the place beyond the object contour.



To let go the line, right-click the line and choose the Let go context menu item.

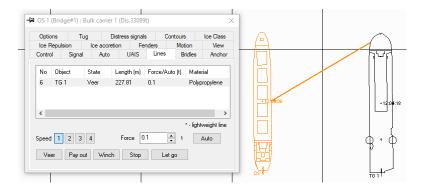


5.1.4 Control of Mooring Winches

During the exercise the mooring winches can be controlled from the **Lines** tab of the own ship or target ship properties panel.

To control the winches, use the following procedure:

Open the object properties panel and select the Lines tab. The screen will display the Lines tab which contains
the table and the winch control buttons.



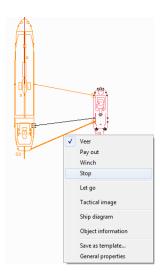
The table shows the following information:

- No mooring point number;
- Object object to which the line is given;
- State winch status (Veer, Winch, Pay out, Stop);
- Length line length, m;
- Force/Auto line tension force/, tons;
- Material line material (* a lightweight line is selected).
- 2. Select a mooring line in the table and click one of the winch control buttons:
 - Veer to pay out the line under the effect of an external force;
 - Pay out to pay out the line by the winch motor (not for all the ship models);
 - Winch to heave the line;
 - Stop to stop a winch, the line length is fixed.
- 2. To maintain the tension force of the line in the automatic mode:
 - a. Select the mooring line in the table.
 - b. In the Force field, set the line tension force value.
 - c. Click the Auto button. In the table, the winch status indicator shows the "Tension Control" state.



- d. To cancel the Tension Control mode, click one of the winch control buttons: Veer, Winch, Pay out or Stop.
- 4. If it is necessary to give a command for several lines at a time, press the <Ctrl> or <Shift> key and select several rows in the table. After a time, the winch will be operating in accordance with the selected command. The table will show the change of data.
- 5. To switch the winch speeds, use the 1234 buttons: where 1 the slowest winch speed, and 4 the fastest winch speed. This option is not available for all models.
- 6. To let go a line, select the relevant row in the table and click the Let go button.

The winch state can be selected from the line context menu that contains the following commands: Veer, Pay out, Winch, Stop, Let go.



When the cursor is pointed at the line, the prompt shows the current line state.



Information on the Trainee line handling performance is displayed on the Events panel.

5.1.5 Setting Mooring Objects

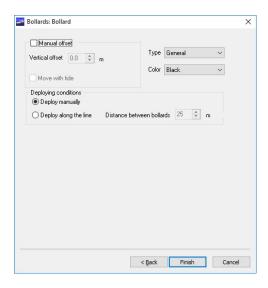
This section describes the setting of such objects as:

- Bollard (see "Bollards" on page 298);
- CALM and SALM (see "Mooring Buoy (CALM, SALM)" on page 299);
- Floating dock (see "Floating Dock" on page 300);
- Floating fender (see "Floating Fender" on page 300);
- Laser Docking System (see "Laser Docking System" on page 301);
- Oil platform (see "Oil Platforms" on page 303);
- SPM (see "SPM Object" on page 304).

5.1.5.1 **Bollards**

To set bollards, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Bollards object and click the Next button. The Bollard tab appears.



- 3. In the Type drop-down list, select the bollard type: General (default), Double, Small.
- 4. In the Color drop-down list, select the bollard colour: black (default), white, red, blue, green, yellow.
- 5. To set the bollard offset relative to the sea level, select the **Manual offset** checkbox and specify the **Vertical offset** value (0-99 meters). For the bollard to move with tide, select the **Move with tide** checkbox.
- 6. Select the bollard deploying conditions:
 - Deploy manually to set each bollard manually.
 - Deploy along the line to set some bollards along the line. In the Distance between bollards field, specify the
 distance between neighboring bollards.
- Click the Finish button and click on the chart.

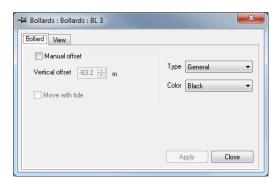


To open the list with all bollards which have been set on the chart:

1. Open the context menu of any bollard and choose the **General properties** menu item.



- 2. To delete a bollard: select a bollard from the list, click the **Delete** button, and click **OK** to confirm the object deleting.
- 3. To open the bollard properties panel, make a double mouse click on the required bollard from the list. The object will be placed in the centre on a chart, and the object properties panel appears.



5.1.5.2 Mooring Buoy (CALM, SALM)

The simulator handles two types of mooring buoys: CALM and SALM. To set a mooring buoy, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the CALM or SALM object, and click the Next button.
- 3. In the Orientation field, specify the buoy orientation.
- 4. In the Anchor count field, specify the number of anchors for the ship mooring (from 1 to 10).



- 5. Click the Finish button.
- 6. Click on the chart. The object is fixed on the chart.



5.1.5.3 Floating Dock

The Floating dock object can be used as a mooring object. To set the object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Floating dock object, and click the Next button.
- 4. In the Orientation field, specify the object orientation.
- 5. In the **Depth** field, specify the object immersion (0 20 meters).
- 6. Click the Finish button.
- 7. Click on the chart.



5.1.5.4 Floating Fender

During the ship berthing alongside the wall or other ships, an object representing a floating fender model can be used.

To set the object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Floating fender (6 m/9 m) object, and click the Next button.
- 3. In the Orientation field, specify the object orientation.
- 4. Click the Finish button.
- 5. Click on the chart. The object will be fixed on the chart.



NOTE: If the line is given from the fender to the ship, its length will always be fixed. If the line is given from the ship to the fender, it will be controlled just like any other line. Floating fenders can be connected with each other.



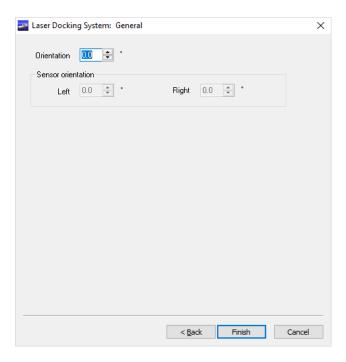
5.1.5.5 Laser Docking System

The exercise can be supplemented with a Laser Docking System which measures distance from the wall to the ship in the process of mooring.

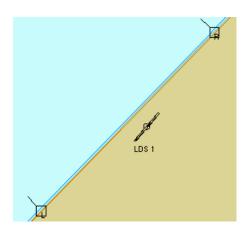
The system consists of two laser sensors and a large digital display board. The laser sensors are placed on the edge of the mooring wall, so it is advisable to display mooring wall contours on the exercise chart. The range of laser sensor is 200 meters.

To set an object on the exercise chart, perform the following procedure:

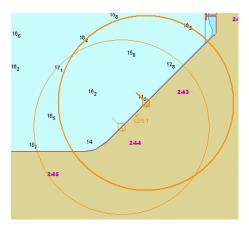
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Laser Docking System object and click the Next button.
- 3. In the Orientation field, set the display board orientation and click the Finish button.



- 4. Click on the chart to specify the place where the display board should be installed.
- 5. Click on the mooring wall to specify the place where the first laser sensor should be installed.
- 6. Click on the mooring wall to specify the place where the second sensor should be installed:



Circular areas on the chart show ranges of the laser sensors (circular areas are shown if the sensor or display board is selected on the chart):

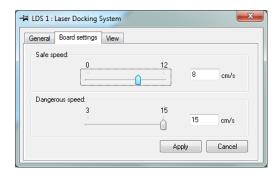


To change properties of the object:

- 1. Open the object context menu and choose the **General properties** item. The object properties panel appears.
- 2. On the General tab:
 - a. In the Orientation field, set the display board orientation.
 - b. In the Sensor orientation group box, set the orientation of the left and right sensors (Left and Right fields).
 - c. Click the Apply button.

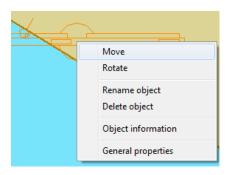


- 3. On the Board settings tab:
 - a. In the **Safe speed** group box, set the safe speed value using the slider or enter this value in the field next to the slider.
 - b. In the **Dangerous group** box, set the dangerous speed value using the slider or enter this value in the field next to the slider.
 - c. Click the Apply button.



NOTE: The green indicator lights up when the ship speed is less than the safe speed value. The yellow indicator lights up when the ship speed is more than the safe speed value but less than dangerous speed value. The red indicator lights up when the ship speed is more than the dangerous speed value.

To move or rotate the sensors or display board, open the objects context menu and choose the Move or Rotate item.



The display board shows the following information:

- Speed range (red indicator if the speed is higher than 15 cm/s, yellow from 15 to 7 cm/s, green less than 7 cm/s);
- Ship motion direction (R ship approaching the wall, E moving from the wall);
- Ship speed, cm/s (2-position digital indicator);
- Distance from the laser sensor to the ship hull along the wall normal, m (3-position digital indicator).



5.1.5.6 Oil Platforms

There are three types of oil platform (Oil platform, Oil platform 1, Oil platform 2).

To set an oil platform object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the Oil platform (Oil platform 1, 2) object, and click the Next button.
- 3. In the Orientation field, specify the object orientation.
- 4. Click the Finish button.
- 5. Click on the chart.



5.1.5.7 SPM Object

The SPM (Single Point Mooring) object is a beam, rotating freely around its foundation under the effect of a force applied in the mooring point.

NOTE: The SPM object has only one mooring point.

To set an SPM object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Mooring category, select the SPM (SPM 2) object, and click the Next button.
- 3. In the Orientation field, specify the object orientation.
- 4. Click the Finish button.
- 5. Click on the chart.



5.1.6 Operations with Inflatable Fenders

Inflatable fenders are non-stationary mooring fenders simulated for some ship models.

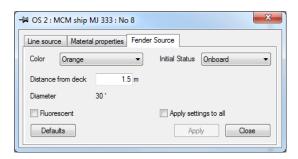


To use inflatable fenders, perform the following actions:

- 1. Position the cursor on the ship's mooring point (except for stern and bow mooring points) and right-click.
- 2. Choose the **Properties** context menu item.



3. Open the Fender Source tab.



- 4. From the Color drop-down list, select the fender colour.
- 5. From the Initial Status drop-down list, select the fender status (Onboard or Outboard).
- 6. In the Distance from deck field, set the distance from the deck to the fender in meters.
- 7. To set the fluorescent fender, select the Fluorescent checkbox.
- 8. To apply fender settings to all ship mooring points, select the Apply settings to all checkbox.
- 9. Click the Apply button to apply new settings, or click the Defaults button to return to the initial settings.

The outboard fenders will be shown on the chart as circles.



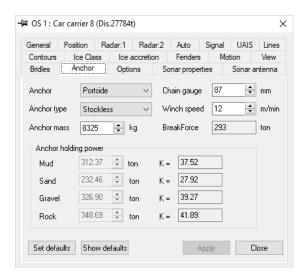
NOTE: The Trainee can control the inflatable fenders on the the Moor/Fenders page of the Conning Display.

5.2 Operations with Anchors

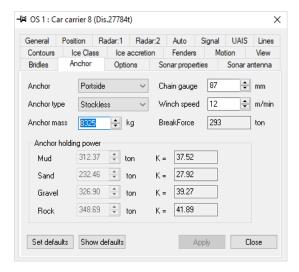
5.2.1 Setting Anchor Properties

To set anchor properties:

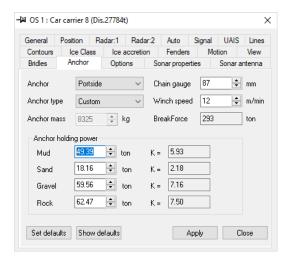
1. Before the exercise start, open the **Anchor** tab of the ship properties panel.



- 2. From the Anchor drop-down list, select the anchor: Portside, Starboard, Stern (if available).
- 3. From the Anchor type drop-down list, select the anchor type: Custom, Gruson's, Hall's, Stockless, Stokes'.
- 4. If the Gruson's, Hall's, Stockless or Stokes' type of anchor is chosen, set the anchor mass in the Anchor mass field. Values and coefficients of the anchor holding power for different types of seabed will be automatically calculated in the Anchor holding power group box.



5. If the **Custom** type of anchor is chosen, set values of anchor holding power for different types of seabed (mud, sand, gravel, rock). The anchor mass value will be automatically calculated.



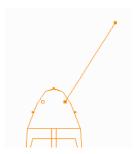
NOTE: To set the seabed type, use the Overall Condition Manager (see "Seabed Properties" on page 75) or Composite condition zone (see "Local Environmental Conditions" on page 84).

- 6. In the Chain gauge field, set the anchor chain gauge (mm).
- 7. In the Winch speed field, set the winch speed (m/min).
- 8. Click the Apply button to apply the set parameter values.
- 9. To view the default values, click and hold down the **Show defaults** button. When the button is released, the last values set by the Instructor are shown.
- 10. Click the Set defaults button to set the default values.

5.2.2 On-Chart Control of Anchors

To drop an anchor, use the following procedure:

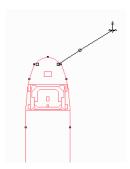
- 1. Increase the display scale so that the anchor points become visible on the object contour. Anchor points are shown as empty squares at the bow/stern of the ship.
- 2. Click on an anchor point and move the cursor beyond the object contour. The screen will display the anchor chain going from the anchor point to the cursor.



3. Click on the chart to set the anchor position. An anchor sign appears on the chart.



After the exercise start, a mark (touch point) will appear on the picture of the anchor line, indicating that the anchor chain has reached the ground. The line from the anchor to the touch point shows the part of the anchor chain lying on the ground; from the touch point to the ship - the sagging part of the anchor chain.

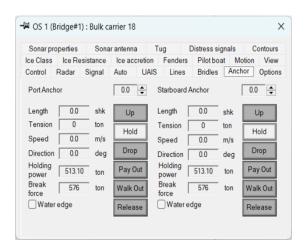


To heave the anchor, use the following procedure:

- Click the anchor sign 4.
- 2. Move the cursor within the ship contour and click.

5.2.3 Anchor Control During Exercise

After the exercise start the Anchor tab of the ship properties panel looks as follows:



To control anchor winches during the exercise, use the following buttons:

- . Up to heave the anchor chain;
- Hold to hold on the anchor chain;
- Drop to drop the anchor chain by the anchor weight;

- Pay Out to pay out the anchor chain (at the rate equal to 50% of the anchor Drop speed);
- Walk Out to walk out the anchor chain (at a preset winch speed);
- Release to emergency release the anchor chain.

To pay out the anchor to the water edge and then stop the winch, select the Water edge checkbox.

NOTE: Please note that the anchor chain is not visible for some ship models.

5.3 Towing Operations

5.3.1 Setting Bridle

Usually, towing operations are performed with the help of a special towing bridle.

To set a bridle, you can use two methods:

- Method 1: Setting the bridle by using mooring points (see "Using Mooring Points" on page 310).
 In this case, the Instructor can create a bridle only with two ends. It is not possible to add or remove the bridle ends afterwards.
- Method 2: Setting the bridle by using the mooring points' context menu (see "Using Context Menu" on page 312).
 In this case, the Instructor can set a single-ended or a dual-ended bridle. It is possible to add or remove its ends any time before and after the exercise start. The Trainee can perform the same modifications during the exercise without recreation of the whole bridle.

5.3.1.1 Using Mooring Points

To set a bridle with the help of mooring points, use the following procedure:

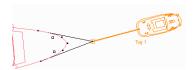
- 1. Increase the display scale so that the mooring points become visible on the object contour.
- 2. On the ship contour, choose the points to which the towing bridle will be connected. Click the first mooring point, then click the second mooring point.



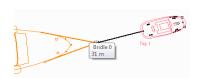
3. To set the bridle length, click the central point of the bridle ("fish plate"), move the cursor and click again (the operation is available before starting the exercise). The position of the "fish plate" will be fixed on the chart.



4. Click the tug attachment point, and then click the "fish plate".

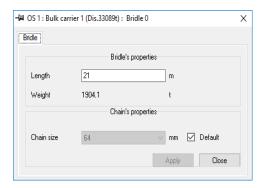


5. Point the cursor at the bridle, the prompt window appears.

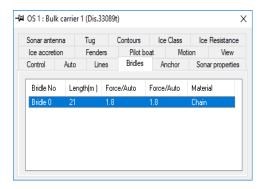


To change the bridle properties:

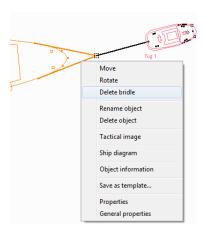
1. Right-click on the bridle and choose the Properties menu item. The bridle properties panel appears.



- 2. To change the bridle length, enter a new value in the Length field.
- 3. To change the chain: clear the Default checkbox and select a new value in the Chain size drop-down list.
- 4. Click the Apply and Close buttons.
- 5. All bridle properties will be displayed on the Bridles tab of the ship properties panel.



6. To delete a towing bridle, open the bridle context menu and choose the **Delete bridle** item.

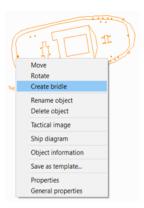


NOTE: For towing operations, use the advanced (6 DoF) target models.

5.3.1.2 Using Context Menu

To set a bridle with the help of the mooring points' context menu, use the following procedure:

- 1. Increase the display scale so that the mooring points become visible on the object contour.
- 2. On the tug contour, choose the point to which the towing bridle will be attached.
- Open the context menu of the mooring point and choose the Create bridle item.



4. To set the bridle line length (to specify a "fish plate"), move the cursor and click (the operation is available before starting the exercise). The position of the "fish plate" will be fixed on the chart.



- 5. Attach one or two bridle ends to the target ship as follows:
 - To attach only one end, move the cursor again, and then left-click and right-click the mooring point to which you want to connect the end. The single-ended bridle is created.

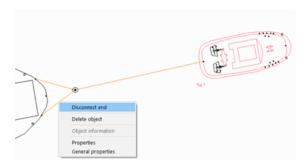


 To attach two bridle ends, move the cursor again, click the mooring point to which you want to connect the first end, move the cursor again, and then click the mooring point to which you want to connect the second end. The dual-ended bridle is created.

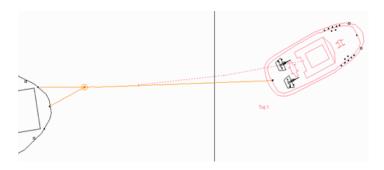


NOTE: You can add or remove the bridle ends during the exercise.

6. To remove a bridle end, open the end context menu and choose the **Disconnect end** menu item.



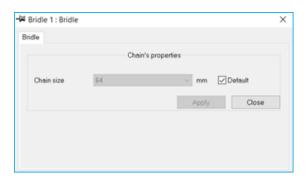
7. To add a bridle end, click the "fish plate" and then click the ship mooring point to which you want to connect this end.



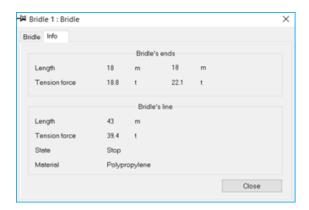
NOTE: Both ends of a bridle should always have the same length. Therefore, please note that you will not be able to add the second end if the distance to the required mooring point exceeds the end's length.

To change the bridle properties:

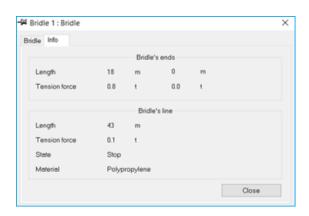
1. Right-click on the bridle and choose the **Properties** menu item. The bridle properties panel appears.



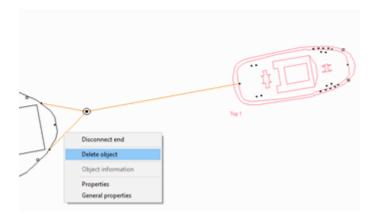
- 2. To change the chain: clear the Default checkbox and select a new value in the Chain size drop-down list.
- 3. Click the Apply and Close buttons.
- 4. All bridle properties will be displayed on the Info tab that appears after the exercise start.



When there is only one bridle end, it is shown as follows:



5. To delete a towing bridle, open the bridle context menu, choose the **Delete object** menu item and then click the **OK** button to confirm the object deleting.



NOTE: For towing operations, use the advanced (6 DoF) target models.

5.3.2 Creating Barge Tow

For use as towing/pushing objects, the barges can be arranged in different tows. Examples of such tows are listed below.

Example 1. Pushing two barges tow



- 1. Position barges and push boat on the chart in accordance with the desired tow configuration.
- Specify characteristics of each object mechanical interaction to prevent the tow collision (see "Specifying Options" on page 114).
- 3. Set properties of mooring points on the push boat and barges (see "Setting Mooring Point Properties" on page 293).
- 4. Connect barges with lines.



5. Connect the barge tow and the push boat.



Example 2. Towing two barges tow



- 1. Position barges and push boat on the chart in accordance with the desired tow configuration.
- 2. Set properties of mooring points on the push boat and barges (see "Setting Mooring Point Properties" on page 293).
- 3. Connect barges with lines and specify characteristics of each object mechanical interaction as required (see "Specifying Options" on page 114).
- 4. Connect the bridle from the first barge to the line from the tug.

5.3.3 Operations with Tow Winches

5.3.3.1 Setting Winch Point Properties

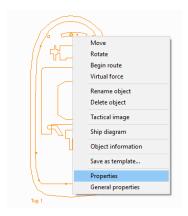
Diesel and hydraulic winches are used on some tug models (own tugs and tug- targets 6DoF). In addition to mooring points, these tugs have special winch points which are displayed on the ship contour as rhombi.



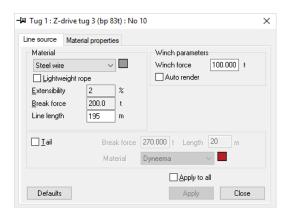
Winch points have a set of parameters which determine the of lines and tow winch properties .

To set properties of a winch point, use the following procedure:

- 1. Before the exercise start, increase the chart scale so that the winch points become visible on the object contour.
- 2. Right-click on the winch point, the context menu appears.

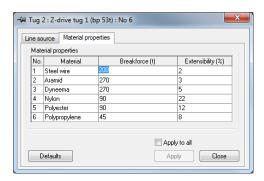


3. Choose the Properties item. The Line source tab of the properties panel will be opened.

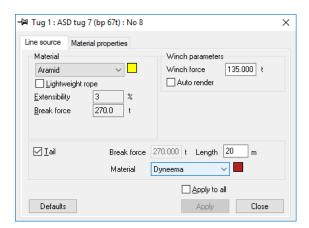


- 4. In the **Material** group box, select the line material from the drop-down list: Steel wire, Aramid, Dyneema, Nylon, Polyester, Polypropylene. The **Extensibility** indicator shows the line extensibility. The **Break force** indicator shows the line break force. The colour indicator shows the line colour.
- 5. If necessary, set the light-weight line, by selecting the Lightweight line checkbox.
- 6. To change the line break force value:

- a. Open the Material properties tab.
- b. In the **Breakforce** column enter a new value for the required material type, press the <Enter> key and click the **Apply** button.



- c. To apply settings to all winch points of the ship, select the Apply to all checkbox and click the Apply button.
- d. To return to the initial settings, click the **Defaults** button.
- e. To continue the editing of line properties, open the Line source tab.
- 7. If necessary, set the maximum line length in the **Line length** field (this option is available if the **Limited length** checkbox was selected on the **Options** tab of the ship properties panel).
- 8. Set the winch parameters in the Winch parameters table on the Line source tab:
 - a. In the Winch force field, set the winch pulling force.
 - b. If necessary, select the Auto render checkbox to automatically pay out the line at large loads.
- 9. To set a line consisting of two different parts line's general part and tail (the tail connects the line's general part with the towed vessel):
 - a. Select the Tail checkbox.
 - b. In the **Length** field, set the line tail length.
 - c. From the **Material** drop-down list, select the line tail material. The colour indicator shows the line tail colour. The **Break force** indicator shows the break force of the line's tail.



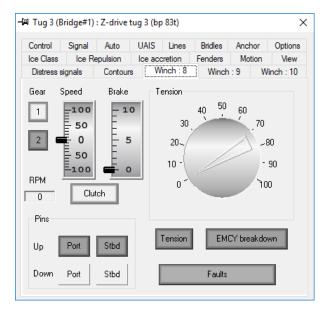
- d. To change the line tail break force value, open the **Material properties** tab. In the **Breakforce** column enter a new value for required material type, press the <Enter> key and click the **Apply** button.
- 10. To apply settings to winch points of the ship, select the Apply to all checkbox.
- 11. Click the Apply button to save new settings, or click the Defaults button to return to the initial settings.

NOTE: Coloured lines will not be displayed on the visualization screen unless the **Colored** checkbox is selected on the **Options** tab of the ship properties panel (see "Specifying Options" on page 114).

5.3.3.2 Hydraulic Winch Control

The model of a hydraulic tow winch is used on some tug objects which have special winch points, for example: Z-Drive tug 1 (\mathbb{N}^2 6), Z-Drive tug 3 (\mathbb{N}^2 6), and Fin First VS tug 1 (\mathbb{N}^2 5).

Winches and pins are controlled on the **Winch** tab of the tug properties panel after the exercise start. The name of the tab specifies the number of the winch point, e.g., **Winch**: 10.



Pins Control

The instructor can raise or lower the pins on the **Winch** page both, before and after the exercise start. To do this, use four buttons in the left bottom corner of the page:



- 1. To raise a pin, click the UP button: Port (portside pin), Stbd (starboard pin);
- 2. To lower a pin, click the **Down** button: **Port** (portside pin), **Stbd** (starboard pin).

Winch Control Modes

There are two tow winch control modes:

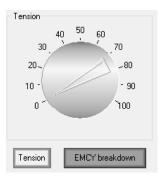
- · Manual winch control;
- Tension mode mode of maintaining the set tension. In this mode, depending on the tension control position, the line is either heaved or slacked away to maintain the constant set tension.

To switch on the manual winch control mode:

- 1. The Clutch button is activated by default, i.e. the cohesion is on.
- 2. Use the Brake handle to release the winch brake which is by default set at the maximum braking force.
- 3. Use the **Speed** handle, to set the winch power in per cent:
 - Push up the handle to heave the line;
 - Pull down the handle to slack away the line.
- 4. Use the Gear 1 and Gear 2 buttons to switch the transmission of the winch drive clutch.
- 5. Click the Tension button or click the EMCY breakdown button to switch off the manual winch control mode.

To switch on the Tension mode:

1. On the **Tension** scale, set the towing tension per cent from 0 to 100% and click the **Tension** button. This button will glow.



NOTE: The value of the tow line pulling force is set on the winch poin properties panel (the Winch force parameter).

2. To switch off the Tension mode, click the Tension button again or click the EMCY breakdown button.

As the EMCY breakdown button is activated, the line is slacking away.

Winch Faults Setting

To set winch faults:

- 1. Click the Faults button on the winch control panel.
- 2. Check\clear winch faults in the list-box.
- 3. Click the OK button.

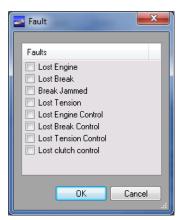


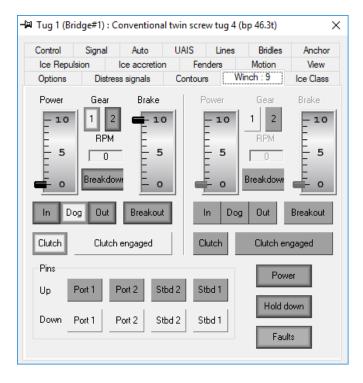
Table 14. Hydraulic winch faults

FAULT	DESCRIPTION
Lost Engine	The winch engine is out of order. The line length does not change when the Speed handle is moved, the winch is actually in the "Stop" state
Lost Break	The brake is out of order. Moving the Brake handle does not affect the line slacking speed as if the brake is in the zero position
Break Jammed	The line is not slacked away as if the brake is set at the maximum value
Lost Tension	The Tension control is out of order as if the Tension handle is in the minimum position
Lost Engine control	Moving the Speed handle has no impact on the winch state
Lost Break Control	Moving the Brake handle has no impact on the breaking effort
Lost Tension Control	Moving the Tension handle has no impact on the tension control effort
Lost clutch control	The Clutch button is out of order

5.3.3.3 Diesel Winch Control

The model of the diesel tow winch functions with the Conventional twin screw tug 4 (bp 46.3t) which has a special line attachment point (No 9). The winch control console is available on the simulator bridge of this tug type.

On the instructor workplace, the tow winch is controlled in the course of the simulator session from the **Winch** tab of the own tug properties panel.



The Winch page contains:

- Drum operating mode setting buttons:
 - In to heave the line;
 - o Dog to stop the winch;
 - Out to pay out the line.
 - Winch power control telegraph (scale from 0 to 100%).
 - Brake control telegraph (scale from 0 to 100%).
 - Gear 1 and Gear 2 buttons to switch the transmission of the winch drive clutch.
 - · Winch RPM indicator.
 - Clutch button and Clutch engaged indicator.
 - Breakout button for the emergency line break.
 - Breakdown button for the emergency winch brake release and turning off the clutch and power. After the button
 activating, the line is slacking away.
 - Pins control buttons (two on the starboard Stbd 1, Stbd 2, and two on the portside Port 1, Port 2), where:
 - Up buttons to raise the pins;
 - o Down buttons -to lower the pins.
 - Power button for turning on the pin power supply.
 - **Hold down** button for turning on holding down of the line's vertical motion. This is necessary to prevent the line from sliding down from the pins.
 - · Faults button for setting winch faults.

To switch from one drum operating mode to another, the line is required to have no speed, and the **Power** telegraph handle should be on the zero.

In the render mode with the gear engaged, the winch is controlled with the **Power** telegraph. With the gear disengaged, the line is rendered by the force of the line itself.

NOTE: Control of the starboard drum winch drive is currently not implemented.

5.4 Towing in Inland Waterways

The articulated tug barge (ATB) and pusher-barge train are used for towing operations in inland waterways.

NOTE: Before the exercise start, it is advisable to disable the **Bow waves** option on the **Environment settings** panel (**Edit\Environment Settings...** menu item).

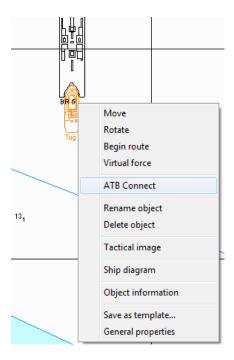
5.4.1 Articulated Tug Barge

The ATB-connection system consists of paired objects of the Tug and Barge categories.

Creating the ATB-connection is available before and after the start of the exercise.

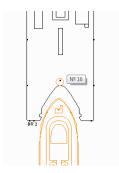
To create an ATB-connection, use the following procedure:

- 1. Set the barge on the chart:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the **Barge** category, select a special barge object, e.g., the **Oil barge 650 series**. Place the object on the chart.
- 2. Set the pusher tug on the chart:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the **Tug** category, select a special tug object, e.g., the **Conventional twin screw tug 6**. Place the object on the chart behind the stern of the barge. For a successful connection, the tug should be close to the barge.
- Open the tug context menu and choose the ATB Connect menu item.

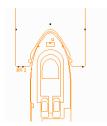


4. Position the cursor over the barge.

NOTE: The O cursor shows that the tug/barge connection is available; the X cursor shows that the connection is not available.

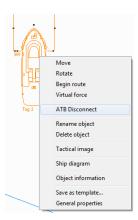


5. Click on the barge. The ATB-connection will be created.



Before the exercise start, you can move and rotate the ATB-connection.

To disconnect the ATB-connection, open the context menu of the tug or barge and choose the ATB Disconnect menu item.

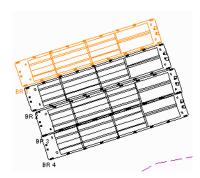


5.4.2 Pusher-Barge Train

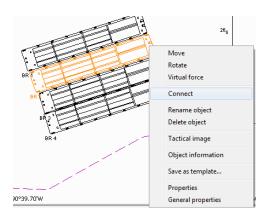
The pusher-barge train can contain any number of barges and one or more pusher tugs.

To create pusher-barge train, use the following procedure:

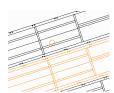
- 1. Set several barges on the chart:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the **Barge** category, select a barge object, e.g., the **Big box barge**. Place the object on the chart. In the same way, select other barge objects and place them on the chart close to each other.



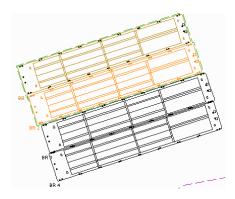
2. To connect two adjacent barges, open the context menu of one of the barges and choose the **Connect** menu item.



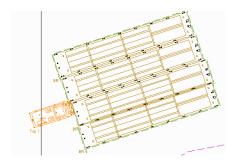
3. Position the cursor over the next barge (connection is not possible unless the cursor is shown on the chart in the form of a circle).



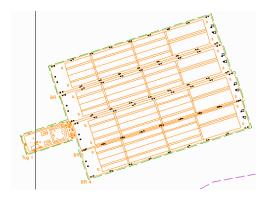
4. Click on the barge. Coupled barges will be outlined with a dashed green line.



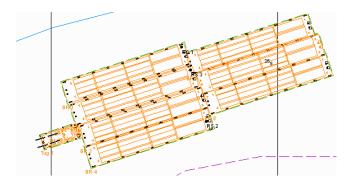
- 5. In the same way, connect other barges together.
- 6. Set pusher tug on the chart:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the **Tug** category, select a tug object, e.g., **Twin screw push boat 4**. Place the object behind the barge train.



- 7. Open the tug context menu and choose the Connect menu item.
- 8. Click the barge train. The pusher-barge train is created.



- 9. To connect the pusher-barge train to another barge train:
 - a. Create a new barge train (see steps 1-5).
 - b. Open the context menu of the new barge train and choose the Connect menu item.
 - c. Left click on the pusher-barge train (barge trains should be close to each other).



The Instructor can move and rotate the pusher-barge train (only before the exercise start) by using the properties panel of barge train or tug.

During the exercise, other barges can be added to the barge train.

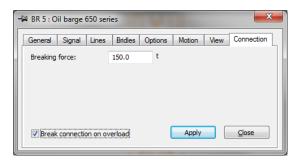
To disconnect an object from the pusher-barge train, open its context menu and choose the Disconnect menu item.

5.4.3 Breaking Connection

By default when a collision or failure maneuvering the connection of barge train with tug or ATB-connection does not break.

To simulate the breaking connection in these cases, use the following procedure:

- 1. Open the context menu of the tug or barge (barge train) and choose the General properties item.
- 2. Open the Connection tab of the properties panel.
- 3. Select the Break connection on overload checkbox.



- 4. In the Breaking force field, set the breaking force value (tons).
- 5. Click Apply and Close buttons.

5.4.4 Attaching Remote Sensor to Barge

To measure the depth and transverse speed on the bow of the train's forward barge or on the bow of the ATB-barge, the remote sensor is used.

Navigation data from remote sensor will be transmitted to the bridge on the IBID panel.

To attach a remote sensor to the barge, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Reference category, select the Remote sensor object. Place the object on the bow of the barge.
- 3. Open the context menu of the remote sensor and choose the Attach sensor menu item.

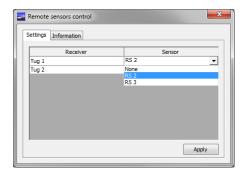


4. Click on any point on the barge. The sensor is attached to this barge.



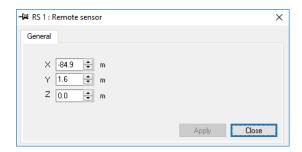
NOTE: In the same way, you can attach other remote sensors to the barges. Navigation data is transmitted to the bridge from only one remote sensor.

- 5. Select the remote sensor to transmit data to the bridge:
 - a. Choose the View\Panels\Remore sensors... menu item or click the toolbar button. The Remote sensors control panel appears.
 - Next to the name of the receiving vessel, select the sensor from the Sensor drop-down list and then click the Apply button.

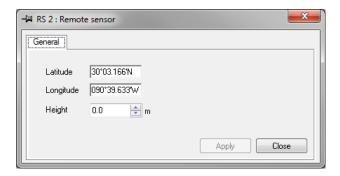


To change the remote sensor position:

- 1. Open the remote sensor properties panel.
- 2. If the sensor is attached to an object, set X/Y/Z coordinates- longitudinal/transverse/vertical shift relative to the object center.



3. If the sensor is not attached to an object, set absolute sensor position: Latitude, Longitude and Height.

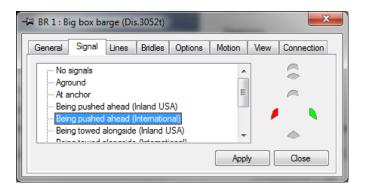


4. Click the Apply and Close buttons.

5.4.5 Barge Navigational Signals

To control navigational signals on the barges:

- 1. Open the Signals tab of the barge properties panel.
- 2. Select navigational situation from the list, for example "Being pushed ahead (International)".



- 3. Switch on/off single lights by clicking on lights symbols (in the right part of the **Signals** tab). For example, set a green light on the right barge, a red light on the left barge.
- 4. Click the Apply and Close buttons.

5.4.6 Applying Assist Force

During the towing operations, the assist force can be applied to any point of the hull of the pusher tug or barge.

To apply the assist force, use the following procedure:

- 1. After the exercise start on the bridge, choose the Chart\Assit Force menu item or click the toolbar button.
- 2. Click on the hull of the object in a point where you with to apply the assist force.

NOTE: The \bigcirc cursor shows that the applying of the assist force is available at this point. The \times cursor shows that the applying of the assist force is not available at this point, because the point is outside the object hull.

3. Move the cursor to set the amount and direction of the force applied.



4. To terminate the assist force applying, right-click.

5.5 Operations with Automatic Tugs

Automatic tugs are target tugs (6-DoF models) operating in the automatic mode.

In the automatic mode, the target tug is controlled by the "virtual pilot" who executes commands, given either by the Trainee from the Conning Display or by the Instructor.

Automatic tugs can work with own ships or with target ship (6-DoF models).

In the automatic mode the tug can perform several typical maneuvers without manual handling of its controls: Follow to point, Escort, Push, Pull, Attach.

Depending on the current operation, the tug contour on the chart changes its colour according to the following table.

Table 15. Target tug contour colour depending on the status/operation

STATUS/OPERATION	COLOUR
Going to carry out an operation	Green
Carrying out an operation	Magenta
Free	Black
Selected as an object	Orange

The target tug can be reserved for operation with one ship. The requests for operations from other ships are ignored at this time.

A non-reserved target tug can work with any other ship, performing operation as requested (each operation is completely performed, to perform the next operation, the previous one is not stopped).

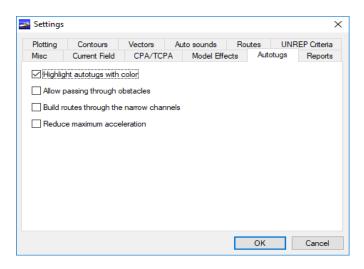
Work with non-reserved automatic tugs is possible only on the Instructor workplace. To switch the tug to the automatic mode on the Conning Display, the Trainee has to reserve it first.

LIMITATIONS: For autotugs, tides are only supported if they are set using the Overall Conditions Manager (see "Tide Properties" on page 77). In other cases, for example, if tides are set using the Condition zone manager or using the tide and current databases, they will not be taken into account by the autotugs.

5.5.1 Automatic Tug General Settings

To enter general settings for automatic tugs, use the following procedure:

Choose the View\Settings... menu item and open the Autotugs tab of the Settings panel.



To switch on/off the option of highlighting autotugs with colour on the chart, select/clear the Highlight autotugs with color checkbox.

If this option is on, contours of autotugs are highlighted with different colours according to the autotug current status or operation fulfilled.

NOTE: It is impossible to switch off highlighting with colour on the Conning Display.

- To allow/prohibit autotugs passing through the obstacles, select/clear the Allow passing through obstacles checkbox.
- 4. To switch on/off the option of building the routes of autotugs through the narrow channels, select/clear the **Build** routes through the narrow channels checkbox. If this option is on, the tug can go through a narrow channel at the expense of decreasing its own safety criteria. It is no advisable to switch on this option unless strictly necessary, otherwise the decrease of the safety criteria may impact the tug-ship interaction and result in collision
- 5. To switch on/off reduction of the maximum acceleration for autotugs, select/clear the **Reduce maximum** acceleration checkbox.
- 6. To confirm the settings, click OK.

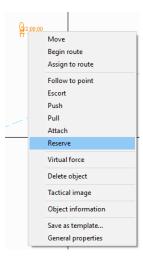
5.5.2 Tug Reserving and Releasing

The target tug can be reserved for operation with an own ship or target ship (6-DoF model).

The target tug, engaged by one ship, cannot be reserved by another ship unless it is released by the "Let go" command.

To reserve the target tug, use the following procedure:

1. Open the target tug context menu and choose the **Reserve** item.



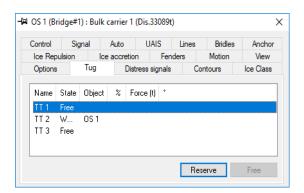
2. Click on the contour of the ship which the target tug should work with.

The target tug will switch to the automatic mode. Hovering on the tug, displays the screen prompt containing the target tug current status: "Working with..." and the name of the ship which the target tug is operating with. The requested tug is marked with the "auto" label on the chart and highlighted with bold contour:



Also, the target tug could be reserved from the **Tug** tab of the own ship (or 6-DoF target ship) properties panel. To do this, use the following procedure:

- 1. Open the ship context menu and choose the General properties item.
- 2. On the properties panel, open the **Tug** tab and select a free target tug from the list of target tugs.



NOTE: Data on the target tugs available in an exercise, is presented in a tabular form, where the following is specified: **Name** - target tug name; **State** - target tug current operation mode; **Object** - the name of the own ship which the target tug is working with; % - current thrust value; current thrust direction, deg.

3. Click the Reserve button.

To release the reserved target tug, click the **Free** button on the **Tug** tab of the own ship (ship-target) properties panel or choose the **Let go** command from the target tug context menu.

5.5.3 Basic Operations

The following basic operations are available for the automatic tugs:

- Follow to point;
- Escort;
- · Push;
- · Pull;
- · Attach;
- Thrust.

5.5.3.1 Follow to Point Mode

In this mode, the target tug follows to the absolute geographic point specified by the Trainee or Instructor, avoiding mobile and static obstacles at the maximum available speed, and sets adrift at point of destination.

To switch the tugboat to the "follow to point" mode, use the following procedure:

- 1. Open the target tug context menu and choose the Follow to point item.
- 2. Click the destination point on the chart.

Having received the command, the target tug will start proceeding to the set point at the maximum speed. Position the cursor over the tug contour, the prompt shows the target tug current status: "Follow".



5.5.3.2 Escort Mode

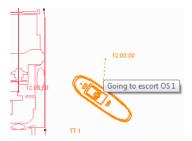
In this mode, the target tug follows the ship at a certain distance.

To switch the tugboat to the "escort" mode, use the following procedure:

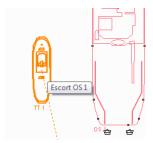
- 1. Open the target tug context menu and choose the **Escort** item.
- 2. If the target tug has not be reserved, click on the contour of the ship which the tug should escort.
- 3. Position the cursor in a point at the selected distance from the ship contour and left-click.

Having received the command, the target tug will start proceeding to the set point at the maximum speed.

Position the cursor over the tug contour, the prompt shows the target tug current status: "Going to escort" and the name of the ship which the target tug is operating with:



In the set point, the current status turns to "Escort":



NOTE: The target tug will be not able to get to the work point if the ship is proceeding at a speed higher than the target tug maximum speed. In the destination point, the tug will have the same speed and the same heading as the escorted vessel.

5.5.3.3 Push Mode

In this mode, the target tug pushes the ship to a given point with a given thrust. The force direction is limited by the target tug's position relative to the pushed ship hull. When the "Push" command is given, the following parameters are set: push position, thrust level, and direction. Thrust level is specified in per cent of the maximum bollard pull.

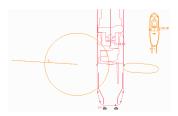
The tug proceeds to the position at maximum speed avoiding mobile and static obstacles. In the destination point, the tug sets on the specified thrust heading and begins pushing.

Alteration of the trust level and direction can be initiated by the "Thrust" command.

NOTE: All tractor tugs (Voith Schneider type) have a pushing point at the stern. All reverse tractor tugs (Z-drive tug) and conventional tugs have a pushing point at the bow.

To switch the tugboat to the "push" mode, use the following procedure:

- 1. Open the target tug context menu and choose the **Push** item.
- 2. Click on the ship contour in the point where the thrust must be applied.
- 3. By moving the mark along the force vector and rotating the tug contour around the fastening point, set the force value and direction, and click the left mouse button:



Having received the command, the target tug will start proceeding to the work point at the maximum speed. Hovering on the tug displays an on-screen prompt containing the target tug current status: "Going to push" and the name of the ship which the target tug is operating with:



Having reachedthe pushing point, the tug starts pushing the ship in the specified direction. Hovering on the tug displays an on-screen prompt containing the target tug current status: "Push", the name of the ship which the ttarget tug operating with, force value, direction of force vector (in degrees) and side of the ship which the work point is situated on: port side (PS) or starboard (SB):



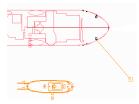
5.5.3.4 Pull Mode

In this mode, the target tug can pull the vessel on a towline of given length in any direction limited by ±90 degrees to the perpendicular to the ship bow in the fastening point. The towing mode is initialized by the "Pull" command whereby the following parameters are set: fastening-to-the ship point, towing position, thrust level and direction.

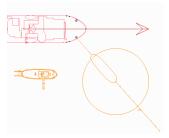
Alteration of the trust level and direction can be initiated by the "Thrust" command.

To switch the tugboat to the "pull" mode, use the following procedure:

- 1. Open the target tug context menu and select the Pull item.
- 2. Position the mouse cursor on the mooring point on the ship contour and click the left button. The line will be fixed on the mooring point:

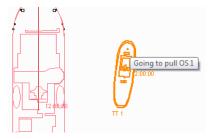


3. Position the mouse cursor on the chart point where the tug-target is required to pull the ship from, and click the left button. The mouse cursor will assume the shape of the target tug contour with a force vector. The contour will be fixed on the mooring point:

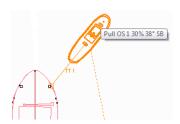


4. By moving the mark along the force vector and rotating the tugboat contour around the mooring point, set the force value and direction and click the left mouse button.

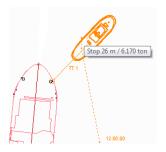
Having received the command, the target tug will start proceeding to the work point at the maximum speed. Hovering on the tug displays an on-screen prompt containing the target tug current status: "Going to pull" and the name of the ship which the target tug is operating with:



Having reached the pulling point, the tug starts pulling the ship in the specified direction. The tug contour grows magenta. Hovering on the tug, as the tug starts to pull the screen, displays a prompt containing the target tug current status: "Pull", the name of the ship which the target tug is operating with, force value, direction of force vector (in degrees) and side of the ship which the work point is situated on: port side (PS) or starboard (SB):



Hovering on the tug winch which is used to pull when the tug's line is fixed the screen displays a prompt containing the winch current status: "Veer" if the line is wound off freely, "Winch" if the line is wound up on the winch, or "Stop" if the line length is fixed, the current line length and winch force of (in tons). When the current status is "Veer" or "Winch" the force is equal to 0 ton.



5.5.3.5 Attach Mode

To imitate the "push-pull" operations, the tug can be made fast either by the bow or by stern line to any mooring point on the vessel.

The line can be fastened by giving the "Attach" command.

At the "Attach" command, the tug moves close to the mooring point and sets a short line either from the bow (reverse tractor and conventional tugs) or the stern winch (tractor tug) to the specified knight of the vessel.

NOTE: Depending on the type, all tugs have either one aft towing point (tractor tug) or two towing points: one on the bow and one on the stern.

To switch the tugboat to the "attach" mode, use the following procedure:

- 1. Open the tugboat context menu and choose the **Attach** item.
- 2. Click on the mooring point on the own ship contour.

Having received the command, the target tug will start proceeding to the mooring at the maximum speed. Hovering on the tug displays an on-screen prompt containing the target tug current status: "Going to attach" and the name of the ship which the target tug is operating with. As the tug is abeam the mooring point, the fastened line appears on the screen. The tug current status does not change at same time.



The tug moves to the mooring point connected to the ship by the line. Hovering on the tug, as the tug is attached the screen displays a prompt containing the target tug current status: "Attach" and the name of the ship which the target tug is operating with:



While the tug is fastened with a short line by using the "Attach" command, the direction and level of thrust can be altered by the "Thrust" command. The mooring point does not change at this time. Whether the status will be "Push" or "Pull" is dependent on the direction of the force vector relative to the ship. At the same time, the current status of the tug contained in prompt is changed to "Pull (attached)" or "Push (attached)".





NOTE: As the "Pull" command is sent the attached tug stays attached in the same point.

At the "Push" command, the attached tug remains attached in the same position unless its transfer to a new operating point for pushing requires the line to cross the ship superstructure. I.e., if the tug is attached with a line to the ship's portside, and the push point is selected on the starboard side.

The tug winch current status, line length and force in the "Attach" mode are contained in the prompt just as in the "Pull" mode.

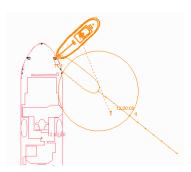
To unfasten the tug, select any context menu command different from "Pull" or "Push".

5.5.3.6 Thrust Command

The thrust level and direction can be altered by the Thrust command which appears in the context menu after the "pull" or "push" and "attach" mode has been enabled.

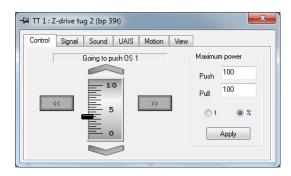
To alter the thrust level and direction, use the following procedure:

- 1. Open the target tug context menu and choose the Thrust item.
- 2. By moving the mark along the force vector and rotating the tug contour around the mooring point, set the force value and direction and click the left mouse button.



5.5.4 Automatic Tug Control Page

To control a tugboat operating in the automatic mode, use the Control tab of the tugboat's properties panel.



The Control tab contains:

- · Status line which displays the tugboat's current operation mode;
- Lever for setting the thrust value (0-100%); the arrow-buttons above/under the lever are used for moving the lever handle to one scale point.



Buttons with arrows for changing the thrust direction (from -90 degrees to +90 degrees with a step of 5 degrees):



In the right part of the tab you can set the maximum power of the auto tug relative to nominal in the "Push" and "Pull" operation mode:

- 1. In the **Push** field, set the auto tug maximum power in the "Push" mode.
- 2. In the Pull field, set the auto tug maximum power in the "Pull" mode.
- 3. Select the maximum power units of measurement relative to nominal: tons or %.
- 4. Click Apply.

5.6 Pilot Boat Operations

5.6.1 Pilot Boat Maneuvers

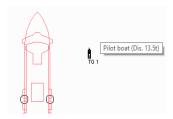
In the automatic mode, the pilot boat as a 6-DoF target model can be reserved for the operation with any own ship or ship target (6 DoF model).

The pilot boat performs the following typical maneuvers:

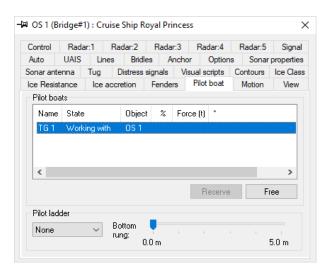
- Follow to point the pilot boat follows to the point specified by the Instructor and then it is drifting at that point.
- Escort the pilot boat is escorting another ship at a certain distance.

To control the pilot boat in the automatic mode, use the following procedure:

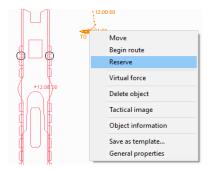
1. Add the pilot boat (e.g., Pilot boat, Pilot boat 1, Lots boat) as a 6-DoF target model. Add an own ship object or ship target (6-DoF model) object which the pilot boat should work with.



- 2. After the exercise start, reserve the pilot boat in one of the following ways:
 - Open the properties panel of the object which the pilot boat will work with. Select the Pilot boat tab.
 In the table, select a free pilot boat and click the Reserve button. The pilot boat state changes to "Working with...".



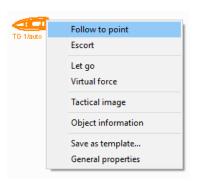
 Right-click the pilot boat on the chart, and choose the Reserve context menu item. Then click the object which the pilot boat will work with.



The pilot boat will switch to the automatic mode.

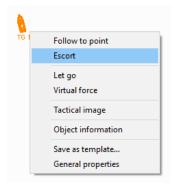


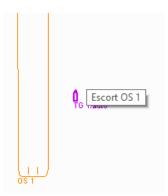
2. To specify the "Follow to point" maneuver: right-click the pilot boat, choose the **Follow to point** context menu item, and then click on the desired point on the chart.



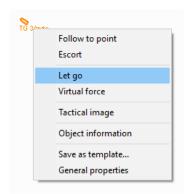


3. To specify the "Escort" maneuver: right-click the pilot boat, choose the **Escort** context menu item, and then click on the escort position point on the chart.





To release the reserved pilot boat, click the **Free** button on the **Pilot boat** tab of the own ship properties panel or choose the **Let go** item of the pilot boat context menu.

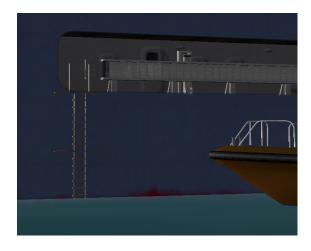


5.6.2 Pilot Transfer

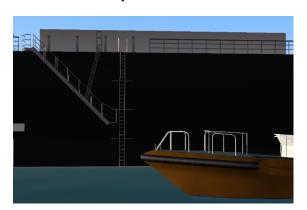
The NTPRO simulates the pilot transfer from the pilot boat to a vessel.

Depending on the vessel type, the following types of pilot ladders are used:

· Pilot ladder



• Pilot ladder in conjunction with an accommodation ladder



· Pilot ladder and a shipside door

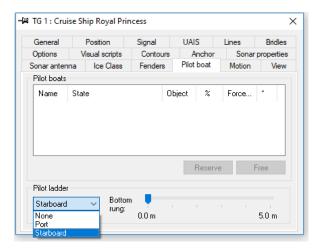


To control the pilot transfer from the pilot boat to a vessel, use the following procedure:

1. Add the pilot boat object (e.g., Rescue vessel, dis.12t) as an own ship and a vessel with a pilot ladder (e.g., Cruise Ship Royal Princess, VLCC 1, 2) as a 6-DoF target ship.

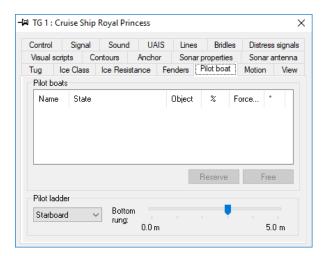


- 2. After the exercise start, open the **Pilot boat** tab of the target ship properties panel and select the pilot ladder in the **Pilot ladder** drop-down list:
 - None no ladder;
 - Port pilot ladder rigged on the port side;
 - Starboard pilot ladder rigged on the starboard side.

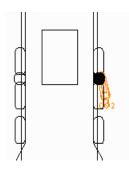


NOTE: The pilot ladder location is marked with a filled circle on the vessel contour.

3. Use the Bottom rung slider to set a distance from the surface of the water to the bottom rung.



- 4. Use the SendPilot IBID panel to set the pilot position on the pilot boat deck: PORT/Left the pilot goes to the portside of the pilot boat; STBD/Right the pilot goes to the starboard of the pilot boat.
- 5. Keep the pilot boat position near the vessel's pilot ladder.



When the pilot is located closest to the ladder, he will automatically start climbing the ladder.



5.7 Operations in Ice Conditions

The simulator enables vessel navigation training in cold weather conditions (below 0°C) and in ice-covered areas.

To set ice conditions, the Instructor can use two methods:

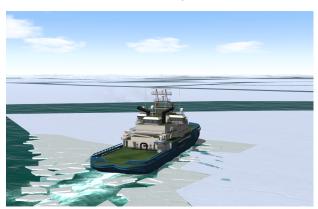
• Method 1: Creating the ice zone by using the Ice object of the Environment category (see "Using Ice Object" on page 345).

In this case, the following ice zone parameters can be set: ice type (ice field, broken ice, brash ice, pancake ice, small floe, and nilas ice); ice thickness and concentration; speed and course of ice zone drifting; ice field channel and radar reflection parameters; number of groups of fishermen on the ice field. The Instructor can add the channels and hummocks to the ice field and plot the ice zone route.



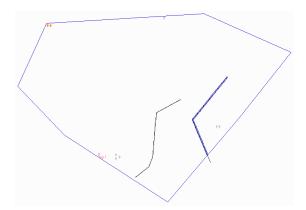
 Method 2: Creating the ice zone by using the Ice generator object of the Environment category (see "Using Ice Generator Object" on page 351).

In this case, the following zone parameters can be set: concentration and strength of ice; underwater ice and broken ice characteristics; speed and direction of ice zone drifting. The Instructor can specify several ice floes of different sizes and thickness within the ice zone. Also, the Instructor can add to the ice zone: channels, ridges of hummocks, fractures, and separate ice floes.



5.7.1 Using Ice Object

For setting an ice zone, you can use the Ice object of the Environment category.



5.7.1.1 Creation of Ice Zone

To create an ice zone, use the following procedure:

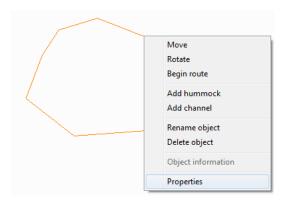
- 1. Before the session start, choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Environment category, select the Ice object and click the Finish button.
- 3. Click on the chart to fix the first zone point.
- 4. Move the cursor to the next point and click to fix the second zone point.
- 5. Repeat the previous action to construct all zone sections.
- 6. To complete the zone construction, right-click.



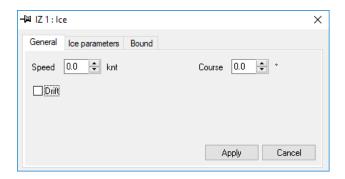
5.7.1.2 Setting Ice Zone Properties

To set the ice zone properties, use the following procedure:

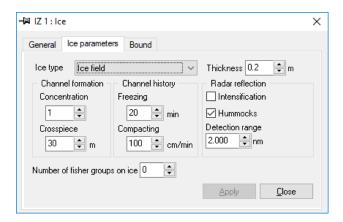
1. Right-click the zone contour and choose the Properties context menu item.



- 2. On the General tab, set the following parameters:
 - a. In the **Speed** field, set the speed of the ice zone motion, knots; in the **Course** field, set the course of the ice zone motion, deg.



- b. Or select the **Drift** checkbox to set the ice zone drift in accordance with the set weather conditions.
- c. Click the Apply button.
- 3. On the **Ice parameters** tab, set the following parameters:
 - a. From the Ice type drop-down list, select the ice type: Ice field, Broken ice, Brash ice, Pancake ice, Small floe,
 Nilas ice.
 - b. If the Ice field type is selected, specify the following settings:

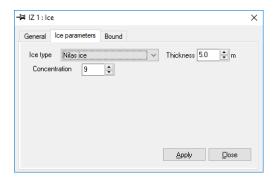


- Thickness ice thickness from 0.1 to 5.0 m.
- Concentration ice concentration in the channel from 1 to 10.
- Crosspiece distance between channel edges (0-70 m).

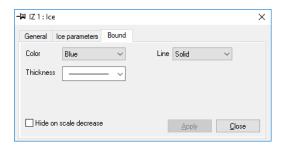
NOTE: If the distance between two channels is smaller than the specified **Crosspiece** value, the solid ice crosspiece is destroyed. If the parameter is set at 0, the crosspiece is not destroyed. The channel width is set on the **Ice class** tab of the ice breaker properties panel (see "Setting Ship Ice Class" on page 349).

- Freezing time of the channel getting covered with ice (2-360 min).
- Compacting speed of broken ice accretion after the ship (1-500 cm/min).
- Intensification switch on/off the intensification of the radar echo from the ice field.
- Hummocks switch on/off the display of hummocks on the radar picture.
- Detection range detection range of ice edge on the radar, in nautical miles.
- Number of fisher groups on ice number of groups of fishermen located on the ice field in random order.

- c. For the other ice types, set the following parameters:
 - Thickness ice thickness from 0.1 to 5.0 m;
 - Concentration ice concentration from 1 to 9;



- j. Click the Apply button.
- 4. On the **Bound** tab, set the following parameters:
 - a. From the Colour drop-down list, select the zone boundary line colour (blue by default).
 - b. From the Line drop-down list, select the zone boundary line type: solid, dashed, dotted.
 - c. From the Thickness drop-down list, select the zone boundary line thickness.

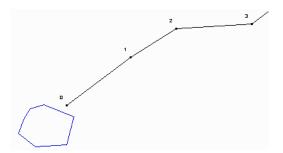


- d. To hide the zone if the chart is zoomed out, select the Hide on scale decrease checkbox.
- e. Click the Apply button.

5.7.1.3 Ice Zone Route

To plot the route of the ice zone:

- 1. Right-click the ice zone contour and choose the Begin route item from the context menu.
- 2. Plot the route in the standard way. The ice zone will be automatically assigned to the route.



To assign an ice zone to any plotted route:

- 1. Right-click the ice zone contour and choose the Assign to route item from the context menu.
- 2. Click the desired route on the chart.

NOTE: The ice zone will be moving so that the start point of the ice zone moves along the route line.

To unassign the ice zone form the route, choose the **Unassign from route** item from the context menu of the ice zone.

The ice zone route waypoints and segments have no properties.

5.7.1.4 Adding Channels to Ice Zone

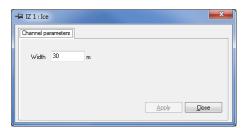
The channels can be added only to the "Ice field" type zone.

To add a channel to the ice field, use the following procedure:

- 1. Right-click the ice zone contour and choose the Add channel context menu item.
- 2. Plot a channel of arbitrary shape within the ice zone.
- 3. To complete the channel creating, right-click.



- 4. Right-click the channel line and choose the Properties item in the context menu of the channel.
- 5. In the Width field, set the width of the channel (from 10 to 70 metres).



6. Click the Apply button to confirm the changes.

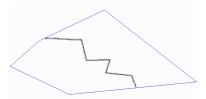
To delete a channel, right-click the channel line and choose the Remove channel item from the context menu.

5.7.1.5 Adding Hummocks to Ice Zone

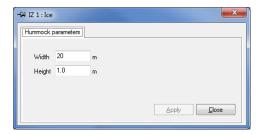
The hummocks can be added only to the "Ice field" type zone.

To add a hummock to the ice zone, use the following procedure:

- 1. Right-click the ice zone contour and choose the Add hummock context menu item.
- 2. Plot a hummock of arbitrary shape within the ice zone.
- 3. To complete the hummock creating, right-click.



- 4. Right-click the hummock line and choose the Properties item from the context menu of the hummock.
- 5. In the Width field, set the hummock width (from 5 to 30 metres).
- 6. In the Height field, set the hummock height (from 0.5 to 3.0 metres).



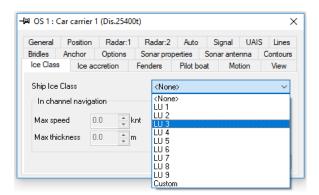
7. Click the Apply button to apply the changes.

To delete a hummock, right-click on the hummock line and choose the Remove hummock context menu item.

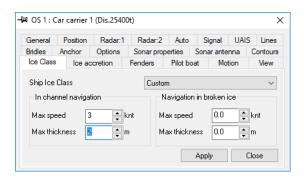
5.7.1.6 Setting Ship Ice Class

To set an ice class of the own ship/tug or target models, use the following procedure:

- 1. Open Ice Class tab of the object properties panel.
- 2. Select the ship ice class from the list (LU1-LU9).



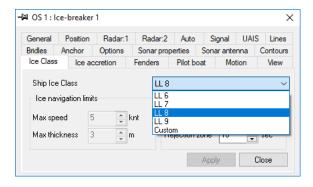
- 3. Or select the Custom item from the Ship Ice Class list and specify the following settings:
 - a. In the In channel navigation group box, set the maximum ship speed (Max speed) and ice thickness in the channel (Max thickness).



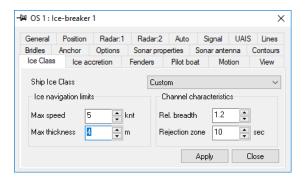
- b. In the **Navigation in broken ice** group box, set the maximum ship speed (**Max speed**) in broken ice and broken ice thickness (**Max thickness**).
- 4. Click the Apply button.

To set an ice class of some ice-breaker models, use the following procedure:

- 1. Open the Ice Class tab of the ice-breaker properties panel.
- 2. Select the ship ice class from the list (LL6-LL9).



- 3. If the Custom ice class is selected,
 - a. Specify the following ice navigation limits:
 - Max speed maximum ship speed.
 - Max thickness maximum ice thickness.



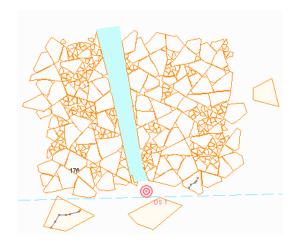
- b. Set the following channel characteristics:
 - Rel. breadth relative channel width;
 - Rejection zone temporary zone of broken ice ejection by the propeller behind the ship stern (sec).

NOTE: By taking into account the temporary rejection zone and icebreaker speed, you can calculate a safe distance for a ship following the icebreaker.

4. Click the Apply button.

5.7.2 Using Ice Generator Object

To set ice conditions within a specific area, you can use the Ice Generator object of the Environment category.



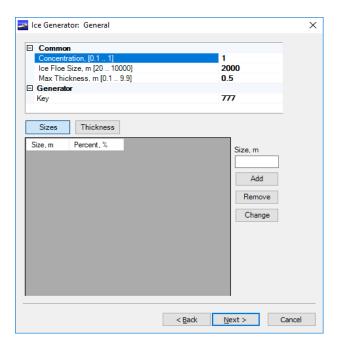
WARNING: If the Ice generator object is used, the following functionality is restricted:

- Anchor Handling, Oil spillage, Fish farming, and Survival craft operations;
- Common scenario;
- Exercise rewind;
- Composite condition zone (only Depth and Seabed are not supported);
- Seabed type simulation;
- Fender zone, Advanced fender zone;
- Jetty walls simulation;
- Simulation of vessel failures upon the over-speed collision with a target/wall/ground (force limitations can be set by using the e-Tutor application);
- Simulation of mechanical interaction sounds;
- Moveable bridges;
- Simulation of locks;
- Force assist tool.

5.7.2.1 Creating Ice Zone

To create an ice zone, use the following procedure:

- 1. Before the exercise start, choose the Edit\New object... menu item or click the toolbar button.
- 2. From the **Environment** category, select the **Ice Generator** object and click the **Next** button. The **General** window appears.

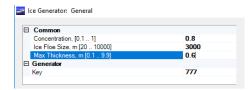


- 3. In the Common group box, set the following parameters:
 - a. In the Concentration line, set the ice concentration (0.1-1) and press <Enter>. A value of 0 means there is ice free zone, while a value of 1 means the area is completely covered with ice (compact ice).
 - b. In the Ice Floe Size line, set the maximum size of the ice floe (20-10000 m) and press < Enter>. The recommended values of the Ice Floe Size parameter depending on the size of the ice zone are shown in the table below.

Table 16. Ice floe sizes for different ice zones

ICE ZONE SIZE (METERS IN DIAMETER)	ICE FLOE SIZE (METERS)
< 1500	20-100
< 5000	100-500
< 10000	500-2000

c. In the Max Thickness line, set the maximum thickness of the ice floe (0.1-9.9 m) and press <Enter>.



- 4. If necessary, in the Generator group box, set the ice zone unique identifier in the Key field and press < Enter >.
- 5. To specify several ice floes of different sizes within the ice zone:
 - a. Click the Sizes button (the button is activated by default).
 - b. In the **Size** field, set the ice floe size in meters (the value should not exceed the maximum Ice Floe Size value.
 - c. Click the **Add** button. A new table row appears. The **Size** column shows the size of the ice floe, the **Percent** column shows the percentage of ice floe in the ice field.



d. To change the percentage of ice floe in the ice field: select required table row, click the **Change** button, set a new value in the **Percent** field and click the **OK** button.



- e. To remove an ice floe, select the required row in the table and click the Remove button.
- 6. To specify several ice floes of different thickness within the ice zone:
 - a. Click the Thickness button.
 - b. In the **Thickness** field, set the ice floe thickness (the value should not exceed the maximum **Max Thickness** value).
 - c. Click the Add button. A new table row appears. The Thickness column shows the ice floe thickness, the Percent column shows the percentage of ice floe in the ice field.

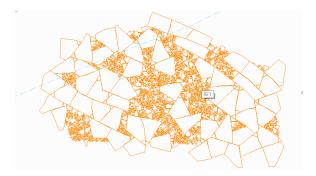


- d. To change the ice floe percentage in the ice field: select the required row in the table, click the **Change** button, set a new value in the **Percent** field and click the **OK** button.
- e. To remove an ice floe, select the required row in the table and click the Remove button.
- 7. Click the **Next** button. The **Characteristics** page appears.
- 8. In the **Sigma** field, set the value of ice bending strength (0.01-10 MPa).
- 9. In the Channel field, set the width of the channel that can be added to the ice zone.
- 10. If it's necessary to disable ice zone drifting, clear the **Drift** checkbox.

NOTE: The **Drift** option is enabled for all ice generation zones by default. In this case, all ice zones are drifting at the speed and in the direction specified on the **Ice Drift** tab of the **Overall Condition Manager** window (see "General Ice Drift Settings" on page 356).

- 11. Click the Finish button.
- 12. Click to fix the start zone point on the chart.

- 13. Move the cursor to the next point and click.
- 14. Repeat the previous action to construct all zone sections.
- 15. To complete the zone construction, right-click. The generated ice zone appears on the chart.



5.7.2.2 Editing Ice Zone

The Instructor can edit the ice zone on the chart before the start of the simulator session.

To edit the ice zone, use the following procedure:

1. Right-click any point of the zone and choose the Edit context menu item. The zone boundary appear on the chart.



NOTE: To cancel zone editing, right-click on the zone, and choose the End editing context menu item.

2. Click the zone boundary and move the cursor to the desired point on the chart.



3. Click to fix a new zone point. After some time, the zone will be regenerated and displayed on the chart.

To move the ice zone to another place on the chart:

1. Right-click any point of the zone and choose the Move context menu item.



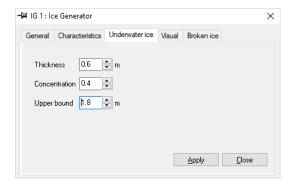
2. Move the cursor to the desired point on the chart and click. The zone will be moved to a new location on the chart.

NOTE: If the additional elements (e.g. channels, ice floes, hummocks, fractures) were added to the zone, they will be moved together with the zone.

5.7.2.3 Setting Ice Zone Properties

To set properties of the ice zone, use the following procedure:

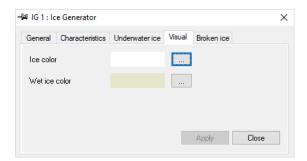
- 1. Before the exercise start, right-click any point of the ice zone. Choose the **General properties** context menu item. The **Ice Generator** properties panel appears.
- 2. On the **General** and **Characteristics** tabs, change the values of parameters as required (for more information, see "Creating Ice Zone" on page 352). Click the **Apply** button.
- 3. On the **Underwater ice** tab, set the underwater ice parameters:
 - a. In the **Thickness** field, set the underwater ice thickness (0-10 m);
 - b. In the Concentration field, set the underwater ice concentration (0, 0.1, 0.2...1);
 - c. In the **Upper bound** field, set the distance between the upper boundary of the underwater ice and water surface (0-10 m). The larger this distance, the faster the speed of the vessel.



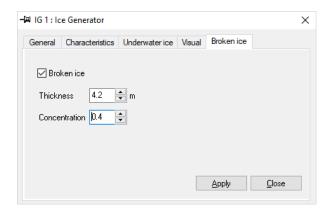
d. Click the Apply button.

NOTE: The underwater ice is not displayed on visualization.

- 4. On the Visual tab, set the ice visual parameters:
 - a. In the Ice color field, click the button. In the Color dialog box, select the ice colour within the ice zone.
 - b. In the **Wet ice** colour field, click the button. In the **Color** dialog box, select the wet broken ice colour within the ice zone.



- c. Click the Apply button.
- 5. On the Broken ice tab, set the broken ice parameters:
 - a. Select the **Broken ice** checkbox to add broken ice to free-ice areas within the ice zone. The broken ice will be displayed on the visualization.
 - b. In the **Thickness** field, set the broken ice thickness (0.1-5 m);
 - c. In the Concentration field, set the broken ice concentration (0-0.9).



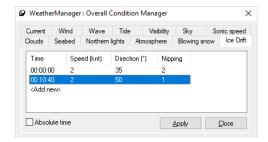
d. Click the Apply button.

5.7.2.4 General Ice Drift Settings

To specify drift settings for all generated ice zones, use the following procedure:

- 1. Choose the Edit\Environment Settings... menu item, or click the toolbar button.
- 2. In the **Environment settings** window, click the Overall Conditions... button.
- 3. In the Overall Condition Manager window, open the Ice Drift tab.
- 4. In the Speed column, set the drift velocity of ice zones.
- 5. In the Direction column, set the drift direction of ice zones.
- 6. In the **Nipping** column, set the ice compression that shows how fast the channel closes behind a vessel: 0 no compression; 1 weak compression; 2 significant compression; 3 strong compression.
- 7. Select the Absolute time checkbox if it is necessary to set the absolute exercise time.

- 8. To add the parameter change at the specified time (within 24 hours):
 - a. Double-click the <Add new>.
 - b. In the **Time** column of the new table row, enter the time when the parameter change comes into effect (counted from the exercise start moment).
 - c. Set the parameter values in the other columns.
 - d. Use a similar procedure to add any number of rows to the table. To delete a row, press the <Delete > key.



9. Click the Apply and Close buttons.

5.7.2.5 Adding Ice Pieces to Ice Zone

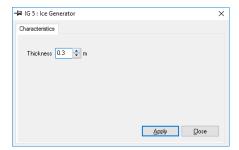
To add an ice piece to the ice zone, use the following procedure:

- 1. Right-click the ice zone and choose the Add ice piece item from the context menu.
- 2. Click the chart next to the ice zone to fix the first point of the ice piece. Draw the ice piece area by clicking and moving the cursor on the chart.
- 3. To complete the construction of the ice piece, right-click.



To adjust the ice piece properties:

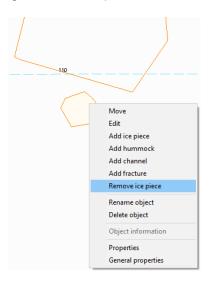
- 1. Right-click the ice piece contour and choose the **Properties** context menu item.
- 2. In the Thickness field, set the ice piece thickness.



3. Click the Apply button.

To delete an ice piece:

1. Right-click the ice piece contour and choose the Remove ice piece context menu item.

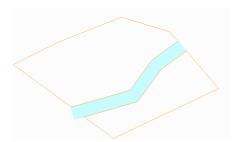


2. Click the ice piece contour or click the ice zone, the ice piece disappears from the chart.

5.7.2.6 Adding Channels to Ice Zone

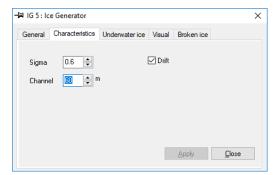
To add a channel to the ice zone, use the following procedure:

- 1. Right-click the ice zone and choose the Add channel context menu item.
- 2. Plot the line of a channel within the ice zone by clicking and moving the cursor on the chart.
- 3. Right-click to complete the channel plotting.



To change the width of the channel:

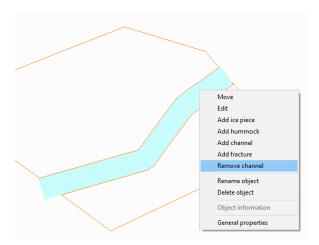
- 1. Right-click the ice zone and choose the **General properties** context menu item.
- 2. Open the Characteristics tab of the zone properties panel.
- 3. In the Channel field, set a new value of the channel width (30-100 m).



4. Click the Apply button.

To delete a channel:

1. Right-click the channel and choose the Remove channel context menu item.

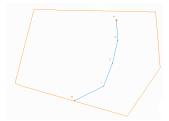


2. Click the ice zone, the channel disappears from the chart.

5.7.2.7 Adding Hummocks to Ice Zone

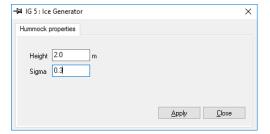
To add a hummock to the ice zone, use the following procedure:

- 1. Right-click the ice zone and choose the Add hummock context menu item.
- 2. Plot the hummock line within the ice zone by clicking and moving the cursor on the chart.
- 3. Right-click to complete the hummock plotting.



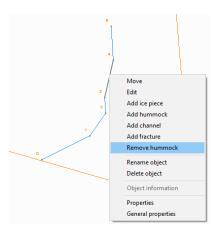
To adjust the hummock properties:

- 1. Right-click the hummock line and choose the **Properties** context menu item.
- 2. Set the following hummock properties:
 - Height hummock height (0.5-20 m);
 - Sigma value of ice bending strength (0.01-10 MPa).



3. Click the Apply button.

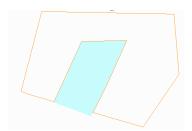
To delete a hummock, right-click the hummock line and choose the Remove hummock context menu item.



5.7.2.8 Adding Fractures to Ice Zone

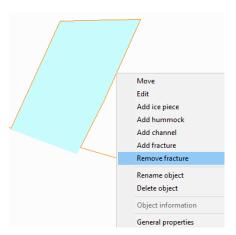
To add a fracture to the ice zone, use the following procedure:

- 1. Right-click the ice zone and choose the Add fracture context menu item.
- 2. Click the ice zone to fix the first point of the fracture. Draw a fracture area within the ice zone by clicking and moving the cursor on the chart.
- 3. Right-click to complete the fracture creating.



To delete a fracture:

Right-click the fracture contour and choose the Remove fracture context menu item.

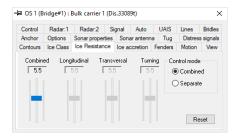


2. Click the ice zone, the fracture disappears from the chart.

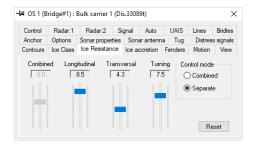
5.7.3 Control of Ice Resistance

Ice resistance can be set for any ship model after starting the exercise. To do this, use the following procedure:

- 1. On the ship properties panel, open the Ice Resistance tab.
- 2. Select the control mode: Combined (default mode) or Separate.
- 3. If the **Combined** control mode is selected, move the **Combined** slider to set the value for the common multiplier of all forces.



- 4. If the Separate control mode is selected, set values for each multiplier separately by using the sliders:
 - o Longitudinal multiplier of a longitudinal force;
 - o Transversal multiplier of a lateral force;
 - o Turning multiplier of the torque.

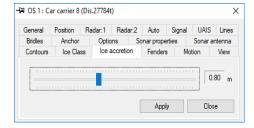


5. To reset values, click the Reset button.

5.7.4 Setting Ice Accretion

To set ice accretion, use the following procedure:

- 1. On the ship properties panel, open the Ice accretion tab.
- 2. Move the slider to set the depth of the icing layer on deck and superstructures (from 0 to 2 m).



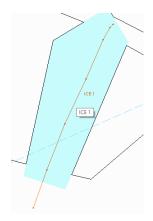
3. Click the Apply and Close buttons.

NOTE: Ship stability during the ice accretion is considerably disturbed.

5.7.5 Ice Channel Evaporation Visual Effect

To set the visual effect of the ice channel evaporation, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Environment category, select the Ice Channel Evaporation object and click the Next button.
- 3. In the Width field, set the width of evaporation area and click the Finish button.
- 4. Plot a polyline by clicking and moving the cursor on the chart, e.g., along the ice zone channel. Right-click to complete the plotting.





To remove the object, right-click the object and choose the **Delete object** context menu item.

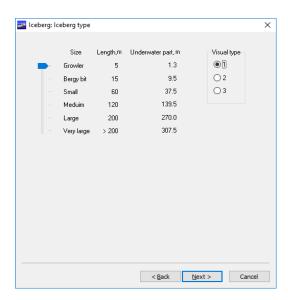
5.7.6 Operations with Icebergs

The simulator allows imitating the ship motion in the areas with icebergs.

There are six types of icebergs depending on their size, and each size type has three visual types.

To set an Iceberg object, use the following procedure:

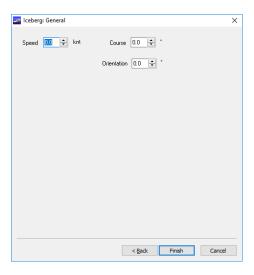
 Open the objects category window and select the Iceberg object from the Environment category. Click the Next button. The Iceberg type window appears.



- 2. Select an iceberg size category using the slider: Growler, Bergy bit, Small, Medium, Large, Very large.
- 3. Select a visual type for the selected iceberg size category: 1, 2, 3.

NOTE: To show description of all the iceberg types, see *Table 17*. You cannot change the iceberg type after setting the object on the chart.

4. Click the **Next** button. The **General** window appears.



5. Set the initial iceberg properties: Speed, Course, Orientation.

NOTE: The selected speed is only an initial characteristic and it changes during the exercise by inertia or under the effect of external conditions.

6. Click the Finish button and click on the chart.

The Iceberg object can be seen visually and is shown on the radar.

Each iceberg has one or several mooring points and can be used for towing by the own ship or target ship.



If the ship collides with an iceberg, a relevant message appears on the **Events** panel. In the case of collision of big vessel with a small iceberg, the iceberg can be automatically removed from the exercise in a few seconds.

Table 17: Overview of iceberg types

SIZE CATEGORY	UNDERWATER PART, M	ABOVE-WATER PART, M	VISUAL SKIN
			(ONE OF THREE POSSIBLE SKINS)
Growler	1.3	Less than 1 m	
(5 m)			
Bergy bit	9.5	1 - 5 m	
(15 m)			
Small	37.5	5 - 15 m	
(60 m)			
Medium	139.5	15 - 45 m	
(120 m)			
Large	270	45 - 75 m	4-4-11/19-10
(200 m)			
Very large	307.5	Over 75 m	
(over 200 m)			

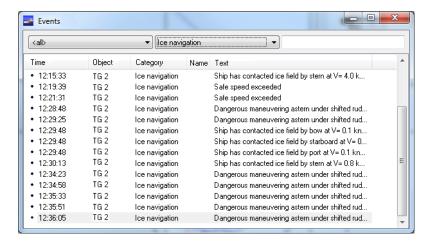
NOTE: Types of icebergs in the NTPRO are designed in accordance with the International Ice Patrol classification.

5.7.7 Monitoring Trainee Performance

Information on the trainee performance in conducting navigational operations in the ice conditions is displayed on the **Events** panel.

To view messages, use the following procedure:

- 1. Choose the View\Panels\Events... menu item.
- 2. Select the Ice Navigation category from the drop-down list.



The Events window may display the following events:

- "Safe speed exceeded" speed permitted for the given ice class exceeded;
- "Ship has contacted ice field by port/starboard/stern/bow at V=..." contact of the specified part of the ship with solid ice at a specified speed;
- "Dangerous maneuvering astern under shifted rudder";
- "Propeller impact into a dangerous zone of target vessel" propeller is in the ice ejection zone from another ship.

5.8 Fishing Operations

A fishing module as part of the Navi-Trainer Professional 6 simulator is designed to form and develop navigational and fishing professional skills, such as:

- ship handling during fishing operations (trawling, purse seining and long lining);
- · handling of fishing gear and deck machinery;
- fish-finding and control hydroacoustic devices.

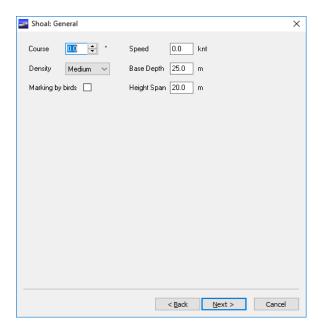
The following fishing vessels can be used in the fishing operations: Fishery training ship 1-7, Fisher, Fish boat, Seiner, Research Ship, Trawler.

5.8.1 Shoal Objects

The fish shoal model simulates the behavior of fishing objects. A fish shoal is a cluster of fish which consists of several groups of fish. Each group is able to respond to external irritants, while trying to stay in one cluster. Fish shoal can move at a defined speed and course on a plotted route.

To set the fish shoal object, use the following procedure:

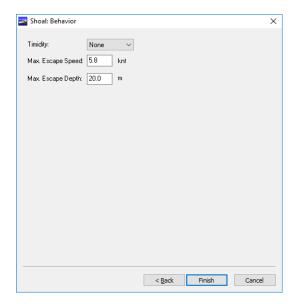
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select the Shoal object in the Fishing category and click the Next button. The General window appears.



- 3. Set the shoal motion parameters: Course and Speed.
- 4. Select the shoal density: Rarefied, Medium, Dense.

NOTE: The shoal density cannot be changed after the session start.

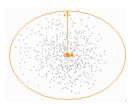
- 5. In the Base Depth field, enter the depth of the shoal's lower edge.
- 6. In the Height Span field, enter the shoal vertical dimension.
- 7. Select the Marking by birds checkbox to show birds near the shoal.
- 8. Click the Next button. The Behavior window appears.



- In the Timidity drop-down list, select the shoal timidity with regard to the passing vessels and trawls (None, Low, Medium, High, Very high);
- 10. In the Max Escape Speed field, set the maximum speed of the shoal escape, m/s;
- 11. In the Max Escape Depth field, set the maximum depth of the shoal escape, m.
- 12. Click the Finish button.
- 13. Click on the chart and move the cursor to set the object area. Click again to fix the object on the chart.



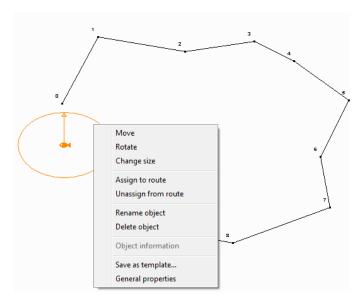
After the session start the object will be displayed as shown below:



NOTE: When all fish are caught, the Shoal object disappears from the chart.

Use the object context menu to do the following:

- move the object (Move);
- rotate the object (Rotate);
- change the object size (Change size);
- plot the route of the object (Begin route);
- assign object to the already plotted route (Assign to route);
- remove the object from route (Unassign from route);
- rename the object (Rename object);
- delete the object (Delete object);
- save object as a template (Save as template...);
- open the object properties window (General properties).



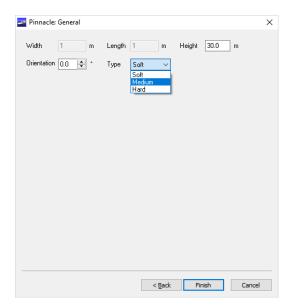
5.8.2 Pinnacle Objects

Objects of this type are designed for imitating the bottom or pelagic trawl's footrope catching on a pinnacle.

To set the object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. Select the Pinnacle object from the Fishing category and click the Next button. The General window appears.
- 3. Enter the object height in the Height field.
- 4. Enter the object orientation (by angle) in the Orientation field.

5. Select the pinnacle type (Soft\Medium\Hard) from the Type drop-down list.

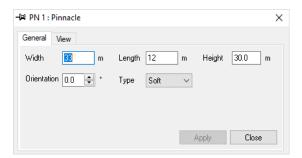


- 6. After the end of parameter input, click the **Finish** button.
- 7. Click on the chart and move the cursor to set the object area. Click again to fix the object on the chart.



To change the pinnacle properties, use the following procedure:

- 1. Open the object's context menu and choose the **General properties** item.
- 2. If necessary, change the dimensions of the object in the Wight, Length and Height fields.
- 3. In the Orientation field, set a new value of the object orientation by angle.
- 4. From the Type drop-down list, select the object type.



5. Click the Apply and Close buttons.

NOTE: Changing the pinnacle properties is not available after the simulator session start .

5.8.3 Fishing Buoys

To set a fishing buoy line, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. In the Fishing category, select the Fishing Buoy Line object and click the Next button.
- 3. In the Count field, set the number of buoys in the line and click the Finish button.
- 4. Click on the chart. The fishing line's start point appears.
- 5. Click in the following point of the fishing line.
- 6. After setting the last point of the fishing line, right-click.



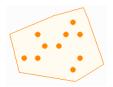
To set a single fishing buoy, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. In the Fishing category, select the Fishing Buoy Single object and click the Next button.
- 3. Click on the chart.



To create a fishing buoy zone, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. In the Fishing category, select the Fishing Buoy Zone object and click the Next button.
- 3. In the Count field, set the number of buoys in the zone.
- 4. In the Min. distance field, set the minimum distance between the buoys.
- 5. Click the Finish button.
- 6. Click on the chart to fix the zone's points.
- 7. To complete the zone construction, right-click. The fishing buoy zone appears on the chart.



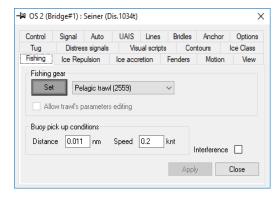
5.8.4 Setting Trawl

The Instructor can set trawl from any fishing vessel (own ship and 6 DoF ship-target model) in the course of an exercise.

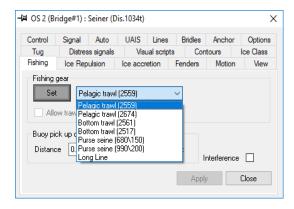
5.8.4.1 Setting Trawl from Own Ship

To set a trawl from an own ship, use the following procedure:

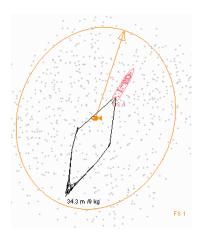
1. Start the exercise. Open the ship properties panel and select the Fishing tab.



2. Select a trawl type from the drop-down list of available fishing gear: Pelagic trawl (2559), Pelagic trawl (2674), Bottom trawl (2561), Bottom trawl (2517).



3. Click the Set button to start the trawl setting. After some time, a picture of the trawl will appear on the chart:

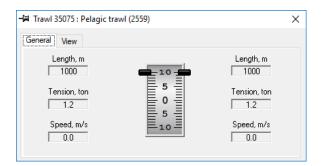


NOTE: The trawl cannot be set unless the ship is proceeding at a speed of more than 1 knot.

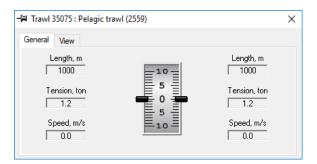
4. Select the **Allow trawl's parameters editing** checkbox, if the trainee is required to be able to edit the trawl parameters during the trawling and click **Apply**.

To control the trawl winches, use the following procedure:

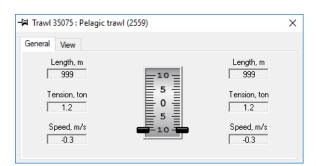
- Position the mouse cursor on the picture of a trawl on the chart and right-click. Choose the General properties
 menu item. The screen will display a trawl winch control panel. The panel will display the following information for
 each warp:
 - length, m (Length);
 - tension, t (Tension);
 - pay out/heave rate, m\s (Speed).
- 2. To pay out trawl warps, set the winch levers to the topmost position.



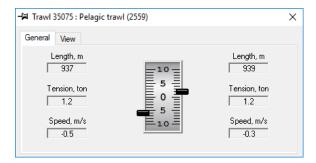
3. To stop the warp paying out, set the winch levers to the "zero" position.



4. To heave the warps, set the winch levers to the bottom position.



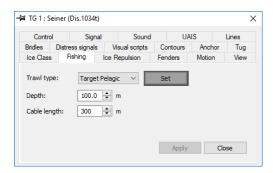
You can control the starboard and portside winches independently.



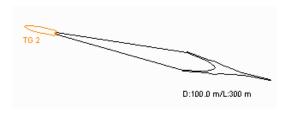
5.8.4.2 Setting Trawl from Target Ship

To set the trawl from a target ship (6-DoF model), use the following procedure:

1. Start the exercise. Open the target ship properties panel and select the Fishing tab.



- 2. Select a trawl type from the Trawl type drop-down list: Target Pelagic, Bottom target.
- 3. Set the depth in the **Depth field** and the cable length in the **Cable length** field and click the **Apply** button.
- 4. Click the Set button to set the trawl. A picture of the trawl will appear on the chart:

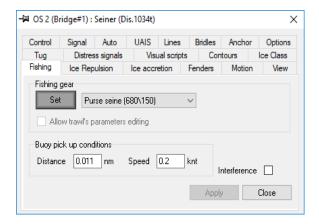


NOTE: The trawl winch cannot be operated.

5.8.5 Setting Purse Seine

The seine can be set from any fishing vessel (own ship) in the course of an exercise. To do this, use the following procedure:

- 1. Open the ship properties panel and select the Fishing tab.
- 2. Select a seine type from the drop-down list of available fishing gear: Purse seine (680\150), Purse seine (990\200).



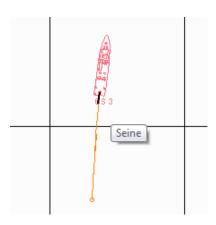
3. Set the buoy pick up conditions: Distance - maximum distance to the buoy, Speed - maximum ship speed.

NOTES: To pick up the buoy anchor:

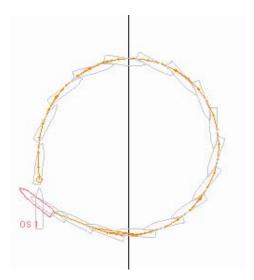
- the entire length of the seine is required to be overboard;
- the buoy is required to be on the starboard side;
- the distance to the buoy and ship speed should not exceed the preset maximum values.
- To turn on\off the interference effect, select\clear the Interference checkbox.
- 5. Click the Apply button.

NOTE: The Instructor can set the buoy pick up conditions and interference effect before the exercise start.

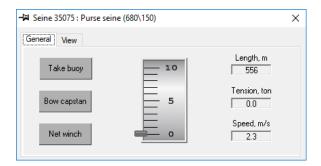
6. Click the Set button to start setting the seine. After some time, the setting will be shown on the instructor chart.



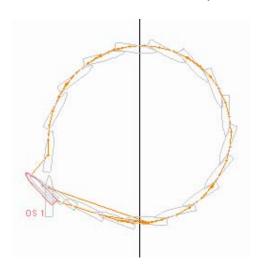
7. In the course of the seine setting, handle the ship so that after making a turning circle, the ship would approach the buoy anchor with its starboard (advisable to turn on the trend mode: View\Settings\Contours menu).



- 8. To pick up the buoy anchor on board:
 - a. Point the mouse cursor on the seine picture on the chart and right-click.
 - b. Click the **Take buoy** button on the **General** tab.



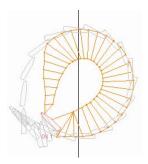
If all conditions are fulfilled, the buoy anchor will be picked up on board.



9. After the buoy has been picked up, you can start the heaving of drag ropes and pursing. To heave the drag ropes, click the **Bow capstan** and **Net winch** buttons on the **General** tab.

10. For pursing, put the seine winch lever in one of the control lever positions.

Pursing process will be displayed on the exercise chart.

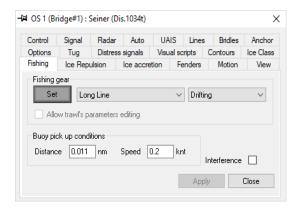


Modeling of the seine is completed automatically when the length of the purse line length is less than 150 m.

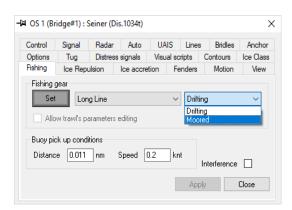
5.8.6 Setting Long Line

The long line can be set from any fishing vessel (own ship) in the course of an exercise. To do this, use the following procedure:

- 1. Open the ship properties panel and select the **Fishing** tab.
- 2. Select the Long Line from the list of available fishing gear.



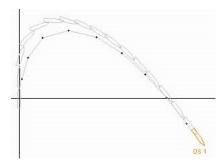
3. Select the long line type from the list: Drifting or Moored.



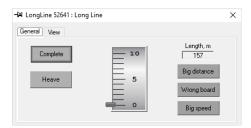
- 4. Set the buoy pick up conditions:
 - o Distance maximum distance to the buoy;
 - Speed maximum ship speed.

NOTE: To pick up the buoy: the buoy should be on the starboard side; the distance to the buoy and the ship speed should not exceed the preset maximum values.

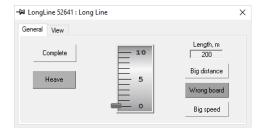
- 5. To turn on the interference effect, select the Interference checkbox.
- 6. Click the Apply button.
- Click the Set button to start the long line setting. After some time, the setting process will be displayed on the instructor chart.



8. Right-click on the long line on the chart. The screen will display the winch control panel. The panel will show the current length of the set long line:



9. After the end of the long line setting, click the Complete button. The setting of the long line will be stopped.



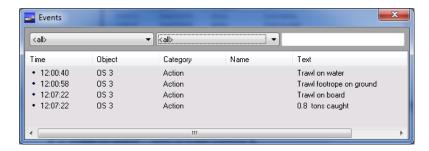


- 10. To pick up the buoy on board, the ship should approach it with its starboard side (advisable to turn on the trend mode: View\Settings\Contours).
- 11. Click the **Heave** button to pick up the buoy on board. The buoy cannot be picked up unless all the three conditions are fulfilled and none of the indicators is lighted:
- 12. To heave the long line, put the winch lever in one of the control lever positions.

Modeling of the long line is completed automatically when the length of the long line is less than 0 m.

5.8.7 Watching Trainee Performance

As a trainee is setting the fishing gear, all the winch operations are displayed at the instructor workstation. In addition, the **Events** panel (**View\Panels\Events...**) displays the events connected with the trainee handling the fishing gear:



In the case of operation with a trawl, the following events are shown:

- "Trawl on water" end of trawl setting;
- "Trawl footrope on ground" footrope touching the bottom;
- "Trawl otter doors on ground" trawl doors touching the bottom;
- "Trawl lost" alarm signifying an error which resulted in the loss of a trawl;
- "N.N tons caught" catch calculations result.

In the case of operation with a seine, the following events are shown:

- "Seine setting" command to set the seine is given;
- "Seine completing the set" end of setting the net and dragrope;
- · "Seine pursing" pursing start indication;
- "Seine on board" pursing end indication;
- "Seine lost" alarm signifying an error which resulted in the loss of a seine;
- "N.N tons caught" catch calculations result.

In the case of operation with a long line, the following events are shown:

- "Long Line setting" command to set the long line is given;
- "Long Line fishing" end of long line setting;
- "Long Line heave" long line heaving start indication;
- "Long Line on board" long line heaving end indication;
- "N.N tons caught" catch calculations result.

5.9 Search and Rescue Operations

To ensure simulator training in the search and rescue (SAR) operations as per the requirements of International MERSAR and IAMSAR Conventions, the simulator has a number of objects for practising search and rescue operations.

The following functionality is implemented:

- Giving signals in compliance with the International Code of Signals (ICS-69) by using flags, flashing light (with appropriate visual effects) and sound signals;
- Giving distress signals associated with appropriate visual effects;
- UAIS functionality;
- Integration with GMDSS Simulator TGS 5000 (transmission of ship position coordinates, speed, local time, SART and radar operation, reception and transmission of messages to Navtex Manager utility (NS 3000 ECDIS-I) for their further processing).

5.9.1 SAR Objects

The objects from the Rescue category can take part in SAR operations.

Rescue category objects are added to the exercise by using standard procedures. The Instructor can use all these objects at the exercise preparation stage and during the exercise, adding them at the necessary time in the required place of the scenario. Besides, SART and Life buoy objects can be thrown overboard by the Trainees themselves: the former by using the GMDSS work station, the latter by using the Conning Display.

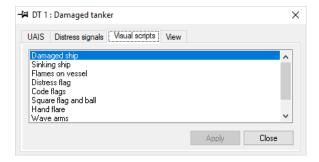
NOTE: Helicopters from the Helicopter category can also take part in SAR operations (see "Helicopter Initial Parameters" on page 403).

This section describes specific features of such Rescue objects as: Damaged Tanker, Damaged fish boat, Overturned small boat, Fire objects, EPIRB, AIS SART and SAR routes.

5.9.1.1 Damaged Tanker

To set a visual script of a Damaged tanker, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the **Rescue** category, select the **Damaged tanker** object, set its orientation, click **Finish** and place the object on the chart.
- 3. Open the object properties panel.
- 4. Select the Visual scripts tab.



- 5. Select the damaged tanker visual script from the list:
 - o Code flags tanker with a distress signals as per the International Code of Signals (November, Charlie);
 - Damaged ship tanker with fire in the stern part;
 - Distress flag tanker with a distress flag;
 - Flames on vessel tanker with a burning barrel on the stern;
 - Hand flare tanker with a person holding a hand flare on the stern;
 - Safe tanker which does not move relative to the water;
 - Sinking ship tanker with a starboard list and slight forward trim;
 - Square flag and ball tanker with a distress signal which is a square flag with a ball under it.
 - · Wave arms tanker with a person waving arms on the stern.
- 6. Click the Apply and Close buttons.

The Damaged tanker object also can send the UAIS messages and give distress signals.

5.9.1.2 Damaged Fish Boat

For the Damaged fish boat object it is possible to set the orientation, visual states, switch on the UAIS transponder and specify AIS data.

To set visual states of the object, perform the following operations:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the **Rescue** category, select the **Damaged fish boat** object, set its orientation, click **Finish** and place the object on the chart.
- 3. Open the properties panel of the object and select the Distress tab.

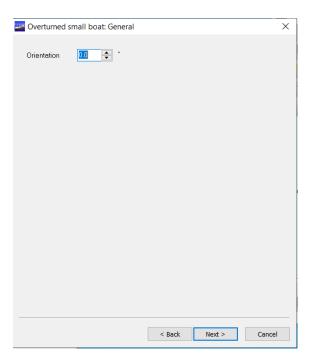


- 4. Select the necessary state (several states can be set up simultaneously):
 - Hull damage burnt hull of the ship;
 - Smoke smoking boat;
 - o Barell smoking barrel on the ship;
 - Distress flag distress flag on the ship;
 - Man man is waving on board.
- 2. In the Fire group box, select the fire type on the ship if necessary:
 - o None no fire;
 - Moderate fire moderate fire (separate flashes);
 - Extensive fire strong fire;
 - Extreme fire maximum fire.
- 2. Click the Apply and Close buttons.

5.9.1.3 Overturned Small Boat

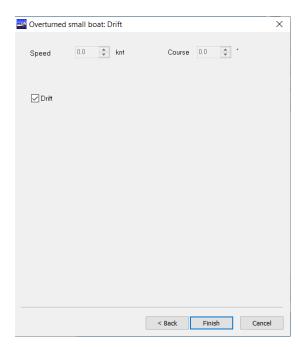
To set the Overturned small boat object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the Overturned small boat object.
- 3. Set the object orientation and click Next.



NOTE: Changing the object orientation during the simulator session will not be available.

4. Leave the **Drift** checkbox selected if you want the object to drift, or clear the **Drift** checkbox and enter the object's course and speed in the **Course** and **Speed** fields.



5. Click Finish and place the object on the chart.



5.9.1.4 Fire

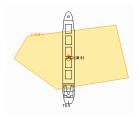
The Fire object can be set in any point on the chart, or can be attached to a stationary or movable exercise object.

To set and assign properties to an object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the Fire object.
- 3. Set the object initial parameters:
 - Flame height (1-50 m);
 - Intensity flame intensity, %;
 - o Smoke.
- 4. Click the Finish button and draw a fire zone on the chart.



To attach a Fire object to an exercise object, open the fire zone context menu, choose the Attach to object item and click on the object which the Fire object should be attached to.



To detach the Fire zone from the object, open the fire zone context menu and choose the Detach from object menu item.

5.9.1.5 EPIRB

The Emergency Position Indicating Radio Beacon (EPIRB) object is designed for determining position of objects in distress and sending a relevant notification signal.

After connection to the power supply, the EPIRB sends an encoded VHF signal which can be received by special VHF/UHF direction finders on the sea ships and aircraft closest to the distress position.

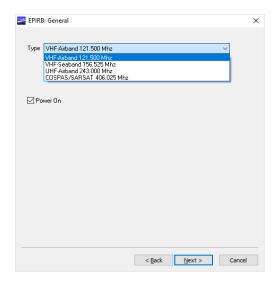
NOTE: COSPAS/SARSAT radio beacon transmission on 406 MHz frequency can automatically generate and store encoded digital messages with the MMSI data of the ship or helicopter in distress.

In the NTPRO simulator, the Instructor can set the EPIRB in an exercise on any ship or helicopter, or drop it into the water at any moment of time. To measure the EPIRB distress signal bearing and to decode it, SAR DF direction finder is used, which is included in the SAR panel equipment of the training bridge Conning Display.

NOTE: The full mission simulator training in operating a direction finder (taking direction to the VHF exchange and receiving coordinates of the ship in distress from the COSPAS/SARSAT satellites) is possible in the NTPRO's joint operation with the GMDSS simulator.

To set the EPIRB object, use the following procedure:

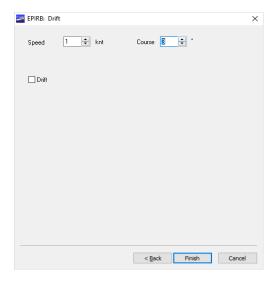
- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the EPIRB object and click the Next button.
- 3. Select the distress signal frequency from the Type drop-down list.



4. To turn on/off the radio beacon power supply, select the **Power On** checkbox.

NOTE: If the EPIRB power supply is turned on at the exercise creation stage, its distress signal will be transmitted immediately after the exercise start. The same can be done later when the exercise is being fulfilled by the Trainee.

- 5. Click the Next button.
- 6. If necessary, set the radio buoy speed and course. To do this, clear the **Drift** checkbox and enter values in the **Speed** and **Course** fields respectively.



7. Click the Finish button and place the object on the chart.

8. To attach an EPIRB to the ship or helicopter, open the EPIRB context menu, choose the **Attach** item and left-click on the ship or helicopter contour.



9. To detach an EPIRB from the object, open the EPIRB context menu and choose the Drop item.

NOTE: After the radio beacon is detached from the object, the object data (MMSI) will be stored in the device memory.

- To move the EPIRB to another place, open the EPIRB context menu and choose the Move item (the EPIRB object should be detached from another object).
- 11. To hide the EPIRB on visualization and Radar, open the EPIRB context menu and choose the **Hide** item. The EPIRB remains in the object list and can be shown by choosing the **Show** item in its context menu.

NOTE: The Hide / Show function is also available for the MOB, LifeRaft and SART objects.

5.9.1.6 AIS SART

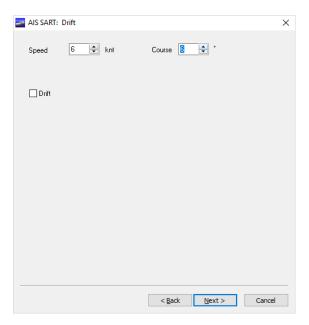
AIS SART is an AIS emergency transmitter used to give off a secondary alarm when search and rescue units are searching for a life raft/lifeboat in distress.

AIS SART includes a built-in GPS which will help the units to pinpoint exactly where the distressed boat is located in a larger area. This is done with the help of the AIS on the searching ship or helicopter. When started, the AIS SART sends its position data in an ordinary AIS message.

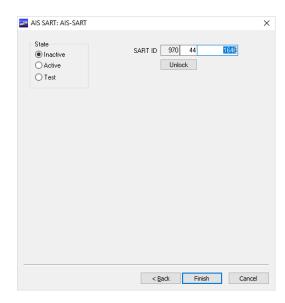
The information sent by the AIS SART can be seen on the simulator radar, Navi-Sailor and UAIS MKD.

To set an AIS SART object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the AIS SART object and click the Next button.
- If necessary, set the device speed and course. To do this, clear the Drift checkbox and enter values in the Speed and Course field respectively.



- 4. Click the Next button.
- 5. Select the device state:
 - o Inactive the device is switched off.
 - Active the device is switched on and ready for operation.
 - Test the device is used in the test mode.



- 2. If necessary, change the SART ID data:
 - a. To change the Device type, click the **Unlock** button and enter a new value in the first field.



NOTE: During the simulator session the Device type can be changed if the AIS SART is in the "Inactive" state.

- b. To change the Manufacture's number, enter the new value in the second field.
- c. To change the Device serial number, enter the new value in the third field.
- 7. Click the Finish button and click the left mouse button on the chart. The object appears on the chart.



To change the AIS SART properties, open the object's context menu and choose the **General properties** menu item. The object properties window appears.



NOTE: Such objects of the Rescue category as Life buoy, Man over board and partly Life Raft have the same AIS SART properties.

5.9.1.7 SAR Routes

SAR operations are performed by search-and-rescue services in compliance with the recommendations of the IMO "IAMSAR manual "Planning and Conducting the Search" 2008 Edition".

With consideration for these recommendations, the Navi-Trainer Instructor enables the creation of four search-andrescue patterns:

- Track line search (see "SAR Track Line Search" on page 386).
- Parallel sweep search (see "SAR Parallel Sweep Search" on page 387);
- Expanding square search (see "SAR Expanding Square Search" on page 388);
- Sector search (see "SAR Sector Search" on page 389).

Both an own ship and target ship can be assigned to the SAR route.

NOTE: An object assignment to a route of a search-and-rescue operation is performed similarly to a ship assignment to any other route (see "Assigning Object to Route" on page 121).

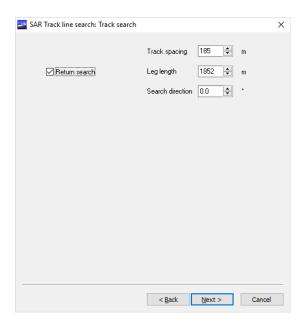
SAR Track Line Search

The search pattern is mainly used for the search of lost ships and aircraft if their estimated route is known. The search is organized along the estimated route of the lost object. The search can be one-way along the route of the lost object or return-way back to the start point. Usually such a search pattern is used at the beginning of rescue operation.

To set the SAR track line search object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the SAR Track line search object and click the Next button.
- 3. Set the following parameters:
 - Track spacing distance between two adjacent parallel tracks, m;
 - Leg length length of the route leg, m;
 - o Search direction direction of search, deg.

4. Select the Return search checkbox, if you need to set a search with a return to the search start point.



- 5. Click the Next button.
- 6. Set the following parameters:
 - o Drift drift velocity of a search object with consideration for currents and wind in the search area;
 - Set drift direction of a search object;
- 2. Click the Finish button and click the desired point on the chart. The route appears on the chart.

An example of a search route with return to the search start point:



An example of a search route without return to the search start point:



SAR Parallel Sweep Search

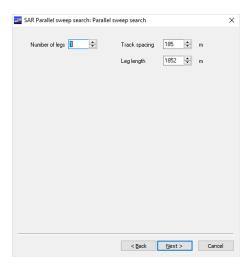
A parallel sweep search pattern is used at large areas when there is no reliable information about the location of the search object.

Large search areas are usually divided into sections where an independent search is performed simultaneously with different means.

To set the SAR parallel sweep search object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the SAR Parallel sweep search object and click the Next button.
- 3. Set the following parameters:

- Number of legs number of route legs;
- Track spacing distance between two adjacent parallel tracks, m;
- Leg length length of the route leg, m.



- 4. Click the Next button.
- 5. Set the following values:
 - Drift drift velocity of a search object with consideration for currents and wind in the search area;
 - Set drift direction of a search object.
- 6. Click Finish and click the desired point on the chart. The route appears on the chart.

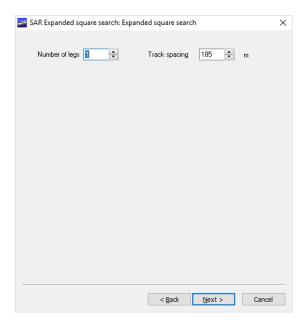


SAR Expanding Square Search

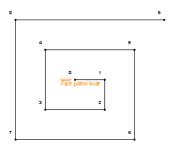
An expanding square search pattern is used if the position of the searched object is known rather precisely. The search start point concurs with the point of the estimated location of the search object. Such a pattern suits search made by using small ships, people or water craft with a weak drift. The first leg is usually oriented against the wind to minimise navigational errors.

To set the SAR expanding square search object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the SAR Expanding square search object and click the Next button.
- 3. Set the following parameters:
 - o Number of legs number of route legs;
 - Track spacing distance between two adjacent parallel tracks.



- 4. Click the Next button.
- 5. Set the following parameters:
 - Drift drift velocity of a search object with consideration for currents and wind in the search area;
 - Set drift direction of a search object.
- 6. Click the Finish button and on click the desired point on the chart. The route appears on the chart.

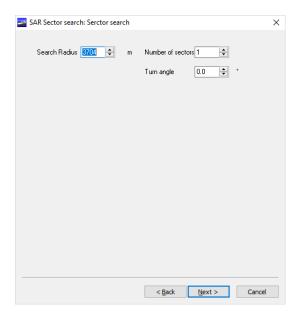


SAR Sector Search

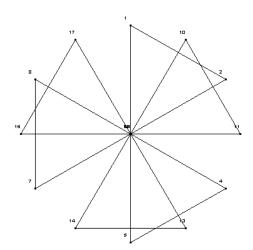
A sector search pattern is most efficient if the position of the searched-for object is precisely known, and the search sector is small. The route is used for search in a circular area with the centre in the point of the search object estimated position. With such a search pattern, only one rescue airplane or one helicopter together with one ship can be used for independent search in the same area. The search radius for ships is usually from 2 to 5 nautical miles.

To set a SAR sector search object, use the following procedure:

- 1. Choose the Edit\New object... menu item or click the toolbar button.
- 2. From the Rescue category, select the SAR Sector search object and click the Next button.
- 3. Set the following parameters:
 - Search Radius.
 - Number of sectors;
 - Turn angle turn angle of the search pattern if there are more than three sectors.



- 4. Click the Next button.
- 5. Set the following parameters:
 - Drift drift velocity of a search object with consideration for currents and wind in the search area;
 - Set drift direction of a search object.
- 6. Click the Finish button and click the desired point on the chart. The route appears on the chart.



5.9.2 Flag Signalling

ICS flags are hoisted on two halyards (portside and starboard): the station callsign on one of them, and a message or answering pennant on the other. The message may consist of groups (from one to six flags). Groups are separated from each other with a tackline. Therefore, the sufficient maximum number of hoisted objects (flags plus tackline) is seven.

NOTE: For some models of naval ships in the simulator it is possible to give NATO signals in addition to the ICS signals; they can be hoisted on six halyards (three on each side).

The transmitting stations may be:

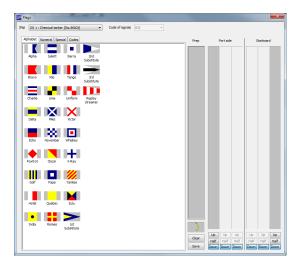
- Own ships/tugs control exercised by a Trainee on the bridge from the Conning Display, or by the Instructor from the Flags panel (View\Panels\Flags);
- Target ships/tugs (6 DoF) control by the Instructor from Flags panel.

The Flags panel contains the following tabs:

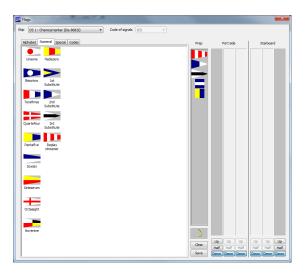
- Alphabet tab: consists of 26 alphabetic flags + 3 substitutes + 1 answering pennant;
- Numeral tab: consists of 10 numeral pennants + 3 substitutes + 1 answering pennant;
- Special tab: consists of 9 special flags;
- Codes tab: consists of flag sets signifying messages commonly used during rescue operations. There are 6 of them in the delivery set, but this number can be increased by the Instructor.

To work with the Flags panel, use the following procedure:

1. Choose the View\Panels\Flags... menu item, or click the toolbar button. The Flags window appears.



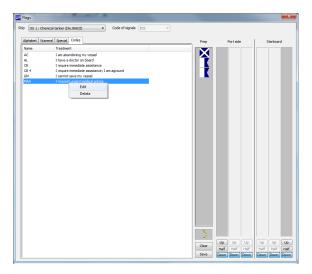
- 2. From the **Ship** drop-down list, select the name of the own ship or target ship.
- 3. From the **Code of signals** drop-down list, select the code of signals (if it is possible). After the ship selection, buttons for hoisting and lowering flags on the halyards will be activated.
- 4. To form a signal in the Prep column:
 - a. Open the Alphabet or Numeral, or Special tab of the Flags panel. Double click on the flag picture, or drag it to the Prep column. If more than 7 flags are set to the Prep column, the topmost flag will be automatically deleted.
 - b. If it is necessary to set a tackline, click on the picture and drag it to the Prep column.
 - c. To remove a flag, double click on its picture in the **Prep** column, or drag it back to the left part of the **Flags** panel.
 - d. To remove all the flags from the Prep column, click the Clear button.



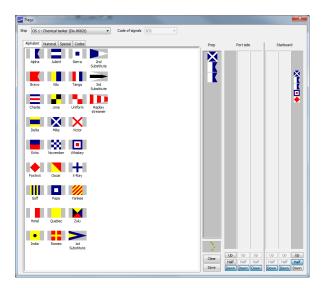
- 5. To save the formed signal:
 - a. Click the Save button.
 - b. In the Flag group properties box, enter the signal treatment in the Treatment field and click the OK button.

NOTE: Work with flag signals is recorded to a log file for the subsequent debriefing.

- 6. To select a ready flag combination:
 - a. Open the Codes tab on the Flags panel.
 - b. Click on the flag combination row.
 - c. To edit or delete a flag combination on the **Codes** tab: right-click on the flag combination row and choose the **Edit** or **Delete** menu item.



- 7. To hoist a signal on the halyard:
 - Click the Up up button to hoist the signal "close up".
 - Click the Half Half button to hoist the signal "at the dip".
- 3. To lower the signal, click the **Down** button.



5.9.3 Giving Distress Signals

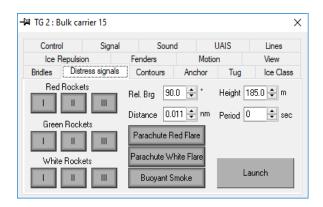
This category includes the following signals:

- Red, white, or green rockets or shells;
- · Parachute red or white flares;
- Buoyant signals giving off a volume of orange-coloured smoke.

These signals can be given from the own ships by the Trainees from the Conning Display (SAR->Distress Signal page), and from all the ship types by the Instructor from the appropriate object property tab.

To send distress signals, the Instructor should use the following procedure:

1. Open the ship properties panel and select the Distress signals tab.



- 2. Select the required signal by clicking the appropriate button.
- 3. Adjust launch parameters for rockets:
 - Rel. Brg. shooting direction relative to the ship bow;
 - Distance distance from the ship at which the rocket will fall down;
 - · Height rocket ascent height;
 - o Period how often the rocket launching will be repeated. 0 value single launch.
- 4. Click the Launch button for giving distress signals.

The **Launch** button is highlighted in white. When a single distress signal is given, the **Launch** button is released on its own. Otherwise, it is for the Instructor to decide when it should be released in order to stop a series of distress signals, as this series duration is not limited in any other way.

NOTE: The signal rocket luminescence duration is 5 seconds. The parachute red flare is lighted for at least 40 seconds, smoke signal - 3 minutes. The time delay between the signal launch command and its visual presentation is 1-2 seconds.

Work with distress signals is recorded to a log file for the subsequent debriefing.

5.9.4 Use of Rescue Boat

In the training the Instructor can perform the rescue boat lowering and hoisting operations.

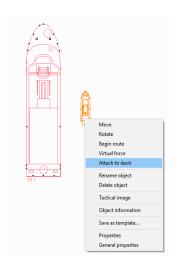
These operations are possible with the use of special ship mathematical models, e.g., IMT 992 type ship as a rescue boat carrier ship, and Offshore Rescue Vessel for the rescue boat itself. At least one of them should be an own ship depending on the exercise purpose.

NOTE: If you are also planning to lower and hoist a lifeboat on the relevant target ships, they should be added in the exercise as 6-DoF mathematical models.

5.9.4.1 Attaching Rescue Boat to Davit

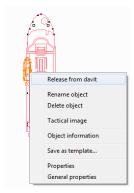
To attach the rescue boat to a davit of the carrier ship, use the following procedure:

1. Open the rescue boat context menu and choose the Attach to davit item.



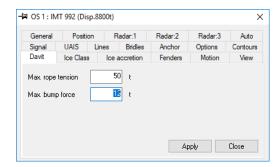
2. Click to the carrier ship on the chart. The rescue boat will be attached to the carrier ship davit.

If it is necessary to return to the autonomous sailing of the rescue boat, choose the **Release from davit** context menu item and then choose the **Move** context menu item.



To set the davit properties:

- 1. Open the carrier ship properties panel and select the **Davit** tab.
- 2. In the Max rope tension, set the maximum tension force for ropes.
- 3. In the Max bump force, set the maximum force of the boat collision with the carrier ship side.



4. Click the Apply button.

5.9.4.2 Lowering and Recovering Rescue Boat

The instructor procedures for assigning an exercise to the bridges are in no way different from the standard procedures.

Own carrier ship is assigned on the main bridge, whereas the rescue boat- to the rescue boat bridge in accordance with the joint sailing scenario.

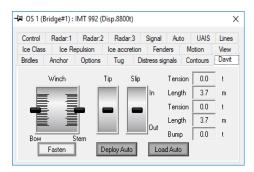
Main recommendations for organising this type of simulator training:

- If the rescue boat is on board the carrier ship in the exercise, after the exercise start, the crew of the rescue boat should be engaged in the VHF exchange with the crew on the carrier ship bridge, awaiting lowering on the water;
- On the main bridge, there is a dedicated visual channel for observing the davit. The captain on this bridge gives commands to lower (or hoist) the rescue boat by using the intercom. However, these commands are fulfilled by the Instructor;
- If in the course of the exercise it is necessary to hoist a rescue boat on board the carrier ship, its crew advises
 the carrier ship crew on the VHF, about the readiness to fasten the bow and stern tackle. These commands are
 also executed by the Instructor.

Lowering Rescue Boat

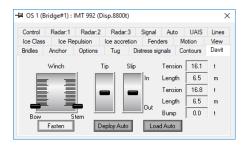
After receiving the "Prepare and lower the portside rescue boat on the water" command from the main bridge via the Intercom, the Instructor uses the following operations:

1. Open the carrier ship properties panel and select the Davit tab.



NOTE: As the rescue boat is on board this ship, and tackle on the rescue boat is fastened, the **Fasten** button is lighted.

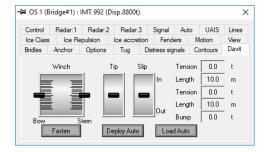
- 2. Move the Slip handle down in order to shift the rescue boat to the side of carrier ship.
- 3. Move the **Tip** handle down to tip the davit over the side of the carrier ship.
- Finally, move down both Winch handles together (the mouse cursor should be in the middle between the two handles).



5. As the bottom of the rescue boat touches water surface, the Instructor should, especially if there are waves, apply brakes separately to the bow winch (the mouse cursor should be positioned on the **Bow** winch handle) and to the stern winch (the mouse cursor should be positioned on the **Stern** winch handle).

NOTE: The winch control should be executed at commands given from the main bridge.

6. At the moment when the captain of the first bridge, as agreed with the captain on the rescue boat bridge, finds it safe, the Instructor lets go the bow and stern tackle by clicking the glowing **Fasten** button.



To lower a life boat in the automatic mode, click the **Deploy Auto** button.

To release the lifeboat in the process, click the Fasten button.

NOTE: The Instructor can monitor the fulfilment of these operations on the visualization monitor, setting the video camera next to the davit. Dangerous letting go and dangerous bump of a rescue boat is automatically recorded by the instructor program in the **Events** panel.

Recovering Rescue Boat

Before the rescue boat approaches the side of the carrier ship, the davit should be tipped and bow tackle should be lowered so as to be half a meter above the water surface if this was not done before.

The simulator automatically checks the distance of the rescue boat bow and stern approach to the tackle, and will not allow the **Fasten** button to be activated if the distance is large (more than 1 m). As the command to fasten the tackle is given from the main bridge (after an appropriate VHF request from the rescue boat bridge), the ill-timed **Fasten** command is recorded by the instructor program in the **Events** table.

If the tackle fastening is successful, the Instructor can recover the rescue boat:

- 1. Move up the **Winch** handles. The tackle is heaved and the rescue boat is hoisted. As the tackle is heaved to its proper position, the winch handles will automatically take the middle position.
- Move up the Tip handle. The davit is topped to the stowed position. As this position is reached, the Tip handle automatically returns to the middle position.
- Move up the Slip handle. The davit is shifted to the stowed position. As this position is reached, the Slip handle automatically returns to the middle position.

To recover the rescue boat automatically on the carrier ship:

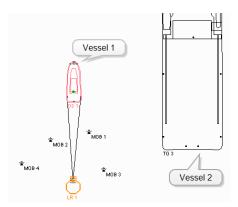
- 1. Click the **Deploy Auto** button to lower the davit.
- 2. Click the Fasten button.
- 3. Click the Load Auto button.

5.9.5 Life Raft Operations

NTPRO provides procedural rescue training with the life raft.

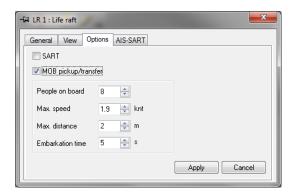
The following objects are involved in the training procedure:

- MOB (man over board) objects of the Rescue category.
- Life raft object of the Rescue category.
- Own ship ("Vessel 1") small craft, e. g., Rescue vessel that tows a life raft.
- Target ship ("Vessel 2") vessel to which people are disembarked from the life raft.



To pick up people on board the life raft and then transfer them to the vessel, use the following procedure:

- 1. Connect "Vessel 1" to the life raft with towlines.
- 2. Open the Option tab of the life raft properties panel and select the MOB pickup/transfer checkbox.



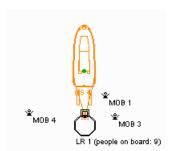
- 3. In the **People on board** field, set the initial number of persons on board the life raft (8 people by default). The life raft capacity is up to 12 people.
- 4. Set the necessary conditions for the picking up people from the water to the life raft and for the disembarkation of people from the life raft to "Vessel 2":
 - o Max.speed the maximum speed of life raft relative to the MOB and to "Vessel 2".
 - Max. distance the maximum distance from the life raft to the MOB and to "Vessel 2".
 - o Embarkation time time it takes a person to get on the life raft and "Vessel 2".
- Click the Apply button.

NOTE: The life raft options can be changed after the start of the simulator session.

- 6. Assign the exercise to the bridge and start the exercise.
- 7. During the life raft towing, move the life raft to the MOB at a distance sufficient for the MOB pickup. The MOB will automatically be picked up into the life raft.



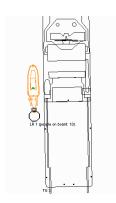
The number of people on board is shown next to the life raft contour on the chart.



If there are 12 people on the raft, the pickup of the MOBs stops.



8. Move the life raft next to "Vessel 2" to start the transfer of passengers from the life raft to "Vessel 2". Passengers will disappear from the life raft.



When the last person is transferred to "Vessel 2", the life raft will disappear after a time.

5.9.6 Using Lifesling for MOB Recovery

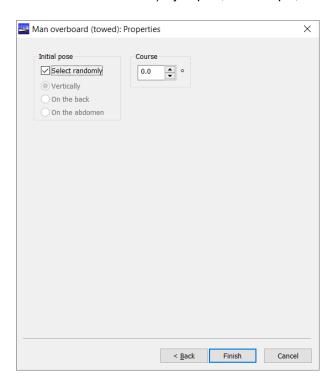
To use a lifesling for recovery of a man overboard (MOB), follow these steps as described in the example below:

- 1. Set an own ship or ship target (e.g., Safe boat 1).
- 2. Set the Man overboard (towed) object:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the Rescue category, select the Man overboard (towed) object, and click Next.
 - c. In the **Initial pose** group box, remove the **Select randomly** checkbox and select the desired initial MOB pose: **Vertically**, **On the back**, **On the abdomen** (see table below).

Table 18. Available Man Overboard Poses



Or leave the **Select randomly** checkbox enabled to randomly select the MOB initial pose (Vertically/On the back/On the abdomen) if you plan, for example, to use several MOBs with different poses in the exercise.

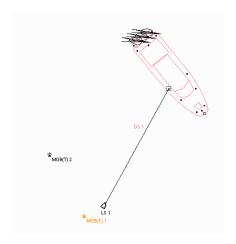


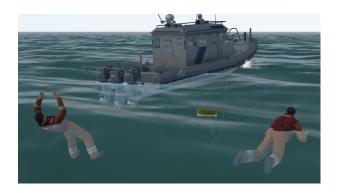
NOTE: The pose of the MOB can be changed in the MOB properties panel during a simulator session.

- d. Select the MOB's initial course in the Course field.
- e. Click Finish and place the object on the chart.
- 3. Set the Lifesling object:
 - a. Choose the Edit\New object... menu item or click the toolbar button.
 - b. From the Rescue category, select the Lifesling object, and click Next.
 - c. Set the object orientation.
 - d. Click Finish and place the object on the chart.
- 4. To attach the lifesling to the ship, click the lifesling object's mooring point and then click any mooring point on the ship.



- 5. Start the exercise.
- 6. When maneuvering the vessel, approach the MOB so that the lifesling is as close to the MOB as possible.





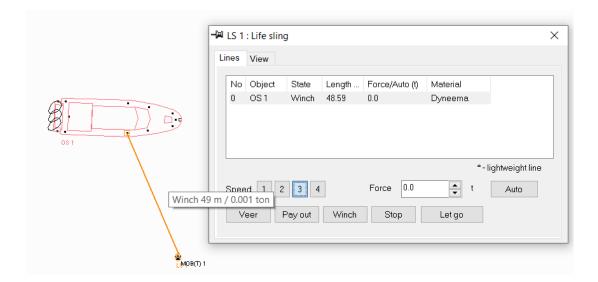
7. When the MOB is within one meter of the lifesling, the MOB will automatically get into the lifesling. The event message Acquired the lifesling 'LS_name' appears on the Events panel. Turn off the engine(s) as soon as the MOB has acquired the lifesling.





NOTE: When the MOB is in the lifesling, the MOB object and the lifesling object cannot be deleted.

- 8. To pull the MOB to the ship:
 - a. Open the lifesling object properties panel.
 - b. On the Lines tab, select a mooring line in the table and click the Winch button to heave the line.
 - c. To switch the winch speeds, use the 1234 buttons (1 the slowest winch speed, and 4 the fastest winch speed).



NOTE: If the MOB is pulled to the ship too quickly, the MOB changes the initial pose and starts swinging arms.

9. When the MOB is within one meter of the ship, it will be automatically hoisted aboard. The MOB disappears from the chart and visualization screen. The event message <code>Successfully recovered by Ship_name</code> appears on the Events panel.

If the vessel hits a MOB:

• the appearance of the MOB on the visualization screen has changed:



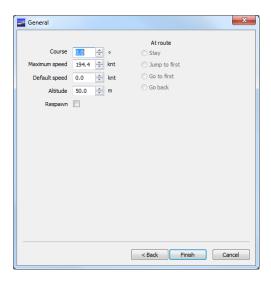
- the event message <code>Hit by OS_name</code> appears on the Events panel
- the MOB object cannot be moved and attached to the lifesling
- properties of the MOB object cannot be edited.

5.10 Operations with Helicopters and Aircrafts

5.10.1 Helicopter Initial Parameters

To set the Helicopter object initial parameters, use the following procedure:

1. Open the list of object categories, select the helicopter type in the Helicopter category and click the Next button.



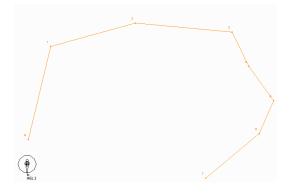
2. Set the initial helicopter parameters: Course, Maximum speed, Default speed, Altitude.

NOTE: The **Respawn** option allows you to restore the helicopter in a minute if it has been hit (in this case, the shooting functionality should be enabled).

3. Click the Finish button and click on the chart.

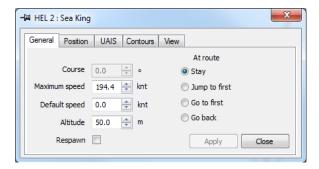
To plot the helicopter route and set the properties of route waypoints and segments, use the following procedure:

- 1. Open the object context menu and choose the **Begin route** item.
- 2. Click on the chart to fix the first route waypoint. Move the cursor on the chart and click to fix other waypoints.
- 3. Right-click to complete the route plotting.

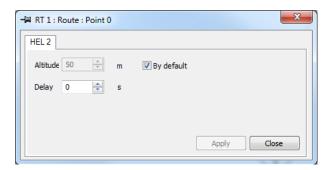


4. Open the helicopter properties panel.

- 5. In the General tab, in the At route group, set actions in the end waypoint:
 - Stay to stay in the end waypoint;
 - o Jump to first to jump to the first waypoint;
 - Go to first to return to the first waypoint by the shortest route;
 - o Go back to go back along the same route.



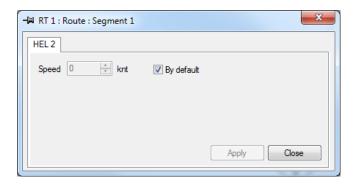
- 6. Click the Apply and Close buttons.
- 7. Set actions in the route first waypoint:
 - Right-click the route first waypoint and choose the Properties item. The route first waypoint properties panel appears.



- b. To specify the default helicopter altitude over the first waypoint: clear the **By default** checkbox and set the altitude value in the **Altitude** field.
- c. If necessary, in the **Delay** field, set the time of the helicopter staying in this waypoint.
- d. Click the Apply and Close buttons..

NOTE: Use the same procedure to set the properties of other helicopter route waypoints.

- 8. Set actions in the route segment:
 - a. Right-click the route segment and choose the **Properties** item. The route segment properties panel appears.



- b. To specify the helicopter speed on the first route segment, clear the **By default** checkbox and set a new value in the **Speed** field.
- c. Click the Apply and Close buttons..

NOTE: Use the same procedure to set the properties of other helicopter route segments.

5.10.2 Helicopter Control

The helicopter is controlled from the Helicopters panel of the Instructor workplace.

To open the panel, choose the View\Panels\Helicopters... menu, or click the toolbar button.



The Helicopters panel contains the following controls and indicators:

- Choose helicopter drop-down list for choosing a helicopter.
- Control type group box contains option buttons for selecting the helicopter flight mode:
 - By Predefined Route the helicopter leaves automatically for the closest point of the route plotted by the Instructor and then follows this route.
 - Automated Flight the helicopter performs the following operations in an automated flight mode: taking-off, landing, HIFR, Vertrep, medical evacuation, moving to any on-chart point set by the Instructor.
 - Manual Control manual control of the helicopter by entering numeric values of flight Course, Speed, and Altitude, or by using the joystick.
- Automated Flight group box contains the following controls:



 Prepare for operation button with the drop-down list for selecting the type of operation: Land from port side, Land from astern side, HIFR, Vertrep, Medevac. The button name changes depending on the selected operation.

NOTE: The HIFR, Vertrep and Medevac operations are available if the helicopter model supports these functions.

- Go to point button for specifying the on-chart point where the helicopter will be proceeding.
- o Cancel action button for cancel the operation.

- Manual Control group box contains controls for helicopter manual control. Choose one of the following options:
 - Select the Use Joystick checkbox and use the joystick to perform the following operations: approach, landing, turn to the right/left on the deck, take-off, departure from the ship.
 - Set the Altitude, Speed, and Course values and click the Apply button.
- **Helicopter status** group box contains buttons for controlling the helicopter take-off and landing in the automated flight mode (for more information, see "Landing and Taking-Off of Helicopter" on page 407):



- o Start start engine;
- Engage engage rotors;
- Remove Lashings remove lashings;
- Launch take off;
- Land On land-on;
- Lash Down install lashings;
- Disengage disengage rotors;
- o Shut Down engine shutdown.

NOTE: The buttons are not available until the helicopter has the required status. Under these buttons, the current helicopter status is displayed ("Waiting For Command", "Preparing for Landing", "Ready To Land", "Ready To Launch" and others).

- HIFR group box contains buttons that can be used in the Helicopter Inflight Refuelling operations.
- Landing scheme group box contains the Choose scheme drop-down list for selecting a landing scheme (only
 default landing scheme is available in the simulator).
- · Helicopter flight indicator.

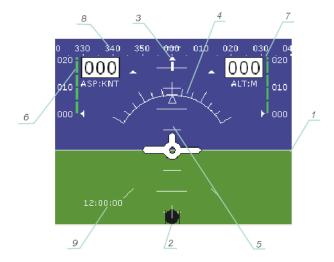


Table 19. Flight indicator components

Nº	COMPONENT
1	Horizon line
2	Slip Ball
3	Turn Rate
4	Roll (scale division is 5 degrees)
5	Pitch (scale division is 5 degrees)
6	Digital Airspeed, Airspeed Tape
7	Digital Altitude, Altitude Tape
8	Magnetic Heading
9	Time on board

- Landing platform conditions group box (the setting of landing platform conditions is not implemented in the simulator).
- Navigation info indicators show the following flight information:
 - LAT latitude;
 - LON longitude;
 - o COG course over ground;
 - SOG speed over ground;
 - HDG heading;
 - LOG airspeed.

5.10.2.1 Landing and Taking-Off of Helicopter

To control the helicopter landing in the automated flight mode, use the following procedure:

- 1. Click the toolbar button or choose the menu View\Panels...\Helicopters... to open the Helicopters panel.
- 2. From the Choose helicopter list, select a helicopter which will make the landing.
- 3. Select the Automated Flight control mode.
- 4. Select the operation from the drop-down list: Land from port side (default operation) or Land from astern side.



5. Position the cursor \bigoplus on the ship contour and left-click. The helicopter approaches the ship and hovers in a waiting position. The helicopter status is "Ready To Land".



- 6. Click the Land On button. The helicopter moves from the waiting position to the flight deck and lands.
- 7. Click the Lash down button. The lashings are installed.
- 8. Click the Disengaged button. The rotors are disengaged and stopped.
- 9. Click the Shut Down button. The engines are shut down. The helicopter status is "On Deck".

NOTE: Before the simulator session start, you can land the helicopter on the ship helipad. To do this, open the helicopter context menu, choose the **Landed on...** item, and click on the ship.

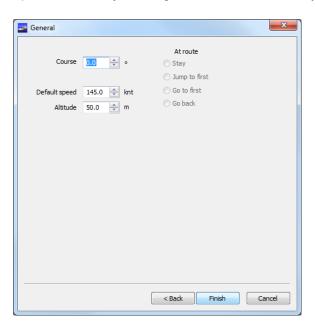
To control the helicopter taking-off in the automated flight mode, use the following procedure:

- 1. On the **Helicopters** panel, check that the helicopter has the "On Deck" status, click the **Start** button. The engines are started.
- 2. Click the **Engage** button. The rotors are engaged.
- 3. Click the Remove Lashings button. The lashings are removed. The helicopter is ready to launch.
- 4. Select the taking-off direction from the Launch drop-down list: Launch to Port side, Launch to Stbd side. The helicopter takes off to port/starboard side.

5.10.3 Operations with Aircraft

To enter the Aircraft object initial parameters, use the following procedure:

1. Open the list of object categories, select the aircraft type in the Aircraft category and click the Next button.



- 2. Set the aircraft initial parameters:
 - o Course:
 - Default speed;
 - o Altitude.
- 3. Click the Finish button.
- 4. Clicn on the chart to fix the object.



The airplane route is plotted, and the waypoint properties and route segments are set in much the same way as in the work with helicopters (see "Helicopter Initial Parameters" on page 403).

5.11 Underway Replenishment Operations

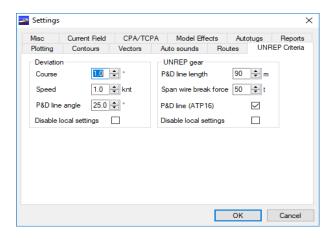
Underway replenishment (UNREP) operations can be performed with the use of two ship model types: delivery ships and receiving ships.

Participating in the exercise can be both own ships and target ships (6-DoF models). If the former is the case, a joint sailing scenario is used for loading an exercise on the bridge.

5.11.1 Setting Underway Replenishment Criteria

To specify the general UNREP criteria settings, use the following procedure:

1. Choose the View\Settings... menu item and select the UNREP Criteria tab.



- 2. In the **Deviation** group box, specify the following parameters:
 - o Course admissible course deviation of the receiving ship (1 degree by default);
 - Speed admissible speed deviation of the receiving ship (1 knot by default);
 - P&D line angle admissible deviation of the P&D line from the normal to the plane parallel to the centreline plane of the receiving ship and passing through the P&D line fastening point on the receiving ship (25 degrees by default).

NOTE: The P&D line (phone and distance line) is a rope for checking a distance between the ships fitted out with telephone cable, and also between daytime and night time signal markers apart from each other at a certain distance.

To cancel the local UNREP criteria settings, select the Disable local settings checkbox.

- 4. In the **UNREP** gear group box, set the following parameters:
 - P&D line length maximum length of the P&D line (90 m by default);
 - Span wire break force maximum break force of the span wire (50 tons by default).

NOTE: Span wire is a carrier line which the fuel feeding hose is passed on.

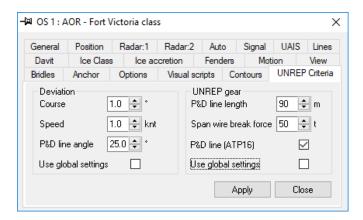
 P&D line (ATP16) - visual presentation of daytime and nighttime markers of distance between the ships fixed on the P&D line (if the parameter is checked, the European marking system is used, if not - then the American system is used).

To cancel the local UNREP criteria settings select the Disable local settings checkbox.

4. Click OK.

For the delivery ships, you can specify the local UNREP criteria settings before the exercise start:

- 1. Open the UNREP Criteria tab of the delivery ship properties panel.
- 2. Clear the Use global settings checkboxes to cancel the general UNREP criteria settings.

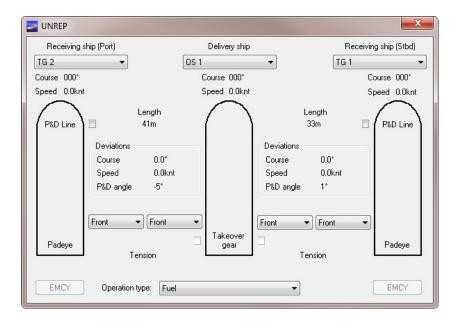


- 3. Set fuel transfer parameters as described above for general UNREP criteria settings.
- 4. Click Apply.

5.11.2 Underway Replenishment Control Panel

The process of fuel transfer underway can be controlled from the UNREP panel.

To open the panel, choose the (View\Panels\UNREP... menu item, or click the stoolbar button.



In the top part of the panel, there are lists for selecting the ships engaged in the fuel transfer underway: Receiving ship (Port), Delivery ship, Receiving ship (Stbd). The list of ships is formed automatically as per the ships set in the exercise.

Two ships can participate simultaneously in the fuel transfer underway: on the starboard and on the portside of the delivery ship.

If the receiving ship is selected from one of the lists, e.g., from the Receiving ship (Port) list, it becomes unavailable on the Receiving ship (Stbd) list.

The **Delivery ship** list is always available to ensure the capability to select another group of ships engaged in the process of transferring fuel underway.

The groups of ships engaged in the fuel transfer underway can be formed both, at the exercise preparation stage and in the course of its fulfillment.

The following parameters will be shown below on the panel:

- current course (Course) and speed (Speed) for each ship;
- current distance between the ships (Length);
- deviations from the set criteria (Deviations): Course, Speed, P&D angle.
- lists for selecting line fastening points. Selection of the fastening point will affect the P&D angle parameter;
- tension of the span wire with the fuel feeding hose attached to it (Tension);
- time in the P&D line rendering/heaving remaining to the end of the operation and the current operation status (rendering/heaving);
- time in the span wire rendering/heaving remaining to the end of the operation and the current operation status (rendering/heaving).

If the value of any parameter is exceeded, or reaches 90% it is highlighted in red.

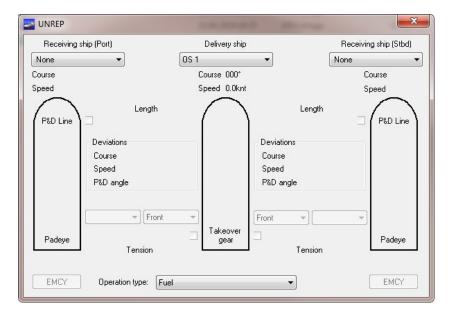
Before the start of the operation, it is possible to choose the operation type from the **Operation type** drop-down list: **Fuil** - fuel transfer, or **Packages** - dry cargo transfer.

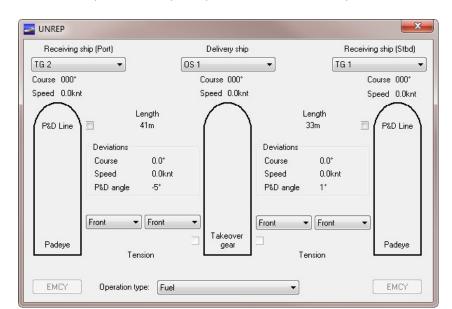
The EMCY button is used for the emergency heaving of all the lines (P&D line and span wire).

5.11.3 P&D Line and Span Wire Rendering/Heaving

To render and heave the P&D line and span wire, use the following procedure:

1. Choose the View\Panels\UNREP... menu item, or click the toolbar button. The screen will display the UNREP panel with a schematic presentation of the delivery ship and two fuel receiving ships.





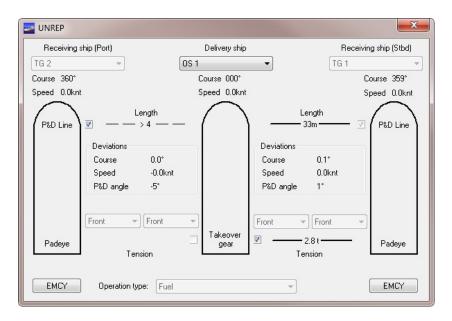
2. Select the ships which will participate in the fuel transfer operations from the drop-down lists.

Deviation parameters will appear next to the schematic presentation of ships.

Red will be used for showing that the deviation set by the parameter is exceeded. The line rendering process cannot be resumed until these parameters assume permitted values.

- 3. Select possible line fastening points for each ship from the list: front, middle, rear.
- 4. Select the P&D line checkbox to pass the P&D line from the fuel receiving ship.

A dashed line connecting the receiving ship with the delivery ship will appear on the panel (in 5 seconds the dashed line is substituted for the solid one, it means that P&D line has been rendered). The distance between the ships in metres will be shown above the line. After the P&D line has been rendered the **EMCY** button is enabled, and the **Takeover gear** field next to the picture of the delivery ship is also activated.



5. Select the Takeover gear checkbox to render the span wire.

NOTES:

- If the distance between the ships is larger than the set value, it is impossible to render the P&D line or span wire.
- If the distance between the ships is smaller than or equal to the set value, the P&D line can be rendered, but the span wire cannot be rendered until the P&D line is put out.
- When the set P&D line angle parameter is exceeded, the P&D line breaks.
- 6. To heave the lines, use the reverse process: first clear the **Takeover gear** checkbox, and after the span wire has been removed, clear the **P&D Line** checkbox.

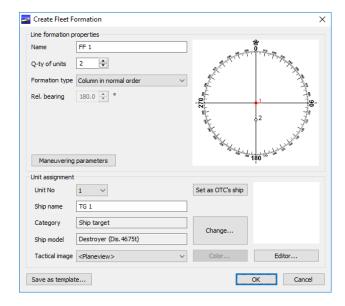
All information about the P&D line and span wire handling operations is displayed on the **Events** panel and recorded to the log file.

5.12 Joint Ship Maneuvering in Formations

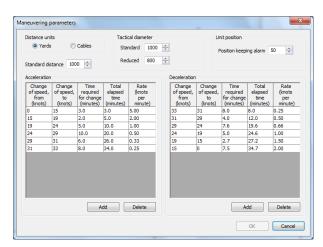
5.12.1 Creating Fleet Formation

To create a fleet formation before the exercise start, use the following procedure:

1. Choose the Edit\Create Fleet Formation... menu item or click the toolbar button. The Create Fleet Formation dialogue box appears.



- 2. In the **Line formation properties** group box, set the following parameters:
 - a. In the Name field, enter the fleet formation name .
 - b. In the Q-ty of units field, set the number of ships in the formation (up to 9 objects can be set all in all).
 - c. Select the type of fleet formation from the **Formation type** drop-down list: Column in normal/reverse order, Line abreast to starboard/port, Line of bearing, Column open order, Diamond.
 - d. If the Line of bearing type formation is selected, set the relative bearing in the Rel. bearing field.
- 3. To set maneuvering parameters:
 - In the Create Fleet Formation window, click the Maneuvering parameters button. The Maneuvering parameters dialogue box appears.



- b. Select units of measurement for the distance between the ships in the formation: Yards or Cables.
- c. In the Standard distance field, set the distance between the ships in the formation.
- d. In the **Tactical diameter** group box, set the tactical diameter of the formation turning circle: **Standard**, **Reduced**.
- e. In the Position keeping alarm field, set accuracy of keeping the ship position.
- f. To change the ship characteristics in the execution of a manoeuvre, use the **Acceleration** and **Deceleration** tables. To add or delete rows in the tables, click the **Add** or **Delete** button respectively.
- g. To apply settings, click the OK button. The Maneuvering parameters window will close down.
- 4. In the Unit assignment group box, set the following parameters:
 - a. In the Unit No field, select the ship number.
 - b. To appoint a ship as that of the Officer in Tactical Command (OTC), click the Set as OTC's ship button.
 - c. In the Ship name field, enter the ship name.
 - d. To change the ship model or category: click the **Change...** button, select the ship category and model and click **OK**.
 - e. If necessary, create a tactical image of the ship on the chart: click the Editor... button; in the Image Editor window, draw the desired tactical image and save it; from the Tactical image list, select the saved image; from the Color list, select the colour of the image.
- 5. Set the properties for other ships in the formation (see step 4).
- 6. If it is necessary to create a fleet formation template, click the Save as template... button.
- 7. To apply all settings, click the OK.
- 8. In the General dialogue box, set the following formation properties:
 - a. In the Speed field, set the formation speed.
 - b. In the Orientation field, set the formation orientation.
 - c. If it is necessary to edit maneuvering parameters of the fleet formation, click the **Maneuvering parameters** button then revert to step № 3.



- d. Click the Finish button.
- 9. Click on the chart.



To place the ship formation template on the chart:

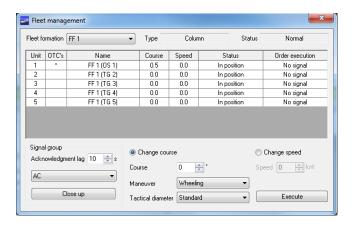
- 1. Choose the Edit\Insert Template... menu item.
- 2. Select ship formation name from the list of templates.
- 3. Enter the name which will be added to the name of each object in the formation.
- 4. Click the OK button.
- 5. Set the ship formation properties in the General window and click the Finish button.
- Click on the chart.

NOTE: You can open the fleet formation properties panel before the exercise start from the Objects panel.

5.12.2 Fleet Formation Management

After starting the exercise, the Instructor can manage the fleet formation from the Fleet management panel.

1. Choose the View\Panels\Fleet management... menu item, or click the toolbar button . The Fleet management panel appears.



- 2. Select fleet formation from the Fleet formation drop-down list. The table contains the following data:
 - Unit ship number in the formation;
 - o OTC's ship which commands are given from;
 - Name ship name;
 - o Course ship course;
 - Speed ship speed;
 - Status ship status;
 - Order execution status of flag signals (No signal, Close up, At the dip).
- 2. To close up the flag signal:
 - a. Select a flag combination from the list.



- b. Set the signal acknowledgement time (delay) in the Acknowledgment lag field.
- c. Click the Close up button.

NOTE: The signal will be closed up in the following stages:

- On the OTC's, the signal will be hoisted to the top, the signal status will be "Close up".
- On the intermediate ships, in a successive order, the acknowledgement signal will be hoisted to ¾ of the height, the signal status will be "At the dip".
- On the farthest from the OTC's ship, the acknowledgement signal will be hoisted to the top, the signal status will be "Close up".
- In the reverse order, on the intermediate ships the signal will be hoisted to the top, the signal status will be "Close up".
- The status of flag signals in the formation will be "Close up".
- 4. To lower the flag signal, click the Execute button.
- 5. To set a formation maneuver:
 - a. Select the Change course option button.
 - b. In the Course field, set the formation course.
 - c. Select a maneuvre type from the Maneuver drop-down list: Wheeling, Turn-together, Search turn.

NOTE: The choice of maneuver depends on the type of formation. Some maneuvres cannot be selected for certain formation types.

- d. Select a tactical diameter of formation from the Tactical diameter drop-down list: Standard, Reduced.
- e. Click the Execute button.
- To change the formation speed:
 - a. Select the Change speed option button.
 - b. In the Speed field, set the ship formation speed.
 - c. Click the Execute button.

5.13 Passing of Locks

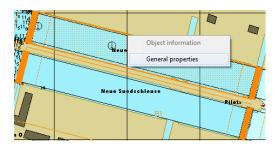
The Instructor can control of locks in an area where there are some locks. The following operations can be performed:

- · control of water level in the chamber;
- · control of gate opening/closing;
- · control of lights.

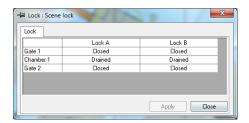
5.13.1 Setting Lock Initial Properties

To set lock properties before the exercise start, use the following procedure:

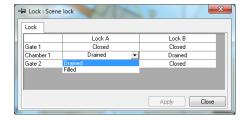
1. Right-click on the lock border on the chart. The object context menu will appear on the screen.



Choose the General properties item. The screen will display a panel with properties of the whole lock system in the area. The Lock page will contain information on the status of lock gates and chambers.



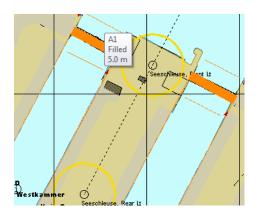
- 3. Set the lock initial properties:
 - Select the lock name from the table: Lock A (B, C...).
 - In the Gate1 (2, 3,...) row, select the gate status: "Closed", "Open".
 - If the gate is closed ("Closed" status), then in the Chamber 1 (2, 3) row select the chamber status: "Drained" draining to a certain level or "Filled" filling to a certain level.



4. Click the Apply and Close buttons.

NOTE: The maximum level of water in the chamber during the filling and draining is determined automatically on the basis of the water level in the area channel and current tide height.

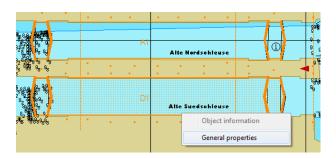
5. As the cursor is position on the lock border, a pop-up prompt with the set lock properties will appear.



5.13.2 Control of Lock

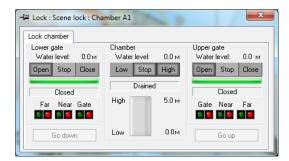
Control of locks after the exercise start is performed from the lock chamber properties panel.

To open the lock chamber properties panel, right-click on the lock border on the chart and choose the **General properties** item.



The **Lock chamber** properties panel contains three groups of controls:

- Lower gate control of the lower gate and lights of the tail bay;
- · Chamber control of the water level in the chamber;
- Upper gate control of the upper gate and lights on the headrace.



The Lower gate and Upper gate groups contain the following controls and indicators:

- Water level indicator of water level in the tail bay (lower gate backwater) and water level in the headrace (upper gate head);
- Buttons for the control of the lock's upper and lower gates:
 - Open to open the gate;
 - Close to close the gate;
 - Stop to stop the process of opening/closing the gate.

NOTE: The lock gates cannot be opened or closed unless the water level is the same on both sides of the gate (the head is equal to backwater). The gate opening/closing time is 100 seconds.

- · Gate opening/closing progress bar;
- Gate status bar: "Open", "Opening", "Closed"; "Closing", "Stopped";
- Buttons for the control of the headrace and tail bay lights:
 - Far to turn on/off the far lights;
 - Near to turn on/off the near lights;
 - Gate to turn on/off the chamber entrance (exit) lights.

NOTE: The green trail bay light is turned on only if the gates are opened.

The Go down, Go up buttons are used in the multi-chamber locks for the successive switching of control from one
lock chamber to another.

The Chamber group box contains the following controls and indicators:

- Water level indicator of water level in the chamber (lower gate backwater);
- Buttons for the control of water level in the chamber:
 - Low to make water level in the chamber equal to the tail bay level;
 - High to make water level in the chamber equal to the headrace level;
 - Stop to stop the chamber draining/filling process.

NOTE: The chamber can be filled if its level is below the upper gate head, and the other way round: it can be drained if the level is above the lower gate backwater. Chamber draining/filling time is 600 seconds.

- Chamber status bar: "Drained", "Draining", "Filled", "Filling", and "Stopped".
- Graphic indicator of the current water level in the chamber.

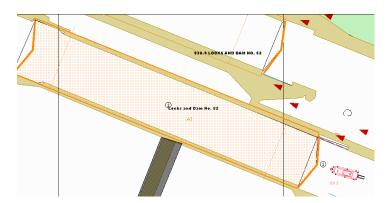
The following hydrological effects are modelled in the course of chamber filling/draining:

- in the chamber filling 0.5 knot stream in the headrace in the direction of the chamber upper gate;
- in the chamber draining current of 0.5 knot in the tail bay directed to the chamber lower gate;
- in the filling and draining irregular waves within the chamber with the wave height of 0.2 m (most significant height).

Example

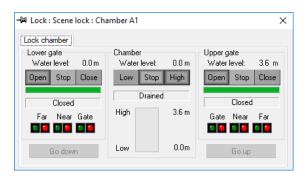
Let us consider the lock control using an example with the following initial conditions:

- the ship is proceeding from the headrace to the tail bay.
- the upper and lower gates are closed (the "Closed" status), the lock chamber is drained (the "Drained" status).

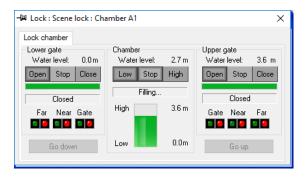


To control locks, use the following procedure:

- 1. Open the exercise where there are locks, and start it on the bridge.
- 2. Right-click on the lock border on the chart. The lock chamber properties panel appears.



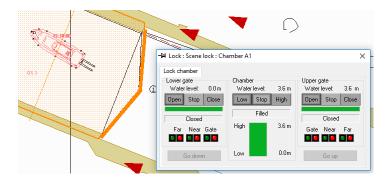
3. For the ship to be able to enter the lock, set the water level in the chamber equal to the level in the headrace. To do this, click the **High** button.



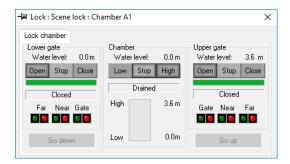
4. After the chamber has been filled, click the Open button to open the upper gate.



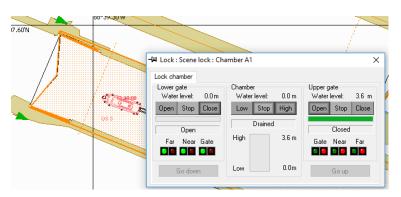
5. After the ship has entered the lock chamber, click the Close button to close the upper gate .



6. To enable the ship exit from the lock, set the water level in the chamber equal to the level in the tail bay. To do this, click the **Low** button.



7. After the chamber has been drained, click the Open button to open the lower gate.



- 8. During the passage through the locks, use the buttons to control the far (Far), near (Near) and entrance (Gate) lights of the headrace and tail bay:
 - green light all-clear signal;
 - red light- restrictive signal.

5.13.3 Monitoring Fulfilment of Exercise with Locks

The following violations of lock passing rules are recorded automatically to enable monitoring and assessment of the exercise fulfilment:

- crossing of the far lights line with the restrictive signal on;
- crossing of the near lights line with the restrictive signal on;
- crossing of the stop sign line in the tail bay or headrace with the restrictive signal of intermediate lights on;
- crossing of the stop sign line in the lock chamber with the restrictive signal of exit lights on;
- · ship touching the gate.

Information on violations is displayed on the Events panel.

5.14 Control of Semaphores

These exercises can be performed in any area where semaphores are available.

To move a semaphore, use the following procedure:

1. Right-click on the semaphore object on the chart and choose the Move item from the context menu.



2. Select the required position on the chart and left-click.

To return the semaphore to the initial position, choose the Return to initial position context menu item.

To set the semaphore properties, use the following procedure:

- Right-click on the semaphore object on the chart and choose the General properties item from the context menu.
- In the General tab, set the semaphore direction (Direction) and coordinates (Latitude and Longitude) then click Apply.

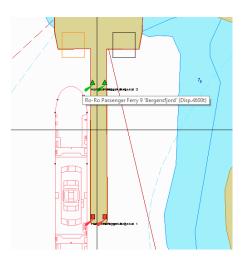


3. In the Traffic Signals tab, select the required signal from the Traffic signal drop-down list, and click Apply.



5.15 Ro-Ro Operations

In ro-ro operations, an own Ro-Ro ship and Scene ramp objects are used.

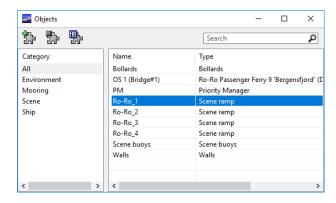


The Instructor controls ramps on the berth and on the ship, opening and closing of the bow and stern gates.

The motor vehicles move along the lines. The number of lines is from 1 to 5 (depends on the ship model and ramp on the berth). The loading/unloading of motor vehicles is controlled with semaphores. The number of motor vehicles which can be loaded/unloaded is unlimited.



To quickly find a Scene ramp object on the chart, you can use the **Objects** panel (menu **View\Panels\Objects...**): double-left click the scene ramp name in the table to show the object at the center of the chart.



To load/unload the motor vehicles, use the following procedure:

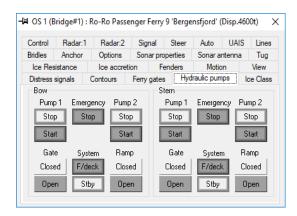
- 1. Before starting the exercise, open the berth ramp properties panel and set the following criteria:
 - Dangerous bump maximum permitted speed of mechanical interaction with walls;
 - o Max. ramp angle permissible angular deviation between the berth ramp and the ship.



- 2. Run the exercise on the bridge.
- 3. To set the berth ramp tilt on the properties panel:
 - a. In the **Angle** field, set the ramp tilt angle (positive value to raise the ramp, negative value to lower the ramp, 0 is set by default).
 - b. Click the Tilt button. If it is necessary to suspend the operation, click the Stop button. The progress bar will show the performance of the operation. The current operation status bar will display: Tilting - ramp tilting operation, Stopped - operation suspended, In position - the ramp is in the set position.

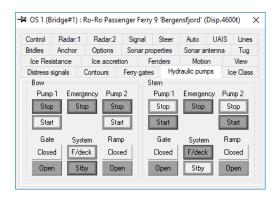
NOTE: The Instructor may not set the berth ramp tilt since the ship-and-ramp docking process will be automatic.

- 4. To start the hydraulic pumps:
 - a. Open the Hydraulic pumps tab of the ship properties panel.



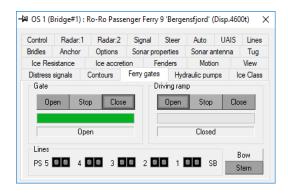
b. Click the F/deck button to start the hydraulic system.

c. Click the Start buttons to start hydraulic pumps 1, 2 on the ship bow and/or stern (in the Standby mode, the Start buttons are disabled).

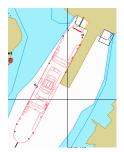


NOTE: The Gate and Ramp indicators show that the gate is closed and the ship ramp is in the vertical position (the Closed is lighted).

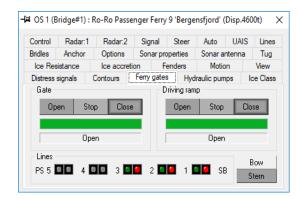
- d. If it is necessary to switch the hydraulic system to the Standby mode, click the **Stby** button. If it is necessary to stop the hydraulic pumps, click the **Stop** buttons. For the emergency stop of the hydraulic system, click the **Emergency Stop** button.
- 5. To open the gate:
 - a. Open the Ferry gates tab of the ship properties panel.
 - b. Select the gate which should be open (bow or stern) by clicking the Bow or Stern button.
 - c. In the Gate group box, click the Open button to open the gate.



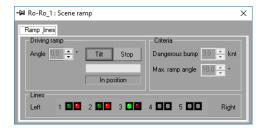
- d. If it is necessary to stop the gate opening process, click the Stop button; to close the gate, click the Close button.
- 6. For the ship docking with the berth ramp:
 - a. Set the ship speed which does not exceed the Dangerous bump criterion value.
 - b. Moor the ship with its bow or stern.



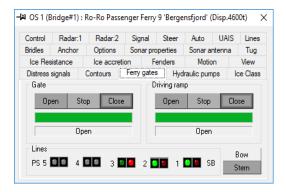
7. On the Ferry gates tab of the ship properties panel, in the Driving ramp group box click the Open button to lower the ship ramp. If it is necessary to stop the operation, click the Stop button. To raise the ship ramp, click the Close button.



- 8. To load the cars to the ship:
 - a. Open the berth ramp properties panel.
 - b. In the Lines group box, select one or several lines for the cars motion and turn on the semaphore green light.



- c. To stop the loading, turn on the semaphore red light.
- 9. To unload the cars from the ship:
 - a. Open the ship properties panel.
 - b. In the **Lines** group box of the **Ferry gates** page, select one or several lines for the cars motion and turn on the semaphore green light.



c. To stop the unloading, turn on the semaphore red light.

NOTE: The motor vehicles loading and unloading may be performed simultaneously on the condition that lines should not intersect.

5.16 ECDIS Training

The following tasks can be started for the ECDIS training:

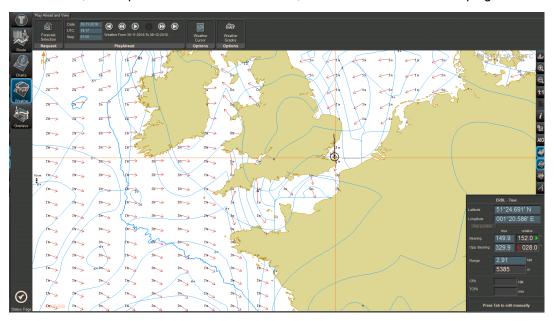
- Chart server task.
- · two Navi-Sailor 4000 tasks on one bridge.
- Navi-Planner 4000 task.

To fulfil an exercise, select the Ficticious Area in the Navi-Trainer instructor application.

NOTE: If AIO (Admiralty Information Overlay) for NS4000 is required, please contact GSS. The Overlay includes all Temporary and Preliminary Notices to Mariners (T&P NMs). The Overlay is displayed as a single layer on top of the basic ENC and it will be available via Navi-Sailor ECDIS. For details, see Navi-Planner 4000 and Navi-Sailor 4000 manuals."

5.16.1 Ordering Charts, Chart Updates and Weather Forecasts

To order charts, chart updates and weather forecasts, use the "Navi-Planner 4000" program.



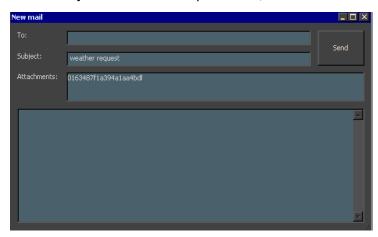
To make an order, the Trainee needs to generate a respective request.

NOTE: The steps for generating requests for ordering charts, chart updates and weather forecasts are described in *Navi-Planner 4000 User Manual*.

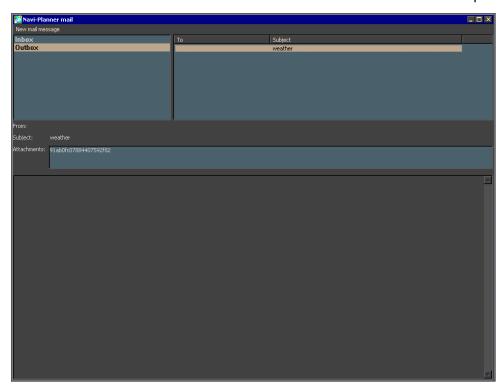
The generated request is sent automatically or by using the email emulator.

To send a request by using the email emulator:

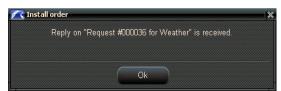
- 1. Open the Navi-Planner mail window.
- 2. Click on the **New mail message** line. The **New mail** window will appear. The address is entered in the **To** field by default.
- 3. In the Subject field enter the request name; in the Attachment field, right-click to add the generated request.



- 4. Click the Send button. The New mail window will close.
- 5. In the Navi-Planner mail window select the Outbox line. The name of the sent request will appear.

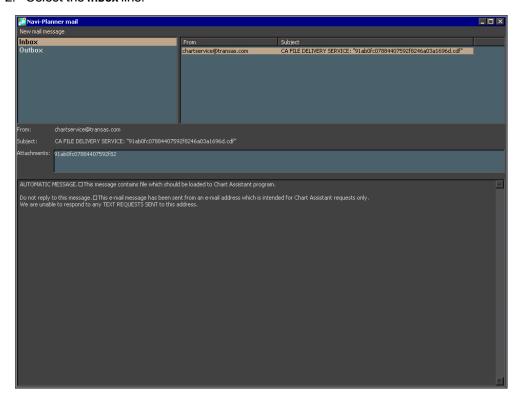


After the Instructor processes the request in the "Navi-Planner", the following message appears.

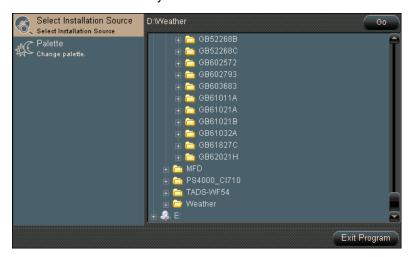


To receive a response to the request:

- 1. Open the Navi-Planner mail window.
- 2. Select the Inbox line.



- 3. Click on the Attachment field. The Save Attachment message will appear.
- 4. Select a folder to save the ordered charts or the weather forecast and click Save.
- 5. Click on the button.
- 6. Select the folder where you saved the ordered charts or weather forecast.

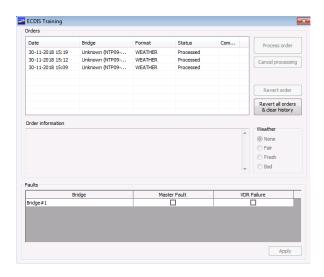


- 7. Click the Go button.
- 8. Click the Install button to install the selected charts or weather forecasts.

5.16.2 Order Processing

Requests are received at the Instructor workplace and displayed on the Ecdis Training panel.

To open the panel, choose the View\Panels\ECDIS Training... menu item.



The Ecdis Training panel contains:

- The Orders group box is used for the display of current information for all orders:
 - o Date request intake date.
 - o Bridge bridge from which an automatic request or an Email has been received.
 - Format order format: WEATHER weather forecast order, SENC CORR chart update order, SENC CAT chart catalogue order.
 - o Status order status: Processed order is processed, Unprocessed order is unprocessed.
 - o Comment comment on the order.
- The Order information group box is used for the display of information on the order selected in the Orders group box.
- The Faults group box is used for setting faults:
 - Bridge bridge number;
 - Master Fault fault of Master Navi-Sailor;
 - VDR Failure voyage date recorder (VDR) failure.
- · Buttons:
 - Process order to start the order processing;
 - Cancel processing to cancel the order processing;
 - Revert order to delete the order selected in the Orders group box.
 - Revert all orders & clear history deletion of all orders selected in the Orders group box.
 - o The Weather group box contains optional buttons for selecting the weather type. None, Fair, Fresh, Bad.

To process an order, use the following procedure:

- 1. In the Orders group box, select an order with the "Unprocessed" status.
- 2. If the order is in the WEATHER format, select the necessary weather type in the Weather group box.
 - Bad stormy weather;
 - Fresh fresh weather;
 - Fair fair weather.

3. Click the Process button. The order status will change to "Processed".

To set faults, use the following procedure:

- To set a Master Navi-Sailor fault, select the Master Fault checkbox on the Ecdis Training panel and click Apply.
 The Master Navi-Sailor will stop working. The Slave Navi-Sailor will become the Master. To turn off the Navi-Sailor fault, clear the Master Fault checkbox and click Apply. The turned-off Navi-Sailor will start working and become Slave.
- 2. To set a voyage date recorder failure, select the VDR Failure checkbox, and click Apply. The VDR Alarm will light up in the Master and Slave Navi-Sailor. To turn off the alarm, clear the VDR Failure checkbox and click Apply.

5.17 FOS Data Import

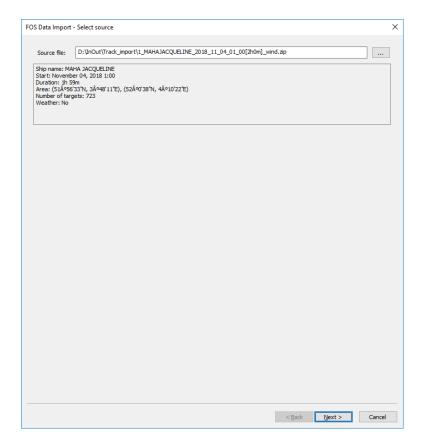
FOS data import functionality provides a capability to automatically create an NTPRO exercise based on the data collected from the FOS subscribed vessel.

NOTES:

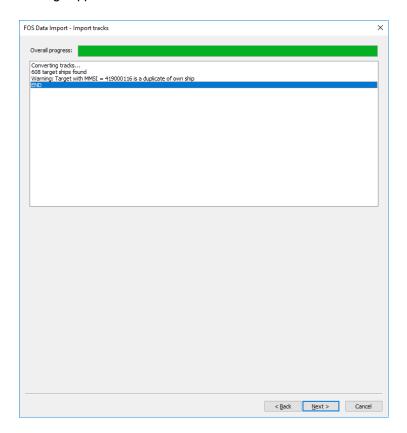
- The exercise content and quality depends on the originally collected data quality and content, and on the set of models and areas available in the simulator.
- FOS collected data file should be downloaded by and obtained from the pertinent FOS subscriber.

To create an exercise using the data collected from a FOS subscribed vessel, use the following procedure:

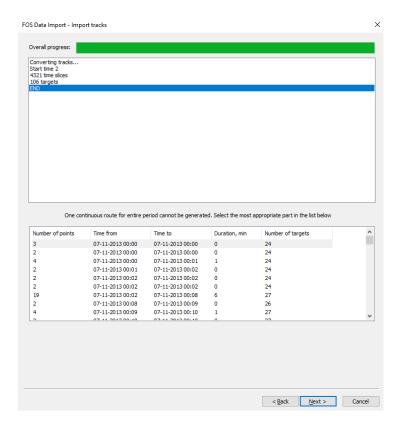
- Run the FOS Data Import program: from the Windows start menu, choose Navi-Trainer 6\FOS Data Import.
- 2. In the Source file field of the Select source window, specify a path to the folder where the track files (FOS data files*.zip, cvt or JSON raw data files*.json) are stored.



3. Click the **Next** button to start file converting. When the converting has been completed, the corresponding message appears in the window.

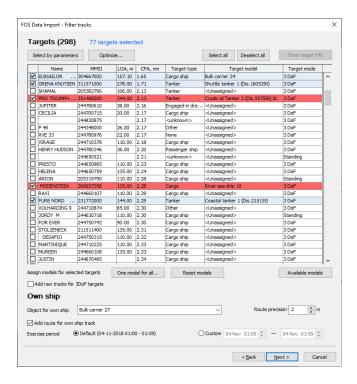


If the track has been split into several parts, the table with information on each part of the track appears at the bottom of the window. Click the table row to select the desired part of the track.



NOTE: The track is split if it is not possible to construct a continuous route without exceeding the vessel (own ship) maximal speed. This may happen, for instance, e.g. due to a significant change of the vessel position caused by the GPS source switch or change of its precision (settings which cannot be filtered automatically).

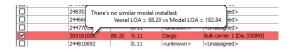
4. Click the Next button. The window with a list of targets appears.



NOTES: The checkbox next to the target name indicates that the system has recognized this target and identified it as similar to the model in the NTPRO simulator.

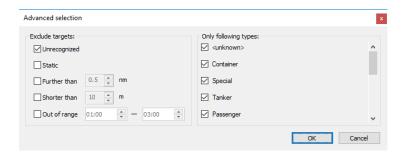
If the selected target is marked in red in the table, it means that there are no similar models available in the simulator. This may occur in the following cases:

- The target's length overall (LOA) and the LOA of the model differ by more than 10 %.
- The target's speed and model's speed differ by more than 10 %. Or the maximal model's speed is less than the target speed on a track.
- The target type does not match the model type.

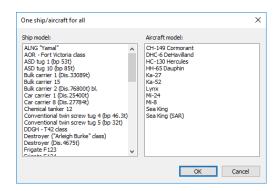


- To select a target in the table, select the checkbox next to the target name or click the Select all button to select all targets.
- 6. To deselect the target, clear the corresponding checkbox or click the Deselect all button to deselect all targets.

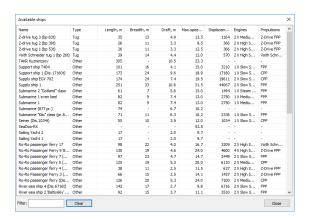
- To use the filter for target selecting, click the Select by parameters button. The Advanced selection window appears.
 - a. In the Exclude targets group box, select characteristics of the targets you want to exclude from the list of targets to be used in the exercise:
 - Unrecognized unrecognized target;
 - Static static target;
 - Further than target is at a distance further than the specified distance, nm;
 - Shorter than target is at a distance closer than the specified distance, m;
 - Out of range target is out of the specified time range.
 - b. In the Only following types group box, select the target types you want to include in the list of targets: Unknown, Container, Special, Tanker, Passenger, Tug, Car Carrier, Bulk Carrier, Fishery, Cargo, Other, Research, Gas carrier (the list of the types depends on the data file content).



- c. Click the OK button.
- 8. To automatically reduce the number of selected targets in the list (up to 100 for the 3 DoF targets, and up to 10 for the 6 DoF targets), click the **Optimize...** button. The system tries to keep only the most important targets.
- 9. To show the selected target information, select the target in the table and click the **Show target info** button.
- 10. To edit target parameters in the table:
 - a. In the Target model column, open the drop-down list and select the desired target model.
 - b. In the **Target mode** column, open the drop-down list and select the desired target mode: 3 DoF (the mode is selected by default) or 6 DoF mode.
- 11. Models can be assigned to selected targets in one of the following ways:
 - To assign one model to all the selected targets: click the One model for all... button; select a ship model or an aircraft model, then click OK.



 To roll back to the automatically chosen (the best-matching) models for the selected targets, click the Reset models button. 12. To show the list of ship models installed in the NTPRO simulator, click the **Available models** button. The **Available ships** window appears.



- a. To find a model in a list, enter the model name in the Filter field. To clear the Filter field, click the Clear button.
- b. Click the Close button to close the window.
- 13. To use historical tracks instead of routes, select the Add raw tracks for 3 DoF targets checkbox.

NOTE: Targets with such tracks will travel strictly along those tracks. They cannot be moved, nor can they be assigned to other routes.

14. From the Object for own ship drop-down list, select the object for your own ship.



NOTES: The object that is the most similar to the own ship model in the NTPRO simulator is selected automatically.

If the selected object is marked in red, it means that there are no similar own ship models available in the simulator. This may occur in the following cases:

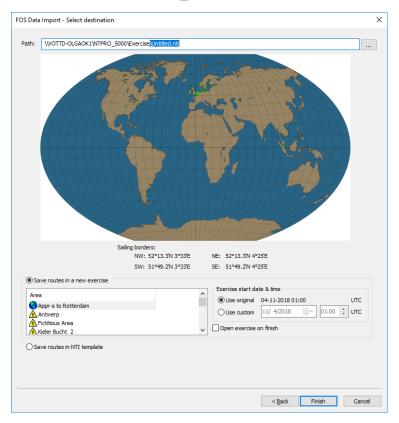
- The object's LOA and the LOA of the own ship model differ by more than 10 %.
- The object's speed and the speed of the own ship model differ by more than 10 %. Or the maximum own ship model speed is less than object speed on a track.
- The object type does not match the type of the own ship model.
- 15. To add a route for the own ship track, select the Add route for own ship track checkbox.
- 16. In the Route precision field, set the maximum deviation of the route from the track in the NTPRO exercise.
- 17. Select the exercise period: **Default** (track time duration) or **Custom** (the exercise time duration).



18. Click the Next button.

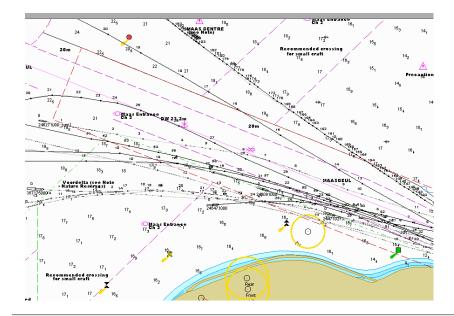
NOTE: The **Performance Warning** dialog box may appear if the number of the selected targets is too large. In this case, you can return to the target selection window (the **Back to adjustment** button), or automatically reduce the number of targets (the **Optimize automatically** button), or continue the action without changing (the **Proceed anyway** button).

19. In the **Path** field of the **Select destination** window, specify the path to the folder where the exercise will be stored. The default folder is *NTPRO_6\Exercise*. The default exercise file name is <code>Untitled.nti</code>.



In the centre of the window, the global map shows the following:

- Yellow round marker indicates the area with the selected tracks. The coordinates of this area are displayed below the map.
- Green round markers indicate the installed training areas.
- 20. To save routes in a new exercise:
 - a. Select the Save routes in a new exercise option button.
 - b. From the **Area** list, select the exercise area. The most relevant area is shown at the top of the list. The areas are ordered according to their closeness to the track. The sign indicates that at least a part of the own ship track is inside the area borders. The sign indicates that the own ship moves outside the area. If there is no relevant area, the "Open Sea" area is selected by default.
 - c. Set the exercise start date and time: original or custom.
 - d. Select the **Open exercise on finish** checkbox, if it is necessary to run the Navi-Trainer Instructor application and to automatically open the exercise with FOS data after completing the FOS data import procedure.
- Or save the routes in the Navi-Trainer Instructor application templates. To do this, select the Save routes in NTI templates option button.
- 22. Click the Finish button. The exercise with FOS data is created.
- 23. Run the Navi-Trainer Instructor application and open the exercise with FOS data.

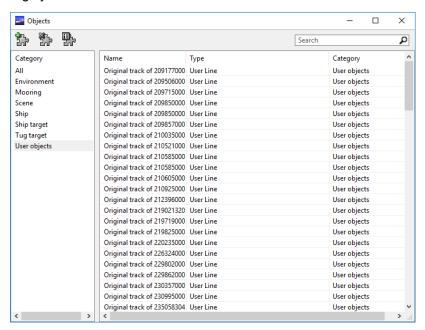


NOTE: The Navi-Trainer Instructor application runs and the exercise with FOS data opens automatically if the **Open exercise on finish** checkbox has been selected.

The imported tracks are presented in the exercise as **User line** objects and displayed with grey lines on the chart providing information about actual motion of the object.



To show the list of all tracks, open the **Objects** panel (menu **View\Panels\Objects...**) and select the **User objects** category.



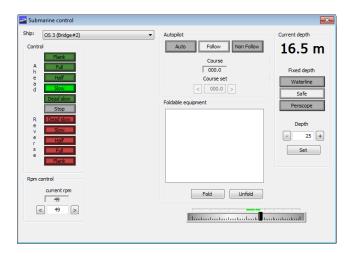
To hide the tracks on the chart, choose the **Chart\Overlays\User Info** menu item, clear the **User Line** checkbox and click **OK**.

5.18 Operations with Submarine

The simulator provides the submarine control feature. To do this, use the Submarine Control panel.

To open the panel, choose the View\Panels\Submarine Control... menu item or click the to





The panel contains

- Ship drop-down list for selecting a ship.
- The Control group box is used for the control of submarine motion and contains the speed control buttons, corresponding to discrete positions of the engine telegraph:



- The RPM control group box contains two buttons with arrows to set the propeller shaft rpm and a digital indicator
 of the current RPM value.
- The Autopilot group box which is used for the control of the submarine autopilot, contains the following controls
 and indicators:
 - The Auto button to switch the autopilot on. When the button is pressed, set the course in the Course set field.
 The Course digital indicator shows the current course value.



- The Follow button to switch on the tracking mode of the submarine motion which provides the submarine steering by means of steering controls;
- The Non Follow button to switch on the Non-Follow Up mode.

- The Current depth group box is used for the control of submarine immersion depth, contains the following controls
 and indicators:
 - Digital indicator of the submarine current immersion depth;
 - Waterline depth button is used to set the submarine imersion at the waterline depth;
 - Safe button is used to set the immersion of the submarine at a depth which is safe enough to avoid a collision with surface ships.
 - Periscope button is used to set the submarine immersion at a periscope depth;
 - Depth field is used for setting the immersion depth from 0 to 100 m. To set the depth, use the ;"+" and "-" buttons and click the Set button.

NOTE: The submarine immersion depth is measured from the keel to the sea surface.

- The **Foldable equipment** group box is intended for the control of the submarine foldable equipment (if the submarine model has foldable equipment) and contains:
 - The list of submarine foldable equipment with status of each device ("Fold/"Unfold").
 - The Fold and Unfold buttons to fold and unfold the selected device.
- Rudder control unit.



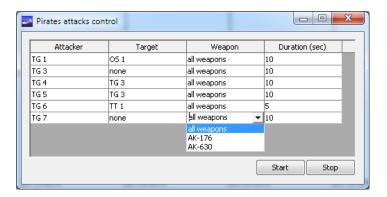
5.19 Pirate Attacks

To simulate the pirate attacks, use the following procedure:

- 1. Open the exercise.
- 2. Add pirate ships to the exercise (the pirate ships are the target ship objects with shooting functionality).
- 3. Start the exercise.
- 4. Choose the View\Panels\Pirates... menu item, or click the toolbar button. The Pirates attacks control panel appears. The panel contains the list of the all pirate ships (the Attacker column).
- 5. For each pirate ship select a target from the **Target** drop-down list. The target can be any own ship/tug, target ship/tug.

NOTE: There will be no shooting if the "none" target is selected.

6. From the Weapon drop-down list, select the type of weapons available on the pirate ship.



NOTE: The pirate ship will shoot from all weapons if the "all weapons" option is selected.

- 7. Set the shooting duration (sec) in the **Duration** column.
- 8. To start the shooting, click the Start button.
- 9. To stop the shooting, click the **Stop** button.

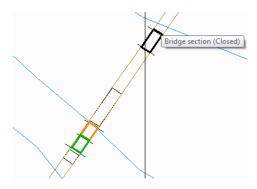
5.20 Operations with Moveable Bridges

The following types of moveable bridges are simulated:

- · vertical-lift bridge;
- rolling bascule bridge (single-leaf bascule);
- bascule bridge (double-leaf bascule);
- swing bridge;
- double swing bridge.

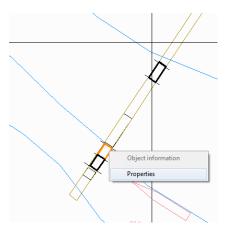
The bridges can consist of one or more moveable sections that can be controlled. Each section is controlled separately. Bridge sections are marked on the chart with the following colours:

- bold black section is fully closed or partially open (by default, all sections of the bridge are fully closed);
- green section is fully open;
- orange section is selected on the chart.

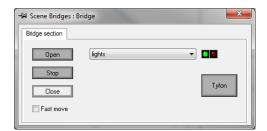


To control the bridges, use the following procedure:

- 1. Create a new exercise in the NTI application. Select the area with moveable bridges.
- 2. Open the context menu of the moveable bridge section and select the **Properties** item.



The properties panel of moveable bridge section appears:



3. Click the **Open** button, to open a bridge section. The **Open** button blinks while the bridge section is fully open. The colour of the bridge section on the chart changes to green.



4. Click the **Stop** button, to stop the bridge section in the current position.



5. Click the **Close** button, to close the bridge section. The **Close** button blinks while the bridge section is fully closed. The colour of the bridge section on the chart changes to bold black.

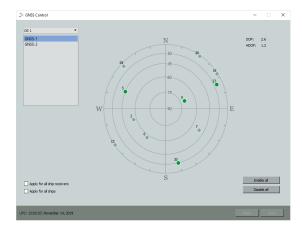


- 6. To instantly open or close the bridge section, select the **Fast move** checkbox (it's available only after the exercise start) and then click the **Open/Close** button.
- 7. To control the bridge section's traffic lights (before and after the exercise start):
 - a. From the drop-down list, select the name of the light or group of lights.
 - b. Click the respective button to switch the green or red traffic light.
- 8. To give the tyfon signal, hold the Tyfon button (the button is available after the exercise start).

5.21 GNSS Control

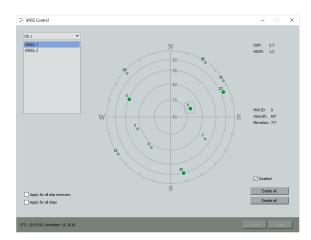
To monitor and control the GNSS satellites, use the following procedure:

1. After the exercise start on the bridge, open the GNSS Control panel: click the View\Panels\GNSS Control... menu item. The panel contains the satellite monitor that shows the position of GNSS satellites in this area. The GNSS system automatically determines which satellites are used to fix a ship position. The DOP and HDOP values appear in the upper-right corner of the panel. The UTC time appears in the lower left corner of the panel.



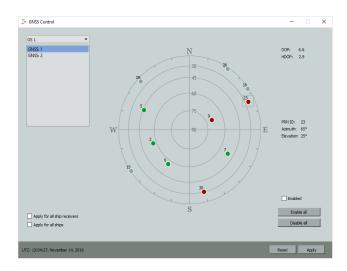
NOTES:

- Green marks show the satellites that are used in fixing the ship position.
- Grey marks show the satellites that are not used in fixing the ship position.
- 2. Select an own ship from the drop-down list.
- 3. Select the GNSS receiver (1, 2, 3...) in the list box. The number of GNSS receivers depends on the ship model.
- 4. To view the satellite data, click to the satellite mark on the monitor. Information about the satellite (PRN ID, azimuth, and elevation) appears in the lower right corner of the panel.



- 5. To disable any of the satellites:
 - Double click the satellite's mark on the monitor.
 - o Or click the satellite mark and clear the Enabled checkbox.

The satellite mark colour changes to red . The DOP and HDOP values are recalculated. The system automatically selects other available satellites to fix the ship position.



To disable all satellites, click the Disable all button.

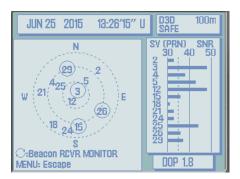
NOTE: Four GNSS satellites are required for fixing a position in three dimensions and time.

- 6. To enable any of the satellites:
 - a. Click the satellite mark on the monitor.
 - b. Select the **Enabled** checkbox. The satellite mark colour will return to the previous one. The **DOP** and **HDOP** values will be recalculated again.

NOTE: All the satellites are enabled by default.

To enable all satellites, click the **Enable all** button.

7. To apply and save the modified settings, click the **Apply** button. The GPS satellite monitor on the Conning Display shows the updated satellite data:



- 8. To return to the last saved settings, click the Reset button.
- 9. To apply the settings to all ship GNSS receivers, select the **Apply for all ship receivers** checkbox and click the **Apply** button.
- 10. To apply the settings to all own ships, select the Apply for all ships checkbox and click the Apply button.

5.22 Control of Motion Platforms

The NTPRO 6 navigation simulator provides control of third-party motion platforms via one of the following bridge tasks:

- Universal Hardware Interface (UHI) task;
- · Motion Platform Interface (MPI) task.

The sections below describe these two cases in details.

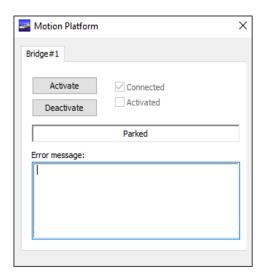
5.22.1 Control of Motion Platforms via UHI Task

The UHI task is currently used for controlling such motion platforms as SCT, QS-V20, Moog, and E2M.

NOTE: The Universal Hardware Interface task must be included in the NTPRO configuration by the Configuration editor and set up as required (see *Navi-Trainer 6 Professional. Technical Description and Installation Manual. Version 6.0*).

To control the motion platform from the instructor workplace via the UHI task, use the following procedure:

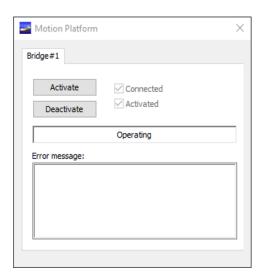
1. To open the Motion Platform control panel, choose the View\Panels\Motion Platform... menu item, or click the toolbar button. The screen will display the Motion Platform window.



2. To start the platform operation, click the Activate button.

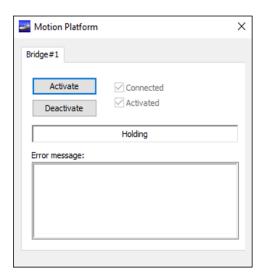
The unparking process is started, during which the platform will smoothly increase the pitch and roll motion.

On completion of activation, the state indicator will display "Operating".

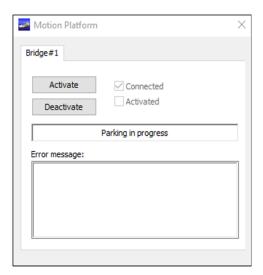


NOTE: Please note that the SCT platform can be activated not only before but also after the exercise start, if the UHI task is set up accordingly (see *Navi-Trainer 6 Professional. Technical Description and Installation Manual. Version 6.0*).

3. If the Instructor pauses the exercise, the UHI task will automatically initialize the "Holding" mode, during which the platform will smoothly move its legs to the neutral position.



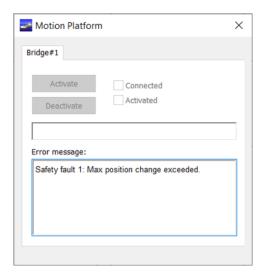
- 4. To resume the platform operation, release the **Pause** button, you will see that the platform smoothly returns to the "Operating" mode.
- 5. To stop the platform operation, click the **Deactivate** button.
 - Immediately after the **Deactivate** button is pressed, the UHI application will initialize the "Parking" mode, during which the platform will smoothly decrease the pitch and roll motion. The state indicator will display "Parking in progress".



On completion of deactivation, the platform state indicator will display "Parked" (see step 1).

NOTE: The "Unparking" mode is initialised automatically after the Instructor activates the **Start** button, and also after the ship has been dragged and dropped in the course of the exercise. The "Parking" mode is initialised automatically after the Instructor activates the **Stop** button, and also in the case of a collision, grounding or model overloading.

If an error occurs during the simulator session, it is shown in the Error message field like that:



To clear the platform error, either stop the exercise or restart the UHI task.

5.22.2 Control of Motion Platforms via MPI Task

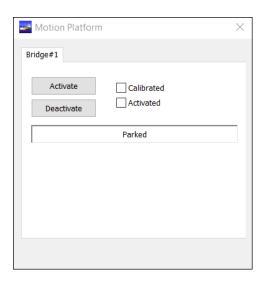
There are certain motion platforms of the NTPRO 6 navigation simulator, which are controlled via the special Motion Platform Interface (MPI) application.

NOTE: Motion Platform Interface task must be included in the NTPRO configuration by the Configuration editor and placed on the computer where the UHI module is connected.

Simple platform control can be executed from the instructor workplace.

To control the motion platform via the MPI task, use the following procedure:

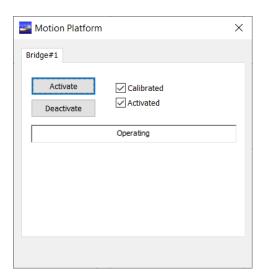
1. To open the Motion Platform control panel, choose the View\Panels\Motion Platform... menu item, or click the toolbar button. The screen will display the Motion Platform window.



2. To start the platform operation, click the **Activate** button.

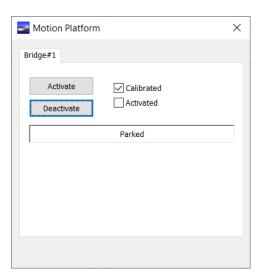
Immediately after the **Activate** button is pressed, the MPI application will initialize the "Unparking" mode, during which the platform will smoothly increase the pitch and roll motion. If the platform is calibrated, the **Calibrated** checkbox is checked.

During platform activation, the **Activated** checkbox is checked, and the state indicator will display "Unparking in progress". On completion of activation, the state indicator will display "Operating".



3. To stop the platform operation, click the **Deactivate** button.

Immediately after the **Deactivate** button is pressed, the MPI application will initialize the "Parking" mode, during which the platform will smoothly decrease the pitch and roll motion. The state indicator will display "Parking in progress". On completion of deactivation, the platform state indicator will display "Parked".



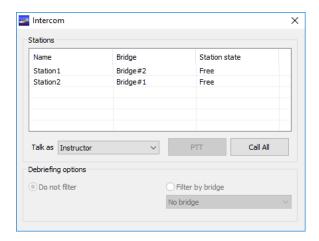
NOTE: The "Unparking" mode is initialised automatically after the Instructor activates the **Start** button or releases the **Pause** button, and also after the ship has been dragged and dropped in the course of the exercise. The "Parking" mode is initialised automatically after the Instructor activates the **Stop** and **Pause** button, and also in the case of a collision, grounding or model overloading.

6. Ship Communication Aids

6.1	Intercom		
	6.1.1	Telephone Communication with Subscribers	456
	6.1.2	Playback of Telephone Communications	459
6.2 VHF Communication			460
	6.2.1	Radio Traffic Monitoring	460
	6.2.2	Keeping of Listening Watch	461
	6.2.3	Playback of Radiotelephone Communications	463
	6.2.4	VHF Background Noise	463
	6.2.5	VHF Notifications	465
	6.2.6	VHF Instructor Panel	466

6.1 Intercom

The shipboard telephone communication is controlled by the Instructor on the **Intercom** panel after the simulator session start.



The Intercom panel contains two group boxes:

- The Stations group box is used for telephone communication with subscribers (see "Telephone Communication with Subscribers" on page 456).
- The **Debriefing options** group box is used for monitoring telephone communication during the log file playback (see "Playback of Telephone Communications" on page 459).

6.1.1 Telephone Communication with Subscribers

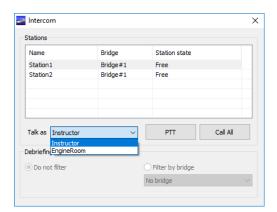
In the course of an exercise, the Instructor can establish telephone communication with any subscriber on the bridge, and receive a call from the subscriber.

NOTE: All telephone communication is logged on the Events panel.

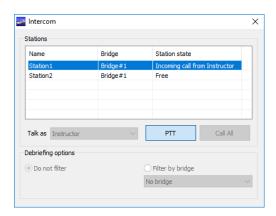
Calling to Bridge Intercom Station

To call to the bridge Intercom station, use the following procedure:

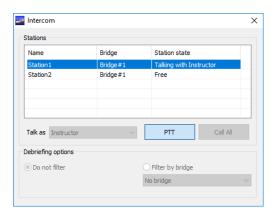
- 1. Choose the View\Panels\Intercom... menu item, or click the toolbar button. The Intercom panel appears.
- From the Talk as drop-down list, select the name of the Instructor calling station.



- In the Stations group box, select a bridge Intercom station where the Instructor will call. The station should have the "Free" state.
- 4. Click the PTT button. A ring tone will be produced. The state of the responding station changes to "Incoming call from...".



After the subscriber answers the call, the state of the responding station changes to "Talking with....".



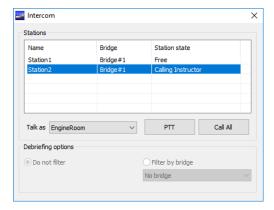
5. To stop communication, click the PTT button again.

NOTE: No telephone connection can be established with a subscriber whose telephone line is busy.

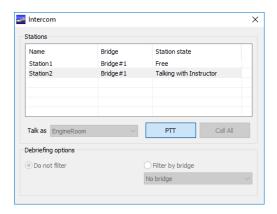
Answering Subscriber Call

To answer a call, use the following procedure:

1. In the Stations group box, select the calling station name. The calling station state is "Calling...".

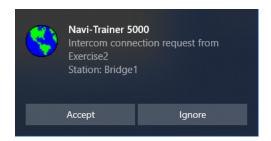


2. Click the PTT button. The state of the calling station changes to "Talking with...".



3. To stop communication, click the PTT button again.

If the Instructor conducts multiple sessions and receives a call from the intercom station in an exercise that is currently not in focus, a notification appears in the lower right corner of the screen.



The notification indicates from which exercise the intercom connection request is sent and which station is calling.

NOTE: Notifications must be enabled in Windows settings.

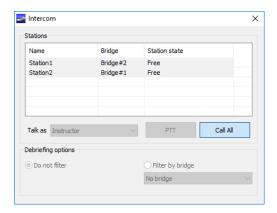
Click **Accept** to view the exercise from which the intercom connection request is sent. The exercise window with actual **Intercom** panel will become active on the screen.

Click Ignore to ignore and close the notification. Or do nothing, and the notification will disappear after a while.

Transmitting Message to All Subscribers

To transmit a message to all subscribers on all bridges, use the following procedure:

1. Click the Call All button and start talking.



2. Click the Call All button again to stop the message transmitting.

6.1.2 Playback of Telephone Communications

In the course of an exercise, all the telephone exchange is recorded. During the log file playback, the Instructor selects a playback mode (the feature is not available in the current version of NTPRO).

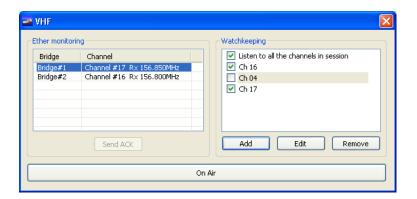
To adjust the telephone communication debriefing options, use the following procedure:

- 1. Choose the View\Panels\Intercom... menu item, or click the toolbar button. The Intercom panel appears.
- 2. In the Debriefing options group box, select one of the following options:
 - o Do not filter to play back all the subscriber traffic (the option is set by default).
 - Filter by bridge to play back the subscriber traffic on a certain bridge. The bridge is selected from the dropdown list.

6.2 VHF Communication

The simulator allows imitation of VHF communications both, between the ships and between the ships and coastal stations. On behalf of the coastal station, the radio telephone traffic is maintained by the Instructor.

The VHF communication is controlled and monitored by the instructor after the simulator session start on the VHF panel (the View\Panels\VHF... menu item, or the toolbar button).

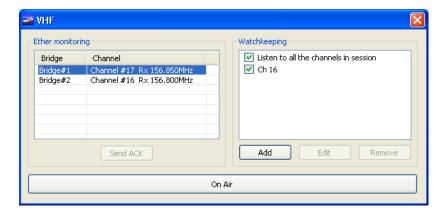


The VHF panel consists of the Ether monitoring group box and the Watchkeeping group box.

The On Air button is used for transferring voice messages by the Instructor on the selected channel.

6.2.1 Radio Traffic Monitoring

The Instructor monitors the Trainee radio telephone traffic with the aid of the **Ether monitoring** group box on the **VHF** panel:



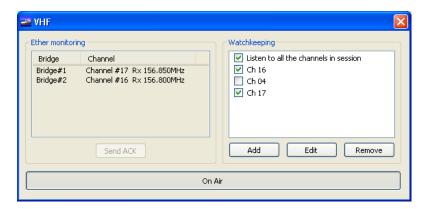
The **Ether monitoring** group box contains:

- Bridge list of bridges on which the VHF communication panel is run;
- Channel the channel number with the current frequency of the transceiver which operates in the receive-transmit mode:
- Send ACK button for sending the distress signal acknowledgement on behalf of the coastal station.

When a Trainee at his/her workplace turns on the VHF station, the **Channel** line displays the current channel number. If the trainee VHF station is in the transmit mode, the **Tx** index will appear after the channel number, and **Rx** in the reception mode. If the trainee VHF station operates in the US channels mode, the **US** index will appear in the **Channel** line after the channel number. If a trainee sends a distress signal, the **Channel** line will contain the words "In distress".

For the radio telephone traffic monitoring, use the following procedure:

- 1. Choose the View\Panels\VHF... menu item, or click the toolbar button.
- To listen to the radio telephone traffic of a certain bridge, select the line with the required bridge name in the Ether monitoring window.
- To interfere in the radio telephone exchange of the trainee workstation, click the On Air button. The radio telephone exchange on the instructor workplace is performed on the channel of the selected trainee workstation.



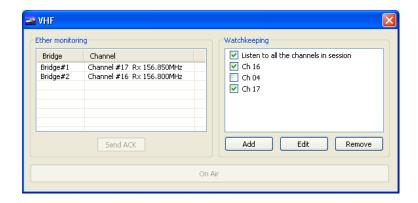
4. To end the radio telephone exchange with the bridge, click the On Air button again.

To acknowledge the receipt of the distress signal on behalf of the coastal station:

- 1. Select the line with the name of the bridge in distress.
- 2. Click the Send ACK button.

6.2.2 Keeping of Listening Watch

For keeping the listening watch, the Instructor uses the Watchkeeping group box on the VHF panel.

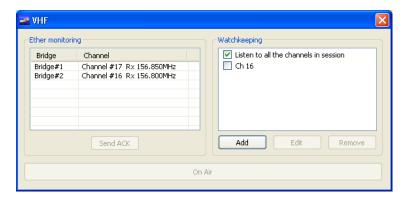


The Watchkeeping group box contains:

- List of channels;
- The Add, Edit and Remove buttons for editing the list of channels.

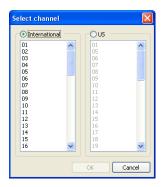
NOTE: The Ch 16 and Listen to all the channels in session lines cannot be deleted or edited.

To keep the listening watch, open the VHF panel (the View\Panels\VHF... menu). By default, the Listen to all the channels in session line is checked, which enables listening to all the channels regardless of the radio waves propagation.

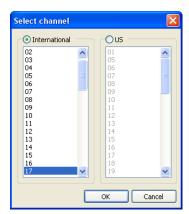


To add a new channel to the list of listening channels, use the following procedure:

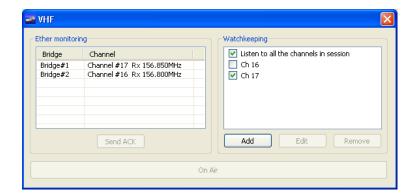
1. Click the Add button. The Select channel window will appear:



- 2. Select the channel mode:
 - International international channels;
 - o US american channels.
- 3. Select the required channel number.



4. Click **OK**. In the **Watchkeeping** group box, the list of channels shows a line with the number of the selected channel; the watchkeeping checkbox is checked opposite the line.



5. Clear the checkbox next to Listen to all the channels in session line.

To enable listening on several channels, repeat the channel addition procedure the necessary number of times. The clearing of the checkbox stops the listening watch on the channel.

To delete a channel from the list, select a line with this channel name and click the **Remove** button. The line with the channel name will disappear from the list of monitored channels. The listening watch will be stopped on this channel.

To edit a list of monitored channels, use the following procedure:

- 1. Select a line with the name of the channel to be edited.
- 2. Click the Edit button.
- 3. Select the mode and name of the necessary new channel.
- 4. Click OK.

All the trainee actions (channel number selection, engaging in the radio traffic, sending of distress signals, sending of distress signal receipt acknowledgement) are logged on the **Events** panel.

6.2.3 Playback of Radiotelephone Communications

In the course of an exercise, all the radio telephone traffic is recorded. During the playback, the instructor selects the playback mode (the feature is not available in the current version of NTPRO):

- simultaneous playback of all the traffic on all the channels;
- selective playback by bridges and channels.

6.2.4 VHF Background Noise

To select sound files with samples of VHF background noise:

- Before the exercise start, choose the View\Panels\VHF background noise... menu item. The VHF background noise panel appears.
- 2. Click the button to add a *wav-file. The audio track name appears in the list cathChicken

NOTE: A set of *wav files is stored in the folder NTPRO 6\Sound\VhfBackground.

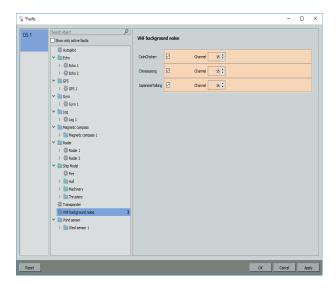
- 3. Click the button to pre-play an audio track on the Instructor workplace. Click the button to stop playing an audio track.
- 4. Click the button to repeat playback of an audio track.

- 5. Move the slider to set volume of the audio track playback.
- 6. Click the button to select another sound file.
- 7. Click the button to remove a sound file. Click the Remove All button to remove all sound files.
- 8. Click **Apply** to apply the settings and continue editing or click the **OK** button to apply the settings and close the window.



To activate the VHF background noise:

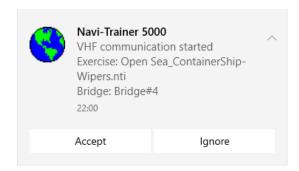
- 1. Choose the View\Panels\Faults ... menu item. The Faults panel appears.
- 2. In the left column of the panel, select an own ship.
- 3. In the middle column of the panel, select the VHF background noise fault category.
- 4. In the right column of the panel, select the checkbox next to the name of the audio track. Set the channel frequency of the VHF station to which background noise will be added.
- 5. Click OK.



6.2.5 VHF Notifications

Instructor can monitor VHF communications in all running sessions at the same time and can switch to the session where communication started and listen to it (or participate).

If the Instructor conducts multiple sessions and receives a call from a VHF communication station in an exercise that is currently not in focus, a notification appears in the lower right corner of the screen.



The notification indicates from which exercise the VHF connection request is sent and which station is calling.

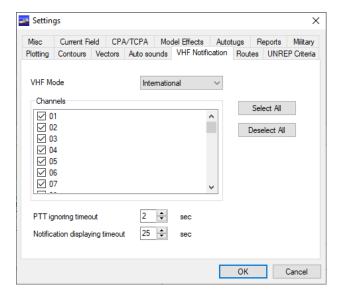
NOTE: Notifications must be enabled in Windows settings.

Clicking **Accept** will switch to the exercise from which the VHF communication is started. The exercise window will be opened in the **Navi-Trainer** instructor window and the **Instructor VHF** IBID panel will be reloaded (if it is used).

Clicking **Ignore** will ignore the call and close the notification. The notification will disappear after a timeout if no action was taken.

To configure notifications function:

- 1. Before the exercise start, choose the View\Settings... menu item. The Settings panel appears.
- 2. Open the VHF Notification tab.



- 3. In the VHF Mode drop down list, select the necessary mode:
 - o International international channels;
 - US USA channels;
 - ATIS European inland channels.

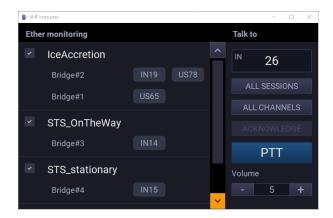
- 7. For each mode, in the **Channels** list, select (set a tick) channels to be monitored (from wich you would like to have notifications).
 - Use the Select All and Deselect All buttons to select (deselect) all the channels in the list.
- 8. In the he **PPT ignoring timeout** box, set a timeout for ignoring a PTT being pressed by trainee for less than X second (2 seconds by default).
- 9. In the **Notification displaying timeout** box, set a timeout for keeping a notification in the Windows notification panel for X second (25 seconds by default).
- 10. Click OK.

6.2.6 VHF Instructor Panel

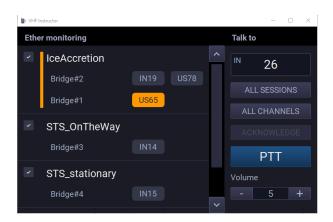
The VHF Instructor panel is designed to help the instructor monitor in one window communications in all running sessions simultaneously. The panel functions are similar to functions of the instructor VHF communication panel described above.

The VHF Instructor panel is a separate task in the Root configuration and has to be set up as described in the *Technical Description* document. According to the configuration settings, the panel can have a fixed size and position on the monitor, or have a title bar as Windows common window. Window with a title bar can be moved and resized as necessary.

A vertical scroll bar appears in the window if the list of sessions is longer than the window size. The scroll arrow is highlighted in orange if communication is activated in a session which line is out of scope.



The panel is divided into Ether Monitoring and Talk to parts.



The Either Monitoring part contains the list of sessions/bridges on which the VHF communication panels are present.

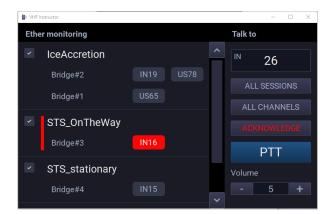
When communication is activated in a session its line is marked by an orange vertical bar.

Each line displays:

- Session (exercise) name and check box STS_OnTheWay. The check box is used to select the
 bridge/session for monitoring communications. If the check box is cleared the instructor will not receive calls from
 the bridge and will not be able to communicate.
- Bridge name Bridge#2
- Button with the channel number and its VHF mode IN19 . When a trainee at his/her workplace turns on the VHF communication station, the button displays the current channel number and mode. When a trainee starts a call the button is highlighted US65. The highlight color is red if the trainee sent a Distress signal.

The Talk to part contains:

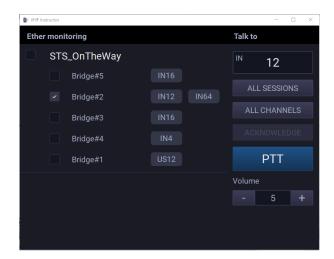
- Display to show the current working channel and VHF mode. To select a channel, click on the station name in the session/bridge list.
- ALL SESSIONS button to select for communication all sessions on the indicated channel. The button changes color to dark blue when pressed ALL SESSIONS.
- ALL CHANNELS button to select for communication all channels of selected sessions. The button changes color to dark blue when pressed
- ACKNOWLEDGE button to acknowledge a distress call. The button is disabled when no distress call is activated.
 The button is highlighted in red and enabled when a distress call is activated.
 To acknowledge the call, press this button. The indication display will show channel 16.



- PTT push-to-talk button.
- **Volume** volume control. Use the "-" and "+" buttons to set the desired value. The volume level is predetermined in the range from 0 (minimum level) to 9 (maximum level).

Common Scenario

In the common scenario, a session is configured to run the same exercise at several bridges. The ether monitoring list then contains selection check box for each bridge. It is possible to talk to all the bridges in the ssession or talk to a one selected bridge.



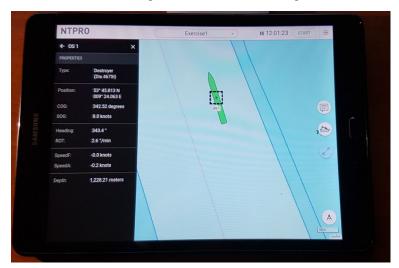
7. Exercise Remote Control

7.1	General	470
7.2	Running Remote Instructor Panel	471
7.3	Exercise Control	472
7.4	Displaying Ship Properties	473
7.5	Displaying Speed Vectors	474
7.6	Use of ERBL	475
7.7	Adding Event Markers	476

7.1 General

The Remote Instructor panel is an application running on an Android\Windows Tablet, communicating with NTPRO via a WEB Server. The WEB Server is a NTPRO Root application which provides access to the simulator data via Wi-Fi.

The Remote Instructor Panel displays exercise charts (same as on main Instructor workstation), own ships, target ships, speed vectors. The Instructor can select the active loaded session, pause/resume it, set different event markers, zoom chart, and get the basic vessel or target information.



7.2 Running Remote Instructor Panel

To run the Remote Instructor panel in the Web browser, use the following procedure:

1. In the tablet PC, open the Web browser.

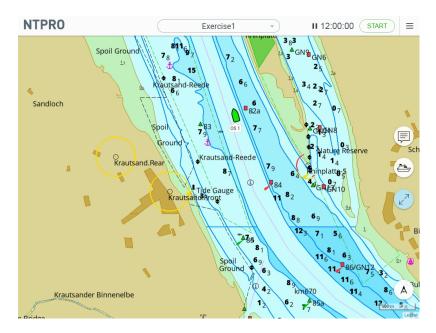
NOTE: It is advisable to use the Google Chrome browser.

2. In the browser address bar, enter the IP address or DNS name of the Web Server host.

For example, enter a web address: http://xxxxxxxx:8080/nti/:

- xxxxx IP address or DNS name of the Web Server host;
- 8080 HTTP port number.

The training area chart and objects of the exercise that is assigned to the bridge will appear on the screen:



To control the screen, use the following procedure:

- 1. To activate/deactivate the full-screen mode, click the button.
- 2. To zoom out a chart, bring together your two fingers on the touch panel. To zoom in a chart, spread apart your two fingers on the touch panel.

The chart scale is shown in the lower right corner of the screen.

7.3 Exercise Control

An exercise cannot be started on the tablet until it has been assigned to the bridge at the instructor's main workplace.

To start and pause an exercise on the tablet:

- 1. Click the START START button to start an exercise. The exercise timer starts 12:00:27 PAUSE
- 2. Click the PAUSE button to pause an exercise II 12:06:33 START
- 3. Click the **START** button to resume an exercise.

NOTE: It is not possible to stop or pause the log file playback on the tablet.

To switch between exercises:

1. Click the button with the name of the current exercise Exercise The list of exercises appears. It contains: bridge number, exercise name and zone (sailing area), current exercise time and status (Running/Paused).



NOTE: The mark shows that a log-file is assigned to this bridge.

2. Select the required exercise or log-file from the drop-down list. The chart of the selected exercise or log-file appears.

If the exercise is stopped at the instructor's main workplace, the following message appears on the tablet screen:



If all the exercises are stopped at the instructor's main workplace, the following message appears on the tablet screen:



7.4 Displaying Ship Properties

To display own ship and target ship properties, use one of the following operations:

- · Click the ship contour on a chart.
- Click the button and select the required ship/target from the Vessels list:



The **PROPERTIES** list appears to the left of the chart. The selected ship is marked with a square and centered in the chart.



The following ship properties are displayed in the **PROPERTIES** list: type, position, COG, SOG, heading, ROT, fore/aft speed, depth.

Click to hide the ship PROPERTIES list.

Click to go to the Vessels list.

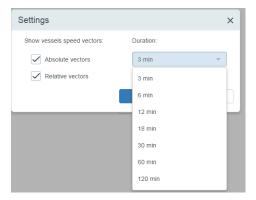
7.5 Displaying Speed Vectors

To show vessel speed vectors on the chart:

- 1. Click the button in the top right corner of the screen.
- 2. In the Settings window, select the required option: Absolute vectors and/or Relative vectors.



3. Select time duration from the **Duration** drop-down list.



4. Click the Apply button.

NOTE: To hide absolute and/or relative speed vectors on the chart, clear the corresponding checkboxes in the **Settings** window and click the **Apply** button.

- 5. If the Absolute vectors option is selected, the blue absolute speed vectors appear on the chart.
- 6. If the **Relative vectors** option is selected, click the reference vessel on the chart. The green relative speed vectors appear on the chart.



7.6 Use of ERBL

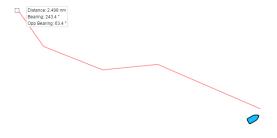
The ERBL can be used for measuring distance and bearing between two points on the screen.

To use the ERBL:

- 1. Click the button to activate the ERBL mode.
- 2. Click the first point on the chart, then click the second point on the chart. A red segment appears. The popup box shows the distance, bearing and reciprocal bearing between these two points.



If several segments are plotted on the chart, the total distance of all the segments is displayed in a popup box.



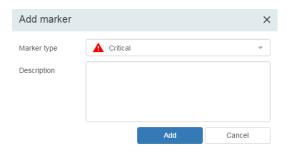
3. Click the button to deactivate the ERBL mode.

7.7 Adding Event Markers

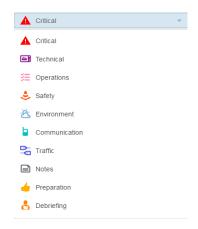
The Instructor can set different event markers. The event message will be displayed on the **Events** panel in the Nave-Trainer Instructor application (menu **View\Panels...\Events**).

To add event markers, use the following procedure:

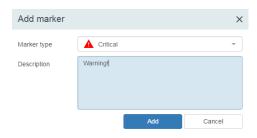
1. Click the button. The Add marker window appears.



2. Select the marker type from the Marker type drop-down list.



3. Enter the event description in the **Description** text field.



4. Click the Add button.

Annex. Menu Commands Description

Table 20 . Description of menu commands

MENU	PURPOSE	HOT KEYS	BUTTON
File	Handling exercise files	<alt>+<f></f></alt>	
New	To create a new exercise and select training area	<ctrl>+<n></n></ctrl>	9
Open	To select and load an existing exercise file (route, log file, log scenario)	<ctrl>+<o></o></ctrl>	*
Close	To close an exercise		
Save	To save an exercise	<ctrl>+<s></s></ctrl>	
Save As	To save an exercise under another name		
Control\Assign	To distribute tasks among the bridges	<ctrl>+<1></ctrl>	
Control\Start	To start an exercise (log file)	<ctrl>+<2></ctrl>	
Control\Pause	To suspend\continue exercise (log file) playback	<ctrl>+<3></ctrl>	
Control\Stop	To stop an exercise	<ctrl>+<4></ctrl>	
Control\Rewind	To rewind an exercise	<ctrl>+<5></ctrl>	(4)
Configuration\Open	To select and load an existing configuration		-20
Configuration\Save	To save a configuration		3
Configuration\Save As	To save a configuration under another name		
Configuration\Set as Default	To set a current configuration as default		Q ;
Configuration\Auto Save Configuration	To save configuration automatically		
External Connections	To adjust external connections with other simulators		
Page Setup	To call the print setup window		
Print Preview	To preview a page before printing		
Print	To print out active exercise window	<ctrl>+<p></p></ctrl>	3
Print Screen	To print out the screenshot with the current printout settings		2

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	виттом
Recent Files	To load recently used exercise files		
Exit	To exit from the program		
Edit	Creating and editing exercises	<alt>+<e></e></alt>	
Comments	To display a window for comments entering		7
Environment Settings	To display a panel for entering the exercise start date, time and hydro meteorological conditions for the entire scene		4
Unit Settings	To display a panel for setting measurement units		
Change Area	To change current training to another area		
Object Properties	To show the object properties window		3
New Object	To add a new exercise object	<ctrl>+<shift>+<lns></lns></shift></ctrl>	2
Cut Object	To copy the object data to the clipboard and delete it from the exercise	<ctrl>+<x></x></ctrl>	*
Copy Object	To copy the selected object	<ctrl>+<c></c></ctrl>	P
Paste Object	To paste the object from the clipboard to the exercise	<ctrl>+<v></v></ctrl>	1
Delete Object	To delete an object from the exercise	<ctrl>+<shift>+</shift></ctrl>	***
Append Point	To add a route WP or zone node	<ctrl>+<ins></ins></ctrl>	%
Delete Point	To delete a route WP or zone node	<ctrl>+</ctrl>	
Readonly Mode	To switch on/off the Readonly mode for such objects as zones, routes, lines, points		%
Set Radar Position	To set the Instructor's Radar on the chart		
Add Camera	To add the Instructor's camera		84
Create Template	To create a template for moving groups of objects to any other place on the chart or to another sailing area		

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	BUTTON
Insert Template	To insert a template in the selected place	<ctrl>+<shift>+<t></t></shift></ctrl>	
Create Fleet Formation	To open the window for fleet formation creating		200
Import Route	To open the window for loading the available route file	<ctrl>+<shift>+<r></r></shift></ctrl>	
Import Current Database	To open the window for loading the available current database file		
Import KML/KMZ file	To open the window for loading the available KML/KMZ file		
Chart	Handling charts	<alt>+<c></c></alt>	
Centre	To centre the chart in the cursor position	<ctrl>+<alt>+<c></c></alt></ctrl>	Q
Reference Point	To set a Reference point	<ctrl>+<shift>+<c></c></shift></ctrl>	+
Zoom Area	To zoom a rectangular chart fragment	<ctrl>+<z></z></ctrl>	
Load Chart	To display a chart/put it on top	<ctrl>+<l></l></ctrl>	
Unload Chart	To delete a chart/put it down	<ctrl>+<shift>+<l></l></shift></ctrl>	
ERBL	To call an electronic range and bearing line	<ctrl>+<e></e></ctrl>	
Chart Info	To obtain general information on any chart fragment	<ctrl>+<l></l></ctrl>	
Force assist	To apply the assist force to any point of the object hull		F
Increase Scale	To increase the chart display scale	<ctrl>+<up> or <+></up></ctrl>	
Decrease Scale	To reduce the chart display scale	<ctrl>+<dn> or <-></dn></ctrl>	
Undo Area	To cancel changes made in the chart display	<ctrl>+<backspace></backspace></ctrl>	8
Overlays\Borders	To show/hide chart boundaries		
Overlays\Grid	To show/hide geographic grid		
Overlays\Rings	To show/hide range rings		

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	BUTTON
Overlays\TX-97	To turn on/off the display of TX-97 format chart elements		
Overlays\DCW converted	To turn on/off the display of DCW converted format chart elements		
Overlays\S57 v3	To display the depth contours on the S-57 format charts		
Overlays\User Info	To show/hide user objects on the chart		
View	Setting the instructor console elements for the display	<alt>+<v></v></alt>	
Panels\Audio logger	To open the panel for control of audio logger		4
Panels\Auxiliary machinery	To open the panel for auxiliary machinery control		**
Panels\Camera control	To open the panel for the instructor visualization control		=
Panels\Control Panels	To open the panel for the control of the ballast water loading and discharging		=
Panels\CPA/TCPA	To open panel for control CPA/TCPA options		-
Panels\Custom alarms	To open the panel for creating custom alarms		
Panels\Depth Section	To open the depth section panel		
Panels\DP System	To open the DP system control panel		DP
Panels\E-Tutor	To open the "e-Tutor" Scenario Editor panel		12
Panels\E-Tutor Performance Monitor	To open the panel for monitoring the trainee performance in the "e-Tutor"		
Panels\ECDIS Training	To open the panel for electronic charts order processing		2
Panels\Events	To open the panel for monitoring of the exercise events		© <u>₹</u>
Panels\Faults	To open the panel for setting the own ship faults		热
Panels\Flags	To open the panel for setting flag signals)_K

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	BUTTON
Panels\Flash Light	To open the panel for flash light signaling		•
Panels\Fleet management	To open the panel for fleet formation control		25
Panels\GNSS Control	To open the panel for monitor of the GNSS satellites		1-1
Panels\GPS Synchronization	To open the panel for adjustment of synchronization with an external source of GPS data		GPS 0
Panels\Helicopters	To open the panel for deck helicopter control		lacksquare
Panels\Infrared camera	To open the panel for advanced infrared camera control		ir •0
Panels\Intercom	To open the panel for Intercom communication control		
Panels\LogScript	To open the panel for recording the exercise log script		
Panels\Magnetic deviation	To open the panel for setting magnetic deviation values		*
Panels\Motion Platform	To open the panel for motion platform control		<u>₹</u> ₹
Panels\Navigational Signals	To open the panel for navigational signal control		(1)
Panels\Navtex	To open the panel for transmitting NAVTEX messages		
Panels\Objects	To open the panel with list of all exercise objects		*
Panels\Pirates	To open the panel for pirate attacks control		(
Panels\Radio Aether	To open the panel for radiotelephone exchange during the joint operation with GMDSS simulator		
Panels\Remote sensors	To open the panel for remote sensors control		
Panels\Reports	To open the exercise report table		
Panels\Route planning	To open the panel for setting and editing routes in a tabular form		
Panels\Screen capture	To open the panel for capturing the Visualization screen		

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	BUTTON
Panels\Ship info	To open the panel of general information on the ships		□ <u></u>
Panels\Spectrum Editor	To open the panel for editing spectrums of user-defined wind wave		<u>**</u>
Panels\Submarine control	To open the panel for submarine control		
Panels\Submarine control (Gotland)	To open the panel for Gotland- class submarine control		4
Panels\Time Display	To open the exercise time panel		(5500)
Panels\UNREP	To open the panel for underway replenishment control		
Panels\VHF Background noise	To open the panel for loading and playing back the sound files with samples of VHF background noise		2
Panels\VHF	To open the panel for VHF-communication control		
Panels\Video logger	To open the panel for control of video logger		*
Panels\Visual Profiles	To open the panel for selecting a visual profile file and loading it on the bridge		<u></u>
Panels\Visual Settings	To open the panel for visual settings adjustment		©
Panels\Wind Turbines control	To open the panel for wind turbine control		
Palettes\Day	To set the "Day" colour palette		Abc
Palettes\Twilight	To set "Twilight" colour palette		Abc
Palettes\Dusk	To set the "Dusk" colour palette		Abe
Palettes\Night	To set the "Night" colour palette		Abé
Regimes\TM 'Follow Up'	To select the True Motion mode	<ctrl>+<alt>+<f></f></alt></ctrl>	
Regimes\RM 'Nord Up'	To select the relative motion mode with North Up orientation	<ctrl>+<alt>+<n></n></alt></ctrl>	4
Regimes\RM 'Head Up'	To select the relative motion mode and Head Up orientation	<ctrl>+<alt>+<h></h></alt></ctrl>	

Table 20. Description of menu commands (continued)

MENU	PURPOSE	HOT KEYS	BUTTON
Regimes\RM 'Fixed Heading'	To select the relative motion mode with fixed heading	<ctrl>+<alt>+<x></x></alt></ctrl>	<u>F</u>
Vectors\Absolute	To turn on/off absolute motion vector on the exercise chart	<ctrl>+<alt>+<a></alt></ctrl>	™ A
Vectors\Relative	To turn on/off the relative motion vector on the exercise chart	<ctrl>+<alt>+<r></r></alt></ctrl>	™ R
Offshore panel	To open the offshore operation control panel		HU.
Settings	To open the window with general display settings	<ctrl>+<alt>+<s></s></alt></ctrl>	**
Customize	To open the panel for customizing menu bar and toolbar		
Window	Handling windows	<alt>+<w></w></alt>	
New Window	To create a new chart window for an active exercise		
Cascade	To arrange chart windows in a cascade		=
Tile Horizontally	To arrange chart windows horizontally		
Tile Vertically	To arrange chart windows vertically		
The list of windows	To display exercise windows opened before		
Help	Help	<alt>+<h></h></alt>	
Contents	To call online help		
Area	To obtain information on the sailing area		
Ships	To obtain information on ships		
About	To obtain information on the developers		



Navi-Trainer Professional 6

Instructor Manual

Version 6.0

Date of issue: February 2024

Technical support

Phone number: + 46 771 460 100

Email: voyage.support@wartsila.com

Web: www.wartsila.com