II. Scenarios

Whenever you use VR-Forces, you work in the context of a scenario. A scenario consists of everything that needs to be specified to run a VR-Forces simulation. Each scenario has a scenario file (*.scn*) and these supporting files:

- A scenario objects file (*.oob*).
- A plan file (*.pln*).
- An object map file (*.omp*).
- A terrain database for the GUI and sim. engine.
- An overlay file (*.ovl*).
- A scripts file (*.spt*). This file contains the scenario scripts, if any.
- A selection groups file (*.sgr*).
- A scenario "extras" file (*.xtr*). This file records hostility relationships and stores spawn pattern templates.
- A saved views file (*.osrx*). If a scenario has captured observer views, they are saved automatically. You can also specify a startup view.
- A settings file that records scenario-specific GUI settings (*.gui_settings*).

By default, scenarios are saved in a compressed zip archive with the extension *.scnx*. If you examine a compressed scenario file in an unzipping application, you will see that it contains the individual files described in the previous paragraphs.

A terrain database does not have to be stored in the same directory as a scenario. It is referenced by the scenario. This lets you use a terrain database, which can be quite large, in more than one scenario, without needing to make multiple copies of it. For more information about terrain databases, see "Terrain".

For details about scenario files, see "Chapter 7. Scenario Files" on page 123.

2. Creating and Running Scenarios

This chapter explains how to create, load, run, and save scenarios.

2.1. Creating a Scenario

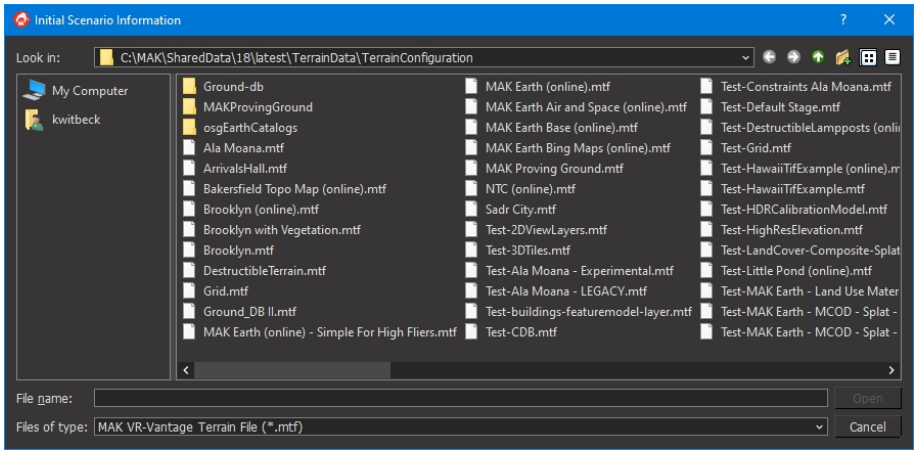
To use VR-Forces to simulate simulation objects, you must load a scenario or create a new one. When you start VR-Forces, the Scenario Startup screen asks you if you want to create or load a scenario (unless you have disabled this feature). Thereafter, to create a scenario, use the procedure in this section. (For details about creating a scenario from the Scenario Startup screen, see "Starting VR-Forces from the VR-Forces Launcher".)

> ⓘ If you are using multiple GUIs in the same session, when one of them creates a scenario, the terrains in all other GUIs are closed (unless the terrain is the same as that for the new scenario) and the terrain for the new scenario is loaded.

➤ **To create a scenario:**

1. Choose **File > New Scenario**, or click the New Scenario button () on the File Toolbar. The Initial Scenario Information dialog box opens (Figure 1). By default, it opens in *./userData/terrains*, the directory that contains the terrain files shipped with VR-Forces. (For a brief description of the databases supplied with VR-Forces, see "Terrain Databases Provided with VR-ForcesMAK ONE Applications". For details about terrain files, see "Terrain".)

Figure 1. Initial Scenario Information dialog box

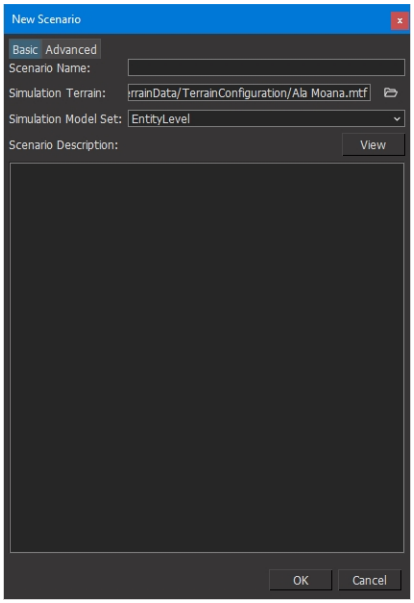


2. In the file list, select a terrain database. (MTF is the preferred file type.) A database must be a flat earth, geocentric, UTM, or Lambert Conformal Conic database. By default, this terrain is used for both the GUI and sim. engine.

> ⓘ If you choose an MTF file and an MTD file with the same name exists, the sim. engine loads the MTD file. (MTD is a legacy format that may be present for older terrains.)

3. Click Open. The New Scenario dialog box opens (Figure 2).

Figure 2. New Scenario dialog box



4. Select the simulation model set (SMS) for the type of scenario you want to run: entity-level or aggregate-level.

5. Optionally, set other basic scenario parameters. Table 1 describes the basic parameters.

Table 1. Basic scenario parameters

Parameter	Description
Scenario Name	Specifies the name of the scenario. This is the name used to identify the scenario when you join a session. If you do not specify a name, the scenario file name is used.
Simulation Terrain	Specifies the terrain to be used by the sim. engine. By default, this is the terrain you selected in the Initial Scenario Information dialog box. However, you can change it if you want to.
Simulation Model Set	The simulation model set to use. For entity-level scenarios, choose *EntityLevel.sms* or an SMS similar to it. For aggregate-level scenarios, choose *AggregateLevel.sms* or a similar SMS.
Scenario Description	The scenario description lets you provide a description of the scenario. This may be useful to remind you or other VR-Forces users of its purpose or special details. You can enter the description as plain text or as HTML. If you use HTML, when the description is displayed in the Scenario Properties dialog box, it is displayed using the HTML formatting.
Attach Components to Remote Entities On	Specifies whether to attach components to remote entities, and if so, the sim. engines on which to manage the components that are being attached to remote entities. Unlike other scenario parameters, you can change this value when you load a scenario. For details about configuring components for attachment to remote entities, see "Attaching Components to Remote Entities".

For more information about these parameters, see "7.2.1 Scenario Parameters" on page 125.

6. Optionally, select the Advanced tab (Figure 3) and set advanced scenario parameters (overriding the default settings).

Figure 3. New Scenario dialog box, Advanced tab

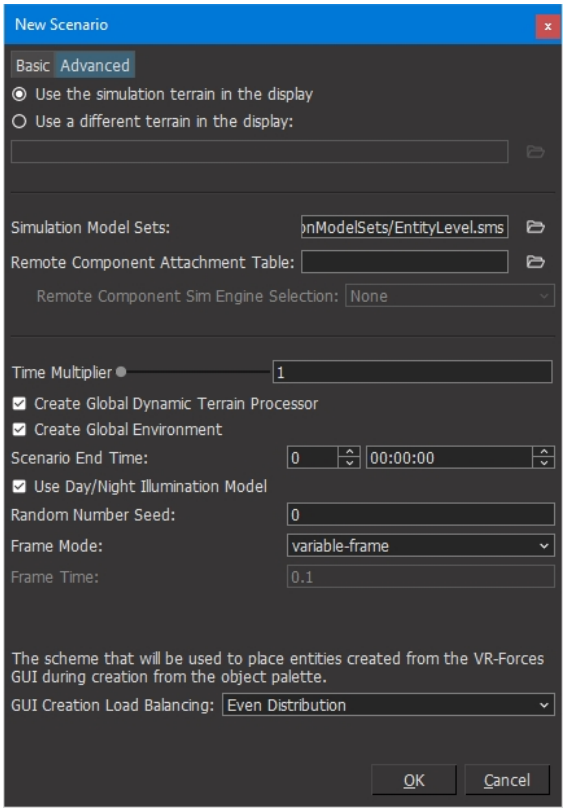


Table 2 describes the parameters you can set.

Table 2. Advanced scenario parameters

Parameter	Description
Use the Simulation Terrain in the Display	If selected (the default), the scenario uses the same terrain for the GUI and sim. engine.
Use a Different Terrain for the Display	If selected, lets you specify the terrain to be used by the GUI.
Simulation Model Sets	Lets you specify simulation model sets other than or in addition to the default SMSs listed on the Basic tab. A scenario must specify at least one simulation model set. For more information, see "2.1.1 Specifying Multiple Simulation Model Sets" on page 9. Default: *EntityLevel.sms*.

Table 2. Advanced scenario parameters (continued)

Parameter	Description
Component Attachment Table	If you plan to attach components to remote entities, specify the component attachment table that you want to use. For details about configuring components for attachment to remote entities, see "Attaching Components to Remote Entities".
Time Multiplier	Specifies a time multiplier for the scenario. For HLA time-managed federations, this parameter is ignored. Default: 1.0 (real-time).
Create Global Dynamic Terrain Processor	If selected, the global dynamic terrain processor monitors the entire terrain for dynamic terrain changes. If disabled, dynamic terrain is not supported unless you create a dynamic terrain area.
Create Global Environment	If selected, global weather is supported. If cleared, the settings on the Environment Conditions dialog box, Weather page have no effect on the scenario. (You can still create weather zones.)
Scenario End Time	The simulation time at which to automatically pause the scenario. For more information, see "2.9.1 Setting the Scenario End Time for a New Scenario" on page 42.
Use Day/Night Illumination Model	If selected, use an ephemeris model in the sim. engine to calculate illumination. This affects sensors. If cleared, the entire world is fully illuminated regardless of the time of day. For more information, see "6.3.3 Choosing the Illumination Model" on page 106.
Random Number Seed	Specify a random number seed as an integer >= 0.
Frame Mode	Select a frame mode from the list.
Frame Time	Specifies the length of a frame, in seconds. This parameter is not enabled if Frame Mode is variable-frame.
GUI Creation Load Balancing	Specifies how to load balance simulation objects created in the GUI when you have more than one simulation engine. If you select Even Distribution, VR-Forces automatically distributes objects across the available sim. engines. If you select Rule Based Distribution, the Rule Set list becomes available and you can choose an existing rule set or create a new one. See "Rule Based Load Balancing" on page 32.

For more information about these parameters, see:

- "Simulation Time".
- "Exercise Clock Modes".
- "7.2.1 Scenario Parameters" on page 125.

7. Click OK. The scenario information is displayed and the terrain is displayed in the Plan View (2D) observer mode.
8. Add the simulation objects and tactical graphics that you need for your simulation. For information about adding simulation objects and tactical graphics, see "Creating and Placing Objects".
9. Save the scenario. For details about saving scenarios, see "2.3. Saving a Scenario" on page 22.

2.1.1 Specifying Multiple Simulation Model Sets

You can specify more than one simulation model set for a scenario. One strategy for extending VR-Forces is to use an SMS provided with VR-Forces for the majority of objects and to create a separate SMS for customized simulation objects. As you upgrade to newer versions of VR-Forces, you can more easily migrate your customized SMS to the new version because your changes have not been integrated into the default SMS. (You can also include the SMSs provided by VR-Forces in your custom SMS. In that case you would just load the custom SMS and not multiple SMSs. For more information, see "Simulation Model Sets".)

If you routinely use more than one SMS, consider creating an SMS configuration. For details, see "2.1.2 Creating Simulation Model Set Configurations" on the next page.

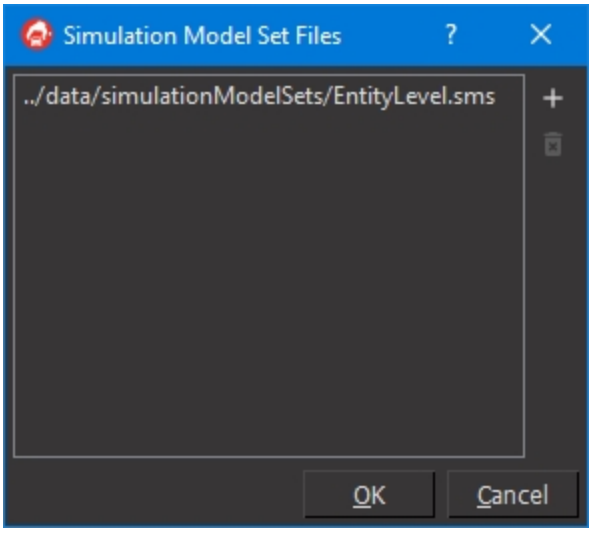
When you load multiple SMSs, if there is any overlap in the objects they specify, the last SMS loaded overrides previous SMSs.

> ⚠ Do not use an entity-level SMS and an aggregate-level SMS together. The simulation objects from the two SMSs will not be able to interact.

➤ **To specify multiple simulation model sets for a scenario:**

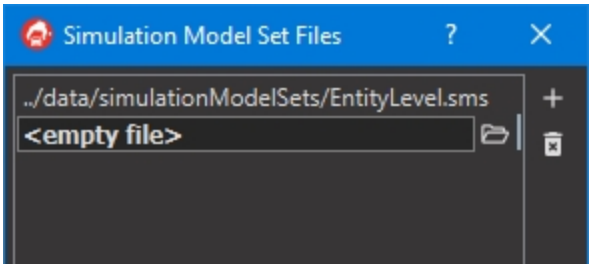
1. In the New Scenario dialog box, click the Browse button on the Simulation Model Set line. The Simulation Model Set Files dialog box opens. It lists the SMS files to be loaded for this scenario.

Figure 4. Simulation Model Set Files dialog box



2. In the Simulation Model Set Files dialog box, click Add Model Set. A blank line is added to the window (Figure 5).

Figure 5. Add simulation model set



3. Click the Browse button next to the blank line. The Choose File dialog box opens.
4. Select a simulation model set and click Open. The SMS is added to the text box.
5. Click OK.
6. Complete the New Scenario dialog box.

2.1.2 Creating Simulation Model Set Configurations

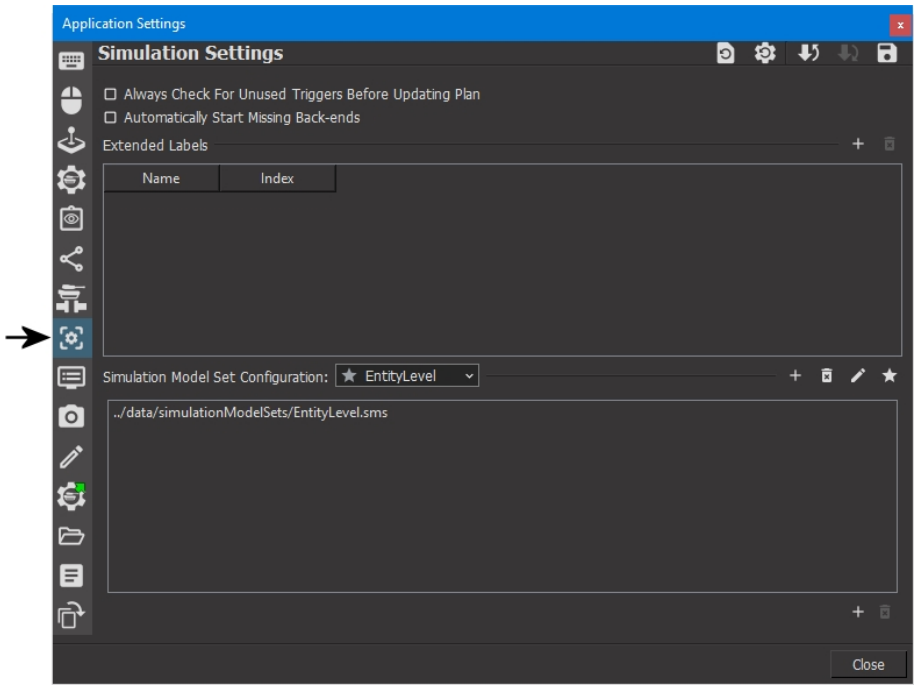
When you create a new scenario, you specify an SMS in the Initial Scenario Information dialog box (see "2.1. Creating a Scenario" on page 4 for details). The Simulation Model Set list lists all SMSs in *./data/simulationModelSets*. The SMSs are listed in alphabetical order. If you want a particular SMS to be the default SMS you can create an SMS configuration and specify that it be the default. You can create as many SMS configurations as you want. An SMS configuration can have more than one SMS in it. So if you routinely create scenarios that use multiple SMSs, creating a configuration can save you from the necessity of always specifying the additional SMSs on the Advanced tab of the New Scenario dialog box (as described in "2.1.1 Specifying Multiple Simulation Model Sets" on the previous page).

> The Simulation Model Set list in the New Scenario dialog box lists all SMS configurations and all SMSs. It does not duplicate names, so if an SMS configuration has the same name as an SMS, only the SMS configuration is listed.

➤ **To create a simulation model set configuration:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Simulation Settings page (Figure 6).

Figure 6. Simulation Settings page



3. Click the Add button (+) on the Simulation Model Set Configuration line. The Configuration dialog box opens.
4. Type a name for the configuration.
5. Click OK. The new configuration is added to the list.
6. Click Add Model Set. A line is added to the window.
7. Click the Browse button at the end of the new line. A file chooser dialog box opens.
8. Select the SMS you want to put in this configuration.
9. Click Open. The SMS is added to the list.
10. Optionally, add additional SMSs to this configuration.
11. Click Close.

Specifying the Default SMS Configuration

You can specify that a particular SMS configuration be selected in the Initial Scenario Information dialog box whenever you create a new scenario.

> ⓘ Specifying an SMS as the default SMS automatically removes that designation from any other SMS.

➤ **To specify the default SMS configuration:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Simulation Settings page (Figure 6 on the previous page).
3. In the Simulation Model Set Configuration list, select the configuration that you want to be the default.
4. Click the Star button (★). A star displays next to the configuration name in the list, indicating that it is the default configuration.

2.1.3 Setting the Scenario Starting Date and Time

By default, when you create a scenario, the starting simulation date and time is the current system date at 10:00 A.M. local time, where the local time zone is determined by the longitude of the first simulation object that you add to the scenario.

You can change the date and time at which a scenario starts in either of the following ways:

- A date and time other than the default, but still at the location where the first simulation object is created.
- Use a date, time, and time zone without regard to where the first simulation object is created.

By default, before you create the first simulation object, as you move the observer through the terrain, the time shown on the Time of Day toolbar and in the Date and Time page changes to show you the time at the location of the observer relative to the selected time zone. (The Time of Day toolbar and the Date and Time page defaults to Coordinated Universal Time (UTC, also called Zulu time).) Furthermore, the sun moves with the observer so that the observer location is always in daylight.

> ⓘ The time zone is an approximation. It does not take into account unusual time zone boundaries or daylight saving time. The time zone is set once. If you create a simulation object and then delete it, the location of that entity still determines the time zone. Furthermore, if for some reason you were to create a scenario and then run it without creating any simulation objects, if at any point you created a simulation object, the starting date and time would be reset to the default value based on the location of that entity.

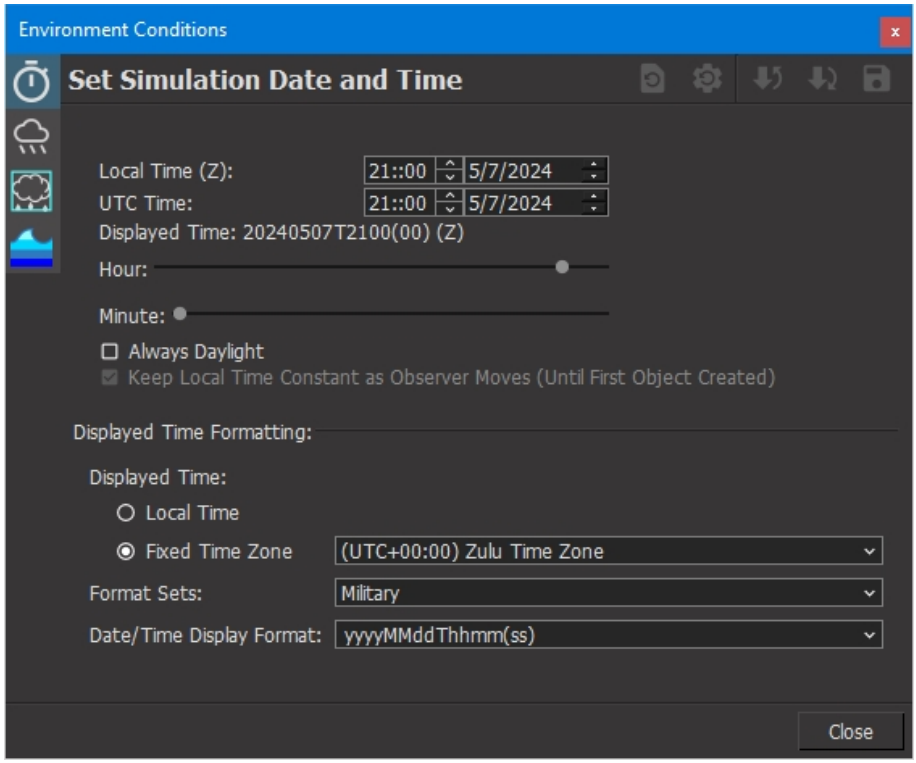
You can change the starting date and time when you create a scenario.

For more information about simulation time, see "6.3. Setting the Date and Time of Day" on page 103.

➤ **To set the scenario start date and time using the time zone of the first simulation object:**

1. Begin the scenario creation process, as described in "2.1. Creating a Scenario" on page 4.
2. Before you create the first simulation object, choose **Simulation > Environment Conditions**, or click the date on the Environment Settings Toolbar. The Environment Conditions dialog box opens.
3. Select the Date and Time page (Figure 7).

Figure 7. Set Simulation Date and Time page



4. Change the date and current time to the desired date and current time.
5. Create a simulation object in the desired location.

➤ **To set the scenario start time and time zone without respect to the first simulation object:**

1. Begin the scenario creation process, as described in "2.1. Creating a Scenario" on page 4.
2. Before you create the first simulation object, choose **Simulation > Environment Conditions**, or click the date on the Environment Settings Toolbar. The Environment Conditions dialog box opens.
3. Select the Date and Time page (Figure 7).
4. Clear the Keep Local Time Constant as Observer Moves (Until First Object Created).
5. Set the date, time, and time zone that you want the scenario to start at.

6. Optionally, specify the date and time format (standard or military) and the display format.
7. Continue the scenario creation process.

2.2. Importing Objects into Scenarios

If you frequently need to create scenarios with the same simulation objects, scenario-level scripts, or other scenario components, consider using one or more of these strategies:

- Create simulation object groups. Simulation objects groups are heterogeneous groups of simulation objects and tactical graphics that you can add to any scenario. Like the individual simulation objects that you can add to a scenario, they are not tied to a terrain location until you add them. Use them if you want to create a repeatable set of objects that do not fit into a unit hierarchy. For more information, see "Simulation Object Groups".
- Use scenario import. If you have a base scenario that you want to use repeatedly, create the scenario, then open it and rename it to preserve the original, or open a new scenario and import the base scenario. You can do this with a fully realized scenario or a scenario layout that you want to add to other scenarios. For example, you might want to create a town with background traffic for repeated use with different training problems added to it. Since this is a scenario, it is tied to the terrain location you use when you create it. Alternatively, you might have a set of scripts that you do not want to promote to system scripts. You can save them in a scenario and import the scenario into the scenarios for which you want to use the tasks. For more information, see "2.2.1 Importing (Merging) Scenarios" below.
- Import MSDL. If you have a MSDL entity laydown, you can import that into different scenarios as needed. For more information, see "2.2.2 Importing and Exporting MSDL" on page 17.
- Import point, linear, or areal objects from a CSV file. For details, see "2.2.3 Importing Scenario Objects from CSV Files" on page 19.
- Export an Order of Battle and import it into other scenarios. For details, see "Creating and Using an Order of Battle".
- Export tactical graphics to shapefiles. For details, see "Exporting Tactical Graphics as Shapefiles".

2.2.1 Importing (Merging) Scenarios

VR-Forces users often assign scenario development to multiple people or teams. Then they merge the separately-developed scenarios into a master scenario. You can easily merge scenarios by importing them. When you import a scenario, you can choose to rename simulation objects to avoid duplicate names, to import scenario events, and to create a selection group for the imported simulation objects.

Some object names in the scenario you are importing might conflict with (that is, might be the same as) object names in the base scenario. You have three options for handling object name conflicts:

- Allow duplicate names in the merged scenario.
- Give every object in the imported scenario a new name by adding the prefix specified in the Deconflicting Name field. The Deconflicting Name prefix is applied to every object name, not just conflicting names, except as noted below in "Object Names in Merged Scenarios" on the next page.
- Change only conflicting names in the imported scenario.

The procedure in "Importing a VR-Forces Scenario" below explains how to use each of these options.

> ⚠ If you import a scenario that was saved (checkpointed) with a simulation time that is different from the current time of the scenario into which you are importing it, there may be unexpected behaviors due to timing issues. For example, an entity executing a Wait Duration task may wait for a longer or shorter time than you expect.

What Is (and Is Not) Included in the Import

Not everything associated with a scenario is included.

What is imported:

- Simulation objects without conflicting UUIDs, including entities, units, and tactical graphics.
- Simulation objects with conflicting UUIDs, only when Discard Duplicate Simulation Objects check box is not selected.
- Scenario events, only when Import Scenario Events is selected.
- Global plans.
- Synchronization matrices.

What is not imported:

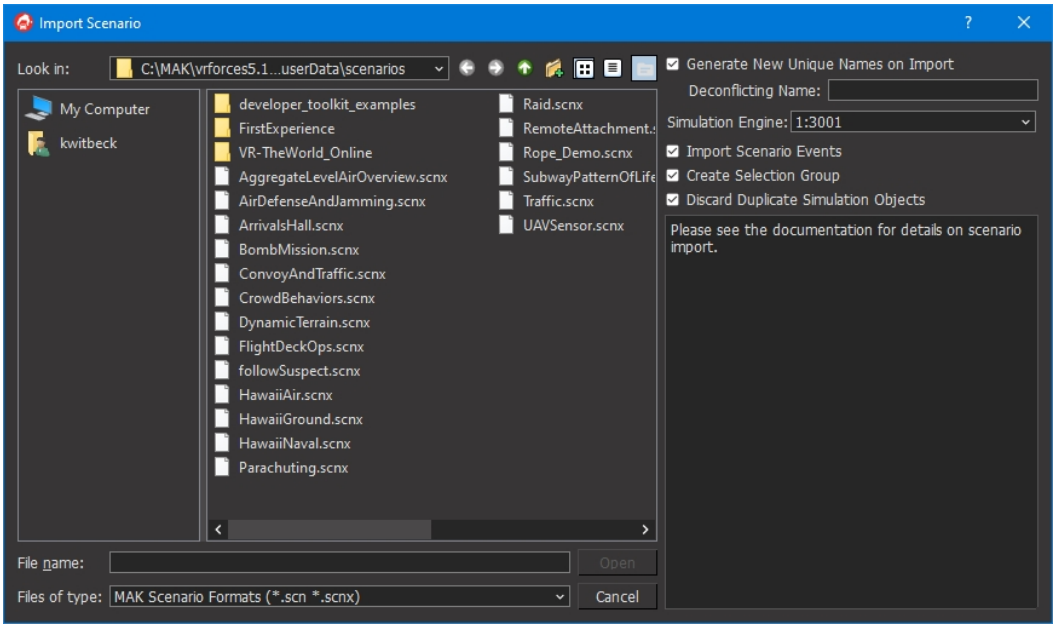
- The SMS associated with the imported scenario.
- The global environment, including the weather, date, and time of day.
- The terrain.
- Observer views.
- Dynamic terrain changes made by the global dynamic terrain manager object.
- Simulation objects with conflicting UUIDs, only when Discard Duplicate Simulation Objects check box is selected.

Importing a VR-Forces Scenario

➤ **To import a scenario:**

1. Open a scenario.
2. Choose **File > Import VR-Forces Scenario**. The Import Scenario dialog box opens (Figure 8).

Figure 8. Import Scenario dialog box



3. Select the scenario that you want to import.
4. Execute only one of the following:
   - To allow duplicate names in the merged scenario, clear the Generate New Unique Names on Import check box.
   - If you want VR-Forces to give every object in the combined scenario a unique name:
     - Make sure the Generate New Unique Names on Import box is checked.
     - In the Deconflicting Name field, enter a text string for VR-Forces to prepend to object names in the imported scenario. (VR-Forces adds the string to all object names, not just to those that match names in the base scenario, except as noted below in "Object Names in Merged Scenarios" below.)
   - To change only conflicting names in the imported scenario:
     - Make sure the Generate New Unique Names on Import box is checked.
     - Clear the Deconflicting Name field.
5. Optionally, specify whether to import scenario events.
6. Optionally, specify whether to create a selection group.
7. Optionally, specify whether to discard duplicate simulation objects.

Object Names in Merged Scenarios

The deconflicting names feature treats units (platoons, for example) and platforms (vehicles) differently from other objects:

- VR-Forces applies the prefix you enter in the Deconflicting Names field to every unit in the imported scenario, whether its name conflicts or not.
- VR-Forces renames platforms in the imported scenario that have names that conflict with platforms in the base scenario, but does not use the prefix to do it. Instead, VR-Forces changes the number at the end of the platform's name, so a tank called M1A2 1 in the imported scenario might become M1A2 4 in the merged scenario.

In addition, if adding the Deconflicting Names prefix would make an entity's name too long, VR-Forces gives the entity a new unique name that does not include the prefix.

2.2.2 Importing and Exporting MSDL

Military Scenario Definition Language (MSDL) is a SISO standard for describing military scenarios using an XML format. It allows for interchange of scenario descriptions between simulation applications and systems that use different internal formats. You can import MSDL files into a VR-Forces scenario.

VR-Forces does not import all of the information that can be in an MSDL file. It imports and exports:

- Simulation objects and their locations. Locations are always exported in geocentric coordinates.
- Force relationships and hostility relationships. Any force with the word "neutral" in it (not case sensitive) is considered a neutral force, and all other forces are considered neutral to it unless specifically marked as hostile in the VR-Forces hostility matrix. Other forces not marked as hostile to one another are considered friendly.
- Country codes. If the scenario's hostility matrix does not have countries assigned to forces, the country code is empty. You should define countries for the forces, including the Neutral force, to have country codes in the MSDL file.
- Unit formations.
- Tactical graphics.
- Area of Interest (bounding box).

ⓘ If you try to import a unit type that is not available in the Simulation Object Editor, you will not be able to import it because it cannot be assigned to a unit type. You can add empty units in the Simulation Object Editor that will then be available for MSDL import. For more information, see "Editing Units for Entity-Level Scenarios".

➤ **To import MSDL:**

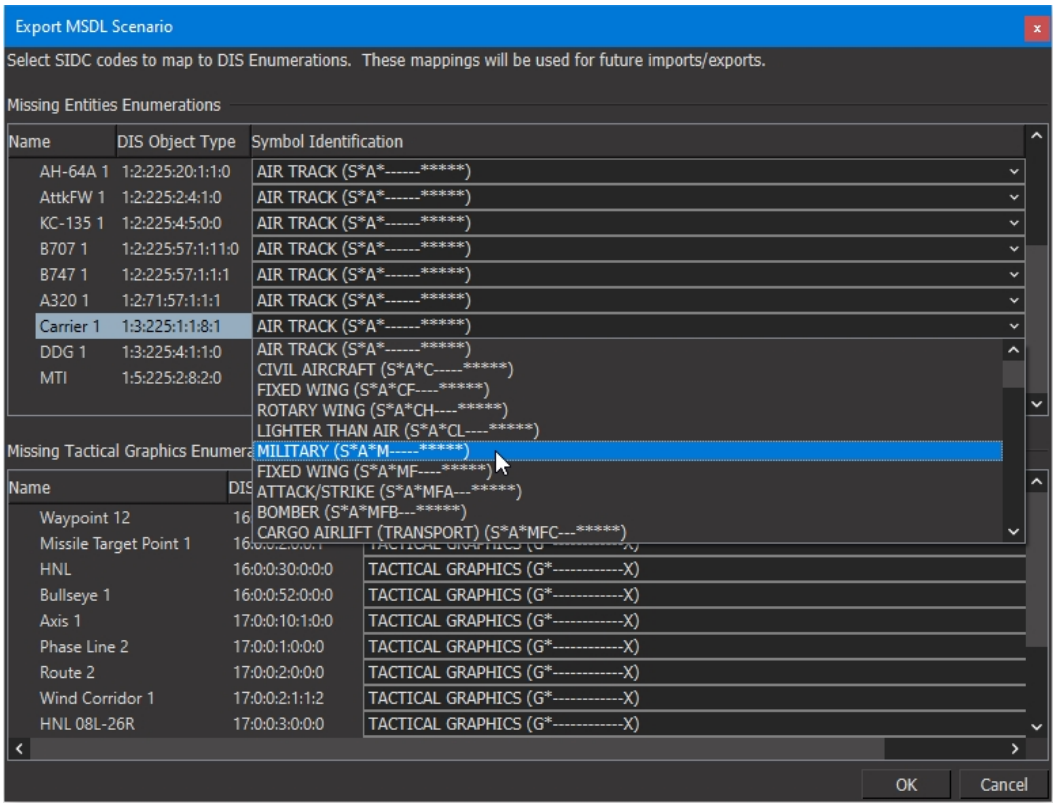
1. Create or load a scenario.
2. Choose **File > Import Scenario Objects**. The Import Scenario File dialog box opens.
3. In the Files of Type box, set the filter to MSDL Scenario File (*.xml).
4. Select the MSDL file that you want to import.
5. Click Open.

Exporting MSDL

➤ **To export a scenario to MSDL:**

1. Create or load a scenario.
2. Choose **File > Export Scenario Laydown**. The Export Scenario dialog box opens.
3. In the Save As Type box, set the filter to MSDL Scenario File (*.xml).
4. Select the directory in which you want to save the file.
5. Type a name for the file.
6. Click Save. If VR-Forces is not sure how to export an entity type to MSDL, the Export MSDL Scenario dialog box opens (Figure 9).

Figure 9. Export MSDL Scenario dialog box



7. For each object type listed in the Export MSDL Scenario dialog box:
   a. In the Symbol Identification column, click the value. A list of possible identification codes is displayed. (These codes are from the MSDL specification, MILSTD 2525B. You need to familiarize yourself with the specification to fully understand them.)
   b. Select the proper identification.
8. Click OK.

2.2.3 Importing Scenario Objects from CSV Files

Some VR-Forces users like to plot tactical graphics and simulation object locations offline and then import them into a scenario. VR-Forces supports importation of points, lines, and areas from comma separated values (CSV) files.

A CSV file can contain only one kind of object:

- One area.
- One route.
- One or more point objects. Points can represent tactical graphics, such as waypoints, or individual entities.

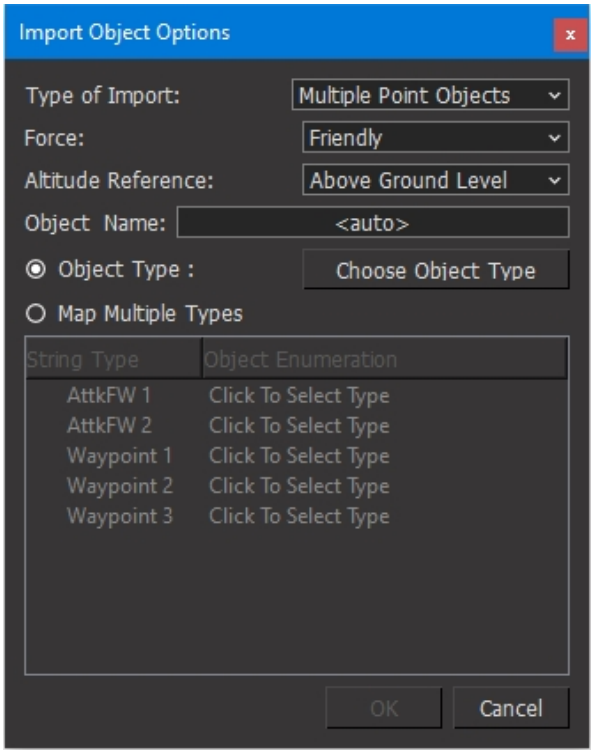
VR-Forces includes some example CSV files in *./userData/csvLaydown*.

> You cannot export simulation objects to a CSV file. To make the objects in a scenario available to other scenarios, try some of the strategies in "2.2. Importing Objects into Scenarios" on page 14.

➤ **To import scenario objects from a CSV file:**

1. Choose **File > Import Scenario Objects**. The Import Scenario File dialog box opens.
2. In the Files of Type box, set the filter to CSV Scenario File (*.csv).
3. Select the CSV file that you want to import.
4. Click Open. The Import Object Options dialog box opens. If you selected a CSV file that has point data, the points are listed in the window

Figure 10. Import Object Options dialog box

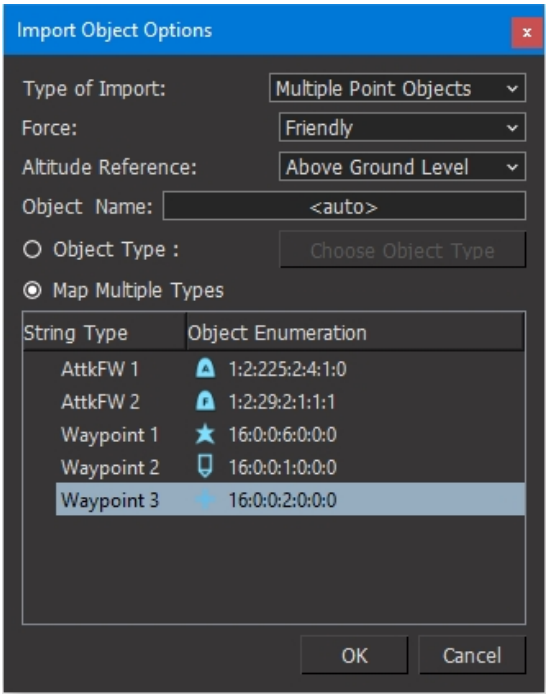


5. In the Type of Import, select the type of data you are importing.
6. Optionally, specify a force. All imported objects are imported with the same force.
7. Optionally, specify the altitude of objects as AGL (above ground level) or ASL (above sea level).
8. Optionally, in the Object Name box specify a name for the object. The default is to use VR-Forces naming conventions to automatically assign a name.
9. If you are importing an area or route:
    a. Select the Object Type option.
    b. Click Choose Object Type. The Entity Type dialog box opens. It lists objects of the type you selected in the Type of Import list. (The layout of the dialog box is similar to the object creation palettes.)
    c. Select the specific type of area or route that you want to use for the imported object. The Entity Type dialog box closes.

    If you are importing a list of waypoints:

    a. Select the Map Multiple Types option.
    b. Click a point in the list. The Object Type dialog box opens.
    c. Select the specific type of point or entity that you want to use for the imported object. The Object Type dialog box closes. The Import Object Options lists the object type enumeration for the object.

Figure 11. Import Object Options dialog box



d. Repeat this process for each object in the list.

10. Click OK.

2.2.4 Importing Airspace Control Orders

An airspace control order (ACO) defines and establishes an airspace for military operations. It is can be part of an air tasking order (ATO) or a separate document. VR-Forces can import ACOs that are separate documents.

VR-Forces includes an example ACO file, *./userData/aco/exampleMontereyACO.aco*. The file's format is documented in *./userData/aco/acoFormattingReadme.txt*.

➤ **To import an ACO:**

1. Choose **File > Import Scenario Objects**. The Import Scenario File dialog box opens.
2. In the Files of Type box, set the filter to ACO Scenario File (*.aco).
3. Select the file that you want to import.
4. Click Open.

2.2.5 Importing Two-Line Element Sets

You can import two-line element set files (TLE) to display orbital objects, such as satellites.

> ⓘ You cannot export simulation objects to a TLE file. To make the objects in a scenario available to other scenarios, try some of the strategies in "2.2. Importing Objects into Scenarios" on page 14.

➤ **To import a TLE:**

1. Choose **File > Import Scenario Objects**. The Import Scenario File dialog box opens.
2. In the Files of Type box, set the filter to TLE Scenario File (*.tle).
3. Select the file that you want to import.
4. Click Open. The Import Two-Line Elements dialog box opens.
5. Click Choose Object Type. The Object Type dialog box opens, showing available space simulation objects from the SMS you selected for the scenario.
6. Select the object you want to use for the imported objects. The Object Type dialog box closes.
7. Click OK.

2.2.6 Loading Object Tags

You can load a CSV file of entity types and tags. The tags are added to the currently loaded SMS of objects in the palettes but are not written back to the SMS. The next time you load the SMS, your custom tags are cleared out. This allows you to temporarily change the tags that are available in the palettes for filtering the list. There is an example file, *./SharedData/##/latest/ModelData/Configuration/modeltags/tags.csv*.

➤ **To load object tags into your scenario:**

1. Choose **File > Load Object Tags**. The Load Object Tags dialog box opens.
2. Navigate to the location of your CSV file and then select it.
3. Click Open.

   The updated tags are now available in your palettes.

2.3. Saving a Scenario

The first time you save a scenario, VR-Forces saves the path of the terrain database and other scenario configuration data to a scenario file (extension *.scn*). (Paths are saved relative to the *./bin64* directory.) This information usually does not change between runs of a scenario. Every time you save a scenario, VR-Forces saves the list of simulation objects and tactical graphics, the plans, and object mapping information to individual files that have the same name as the scenario, but with file extensions that identify the information in the file (*.oob*, *.pln*, *.xtr*, *.omp*, and so on). By default, all these files are saved to a compressed archive with the extension *.scnx*. However, you can save them as individual files, as explained in this procedure.

- If you want to keep the initial state of a scenario (simulation time = 0 and no simulation has taken place), so that you can repeatedly run a simulation from the beginning, save the scenario when you have finished creating it, but before you run it. Then, do not save the scenario again under the same name (unless you specifically want to change the original state of the scenario). (If you forget to save a new scenario, you can rewind to the initial snapshot and save it.)
- If you load an alternate terrain in a GUI when it joins a session, the alternate terrain is not saved as part of the scenario.

When you save a scenario, whether during the course of setting it up or during a simulation, all object state information is saved. Essentially, every save is a checkpoint. When you run a saved simulation, all simulation objects resume whatever they were doing when the scenario was saved.

- If simulation objects are executing scripted tasks, some information regarding the tasks is not saved. For details, see "Limitations for Checkpointing Scripts".
- When you task an entity to move and then start playing the scenario, it generates a path. If you save the scenario with that path already planned, it is saved in the scenario. The next time you play the saved scenario, the entity does not need to calculate a path. This can improve the scenario performance.

Save behavior depends on when you save a scenario:

- If you save a previously saved scenario after it has started (whether it is paused or running), it is not saved to the original scenario file. It is saved to a checkpoint that is named using the convention *scenario_name*_at_time_*time*.scn. This ensures that you cannot inadvertently overwrite the original scenario file. To overwrite the original scenario, you must use Save Scenario As. For more information, see "2.8.1 Saving Checkpoints" on page 38 and "2.3.1 Saving a Previously Saved Scenario" on the next page.
- If you run a new scenario without saving it and save it at some simulation time greater than zero, it is saved using whatever name you provide. The time is not included in the scenario name.

➤ **To save a new scenario:**

1. Choose **File > Save Scenario** or click the Save button () on the File Toolbar. The Save Scenario dialog box opens.
2. Select the directory in which you want to save the scenario.

   On Windows machines, if you create a new folder using the Save Scenario dialog box, do not enter a space at the end of the name. Windows does not recognize the existence of folders with spaces at the end and it can cause problems. If you try to delete it, the system will not allow you to and it may blue screen. Running ScanDisk can solve this problem.

3. Give the scenario a name.
4. By default, scenarios are saved in a compressed archive (**.scnx*). If you want to save a scenario as individual files, in the File of Type list, select MAK Scenario Format (*.scn).
5. Click Save. All scenario files are saved with the new name.

> ⓘ If you save a remapped scenario, you are prompted to save the scenario with the original mappings or with the remappings.

2.3.1 Saving a Previously Saved Scenario

In most computer applications, when you save a file that has been saved previously, the current version of the file simply overwrites the previously saved version. However, when VR-Forces saves a scenario, the save action depends on the status of the scenario. VR-Forces tracks the simulation time of the scenario. When you save a scenario:

- If a version of the scenario exists with the current simulation time, VR-Forces displays a message indicating that a version of the scenario at that simulation time exists and asks if you want to overwrite it.
- If there is no version with the current simulation time, VR-Forces creates a new checkpoint at the current simulation time, using the checkpoint naming convention.

This behavior ensures that if you change a scenario after you have started a simulation, you will never inadvertently overwrite the original version or another checkpoint.

➤ To save a previously saved scenario, choose **File > Save Scenario**.

Saving a Load Balanced Scenario

When you load a load-balanced scenario and do not have all of the required sim. engines available, VR-Forces prompts you to allow it to remap objects to the available sim. engines. Then it loads the scenario.

When you save a scenario that was originally load balanced, VR-Forces prompts you to either save the original mappings or to save the scenario with the sim. engine mappings used to load it in the current session.

If you save the original mappings, all objects that were part of the scenario when it was loaded maintain their previous mappings. Any objects that you created during the current session are mapped to the sim. engines that they were created on in the current session.

2.3.2 Saving a Scenario to a New Name or a Different Format

You can save a scenario to a different name. You can also save it to a different format. That is, you can save a scenario that was originally saved as a compressed archive to individual files or save individual files to a compressed archive.

If you save a scenario to a different format and you use the same name, VR-Forces does not replace the previously saved files with the new format. The two sets of files live side-by-side. Therefore we recommend that you give the newly saved scenario a different name or remove the old files to avoid confusion between different versions of the same scenario.

➤ **To save a scenario to a new name:**

1. Choose **File > Save Scenario As**. The Save Scenario dialog box opens.
2. Select the directory in which you want to save the scenario.
3. Give the scenario a name.
4. If you are changing the format, select an option in the Files of Type list.
5. Click Save. The scenario is saved to the new name.

2.4. Loading a Scenario

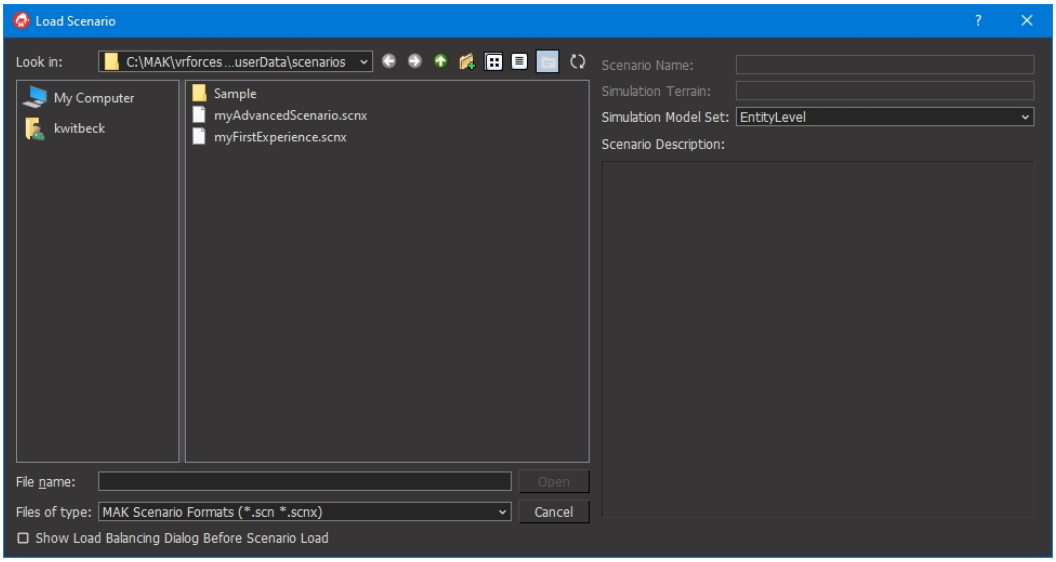
After you have created and saved a scenario, you can load it and run it on any platform supported by the version of VR-Forces in which you created it. Most scenarios are forward compatible. See the *VR-Forces Release Notes* for compatibility issues.

- If you are running multiple GUIs in the same session, when you load a scenario, any terrains that are open on other GUIs close and the terrain for the newly loaded scenario opens.
- When you load a scenario that uses multiple sim. engines, if any of the required sim. engines are not running, VR-Forces prompts you to remap the objects in the scenario. The Remap/Start Simulation Engines dialog box lists the unavailable sim. engines and the available sim. engines. You can specify which sim. engines to map objects to or you can select "map all to" and map all objects to one sim. engine. For more information about remapping sim. engines, see "Working with Multiple Simulation Engines".

➤ **To load a scenario:**

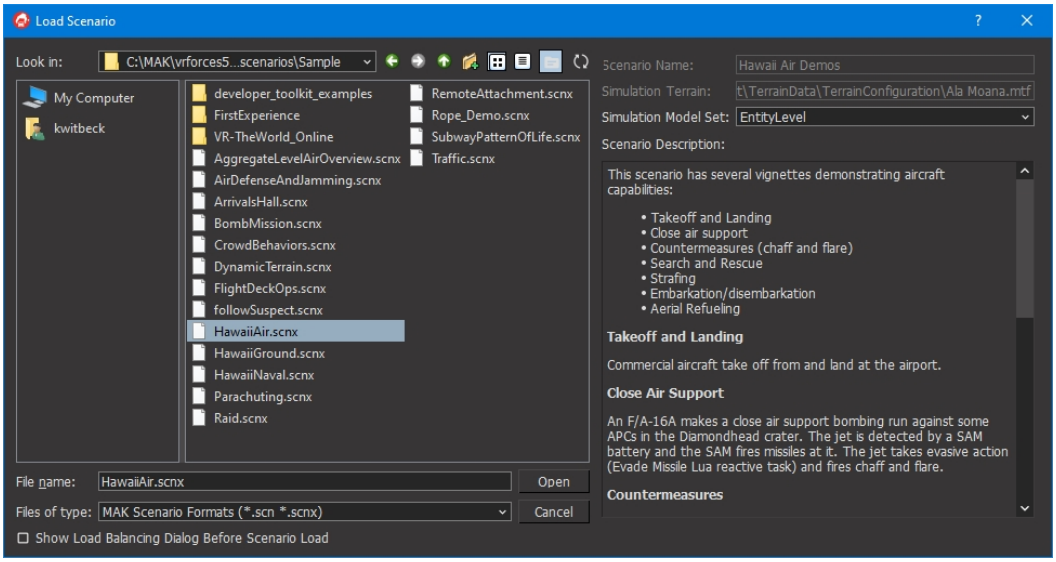
1. Choose **File > Load Scenario** or click the Open Scenario button () on the File Toolbar. The Load Scenario dialog box opens (Figure 12).

Figure 12. Load Scenario dialog box



2. Select a scenario to load. Basic information about the scenario displays (Figure 13). Scenarios are files with the suffix *.scn* or *.scnx*. Sample scenarios are located in the *./userData/scenarios/Sample* directory. (Batch scenarios, described in "2.10. VR-Forces and Batch Mode" on page 43, have the suffix *.bsn*.)

Figure 13. Load Scenario dialog box



3. Optionally, specify whether to allow management of components on remote entities by selecting an option in the Attach Components to Remote Entities On list. For details about configuring components on remote entities, see "Attaching Components to Remote Entities".

4. Optionally, select the Show Load Balancing Dialog Before Scenario Load check box. See "2.5.1 Load Balancing an Existing Scenario" on page 31 for details on the different options.
5. Click Open. If you elected to load balance the scenario, the Sim. Engine Load Distribution dialog box opens. Otherwise, the scenario opens.

As the scenario loads, an indicator near the bottom right of the main window shows what is loading: terrain, insets, navigation, and so on.

2.4.1 Loading a Recently Loaded Scenario

VR-Forces keeps a list of the scenarios you open. You can quickly open a scenario from the list.

➤ To open a recently used scenario, choose **File > Recent Scenarios >** *scenario_name*.

2.4.2 Loading a Scenario from the Command Line

If you start VR-Forces from the command line, you can specify a scenario to load. You can specify the scenario in the sim. engine or GUI. The other executable will load it as soon as it starts up.

To load a scenario when you start the sim. engine, use the `-L` command-line option, for example:

```
vrfSimDIS -s 1 -a 3001 -L "../userData/scenarios/myScenario.scnx"
```

To load a scenario when you start the GUI, use the `--scenarioFile` command-line option, for example:

```
vrfGui --scenarioFile "../userData/scenarios/myScenario.scnx" --dis -
s 1 -a 3000
```

2.4.3 Editing Scenario Properties

When you create a scenario, you specify a variety of parameters (or accept default values) and, optionally, specify a name and description for the scenario. If a scenario has a description, when you load the scenario, the Scenario Information dialog box opens automatically and displays the scenario properties. (You can disable this behavior.) You cannot edit properties in the Scenario Information dialog box.

You can display scenario properties at any time while a scenario is loaded. You can edit the scenario properties. These changes take place immediately:

- Scenario description.
- Scenario name.
- Time multiplier.
- Create global environment.
- Create global dynamic terrain processor.
- Scenario end time.

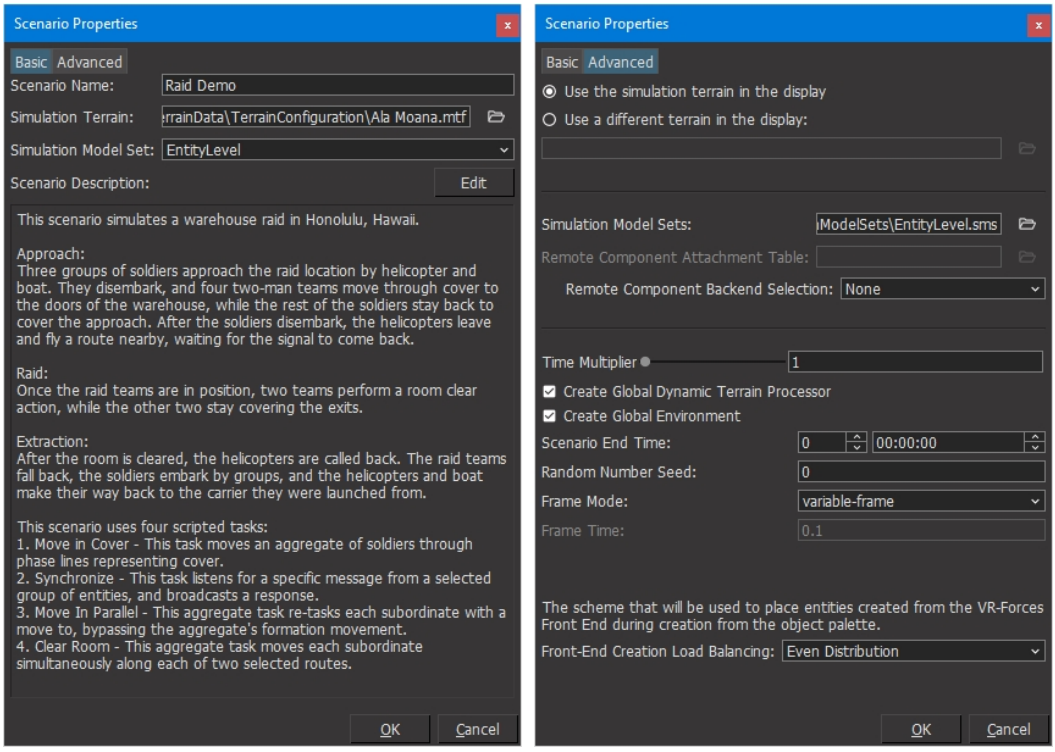
The following changes do not take effect until you save the scenario and reload it. If you edit one of these properties and have not yet saved the scenario, if you open the dialog box again, an icon displays to alert you to save the scenario.

- Simulation terrain.
- Use a different terrain in the display.
- Simulation model sets.
- Component attachment table.
- Random number seed.
- Frame mode.
- Frame time.

➤ **To edit scenario properties:**

1. Choose **File > Scenario Properties**. The Scenario Properties dialog box opens (Figure 14).

Figure 14. Scenario Properties dialog box



2. Edit properties as desired. (For descriptions of the properties, see Table 1 on page 6 and Table 2 on page 7.)

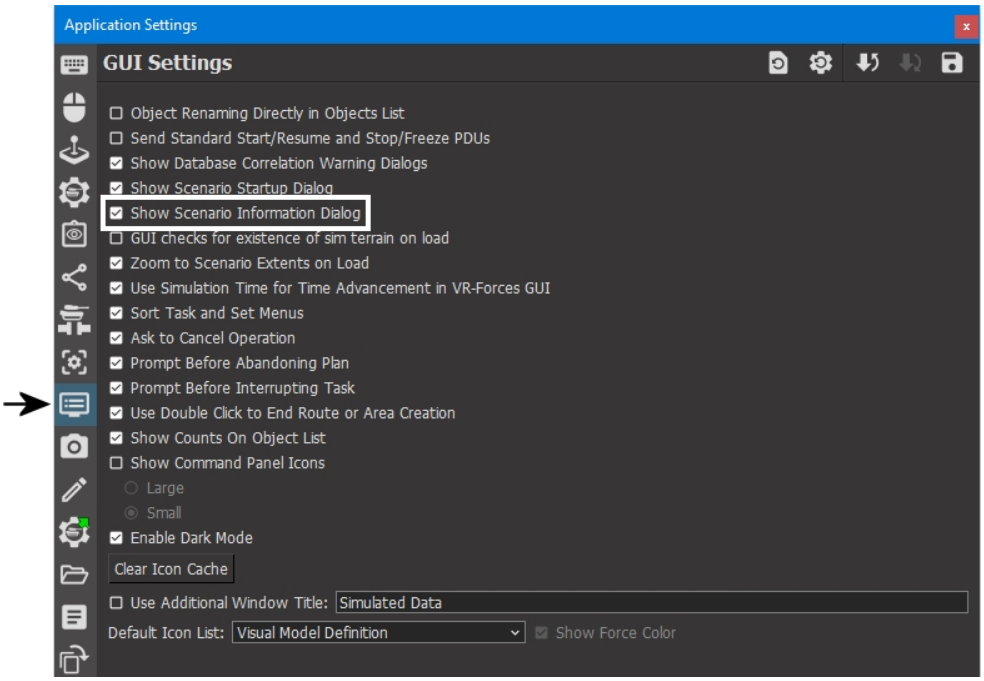
Enabling and Disabling Display of the Scenario Description

By default, the scenario description displays when you open a scenario. You can turn it off in the Scenario Information window and you can enable and disable its display in the Application Settings dialog box.

➤ **To disable display of scenario information when a scenario is loaded, use one of the following methods:**

- When the Scenario Information window opens, select the Never Show Again check box.
- On the Application Settings dialog box, GUI Settings page, clear the Show Scenario Information Dialog check box (Figure 15).

Figure 15. GUI Settings page



➤ To enable the Scenario Information dialog box, on the Application Settings dialog box, GUI Settings page, select the Show Scenario Information Dialog check box.

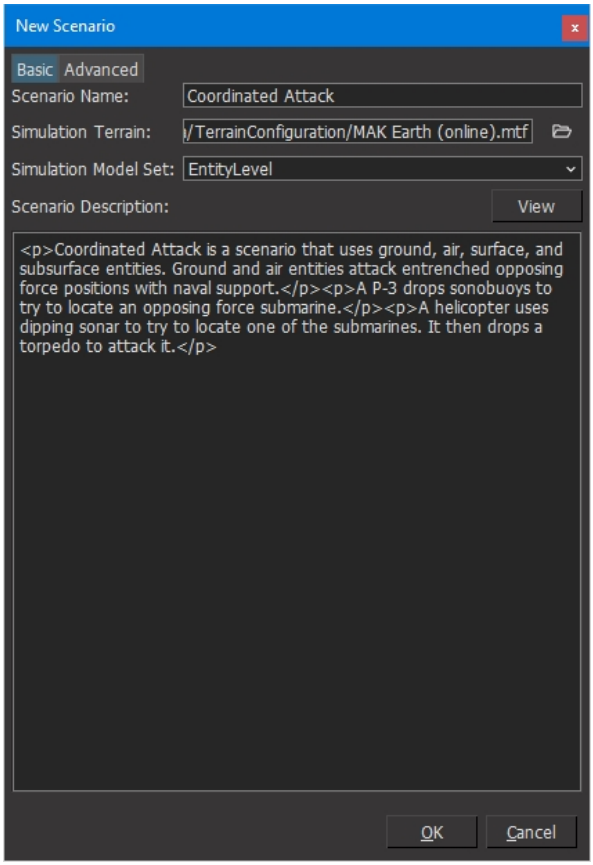
2.4.4 Editing the Scenario Description

You can edit a scenario's description in the Scenario Properties dialog box. If you want to format it, you can use HTML tags.

➤ **To edit a scenario's description:**

1. Choose **File > Scenario Properties**. The Scenario Properties dialog box opens (Figure 16).

Figure 16. Scenario Properties



2. Edit the text.
3. To see how the text will be displayed, click View.
4. Click OK.

2.4.5 Navigating Scenario Directories

When you use the Load Scenario dialog box, the default ("home") directory is *./userData/scenarios*. You can change the default directory using the --userScenarioDirectory *directory* command-line option.

If you open a scenario from another directory, the next time in that session that you open the Load Scenario dialog box, it defaults to that directory. If you want to jump to the home directory, click the Return to Default Scenario Directory button (🔄).

2.4.6 Sample Scenarios

VR-Forces includes a number of sample scenarios. The *VR-Forces First Experience Guide* has several tutorials for scenario creation.

2.5. Load Balancing Scenarios

The preferred method for optimizing the performance of scenarios is to manage multiprocessors usage as described in "Optimizing Multiprocessor Usage for Simulation Engine".

If configuring VR-Forces to take advantage of your specific hardware setup does not improve the performance of a scenario, you can try spreading simulation of objects across multiple sim. engines and machines. This is called load balancing. VR-Forces supports even distribution (done automatically by VR-Forces) and manual load balancing.

You can load balance a scenario when you create it. You can also load balance a scenario after you create it.

➤ **To load-balance a new scenario:**

1. Start multiple sim. engines on separate machines as described in "Starting VR-Forces from the VR-Forces Launcher" and "Starting Independent VR-Forces Executables" .
2. Create the new scenario, as described in "2.1. Creating a Scenario" on page 4.
3. Before you create an object, select the sim. engine on which you want it to be simulated, as described in "Selecting the Object to Create on an Object Palette" and "Selecting the Object to Create on the Create Menu".

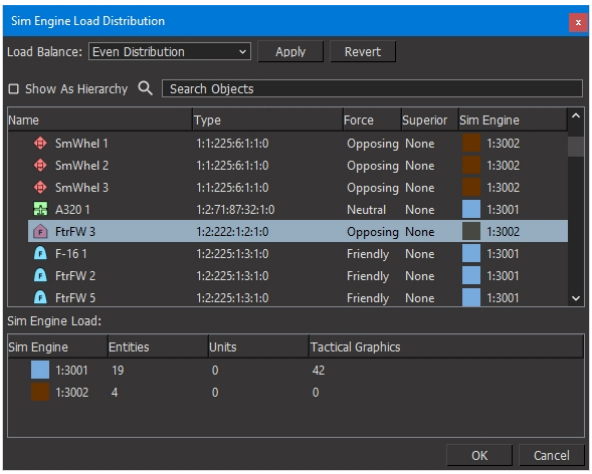
2.5.1 Load Balancing an Existing Scenario

You can change the object mappings for a scenario (load balancing) when you load it. You cannot change mapping after the scenario loads.

➤ **To load-balance an existing scenario:**

1. Start as many sim. engines as you want to use in the scenario.
2. Choose **File > Load Scenario**. The Load Scenario dialog box opens.
3. Select the Show Load Balancing Dialog Before Scenario Load check box.
4. Click Open. The Sim. Engine Load Distribution dialog box opens (Figure 17).

Figure 17. Sim. Engine Load Distribution



5. Choose a load balance method:
   - Even Distribution. VR-Forces automatically distributes objects across the available sim. engines.
   - Unit Based Distribution. VR-Forces ensures that aggregates and subordinates are kept together on the same sim. engine.
   - Rule Based Distribution. Choose an existing rule set or create a new one. See "Rule Based Load Balancing" below to learn about creating a rule set.
6. After selecting the load balance distribution method, click Apply. The Sim. Engine column of the list shows which sim. engine each simulation object will load on, and the Sim. Engine Load shows the totals.
7. Click OK. The scenario loads. If you open the Information dialog box for an object, the State Data page shows the Simulating Sim. Engine.
8. To preserve the new object mappings, save the scenario. When you save the scenario, you are prompted to save the original mappings. Click No to save the new load balanced mappings.

ⓘ You can also right-click on an entity and then assign it to a specific sim. engine.

Rule Based Load Balancing

You can create a new rule set for load balancing. This allows you to specify where simulation objects should load based on object type or name.

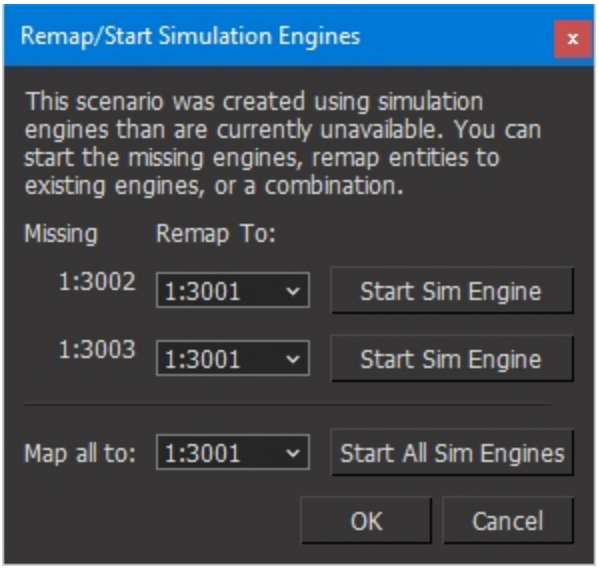
➤ **To create a rule set:**

1. Enter a Title for the rule set.
2. Click the down arrow next to the plus sign and then choose either Entity Type or Object Name.
   - When you choose Entity Type, it adds a row. The default is all entities. Click Edit (✎) and then use the Edit Entity Type dialog box to specify particular simulation objects.
   - When you choose Object Name, the Object Name dialog box opens. Enter a specific simulation object's name or create a regular expression.
3. Add any other rules that you want.
4. When you are finished, click OK.

2.5.2 Loading a Load-Balanced Scenario

When you load-balance a scenario, it saves the mapping between objects and simulation engines in the scenario's object mapping file. When you load the scenario, VR-Forces looks for the listed simulation engines based on their application numbers. If it does not find the required simulation engines, it prompts you for instructions on how to proceed (Figure 18). You can start the missing simulation engines, remap objects to available simulation engines, or if you are using more than two simulation engines, do a combination of both.

Figure 18. Remap/Start Simulation Engines dialog box



If you start a simulation engine as part of the process of loading a scenario, it is started in combined mode and will automatically be closed when you shut down the GUI.

➤ **To load a scenario that uses multiple simulation engines:**

1. Optionally, start all the required simulation engines and at least one GUI. One simulation engine can be started in combined mode with the GUI.
2. Load the scenario. If the scenario cannot find all of the required simulation engines, it prompts you to start them or remap objects to the available sim. engines (Figure 18). For each unavailable simulation engine, you can remap objects to an available simulation engine or start the simulation engine:
   - To start a simulation engine, click the Start Sim. Engine button for the Missing simulation engine.

     If there is more than one unavailable simulation engine, as you start them individually, the dialog box redisplays to show only those that are still unavailable. If you start all of the required simulation engines, the dialog box closes and the scenario loads.
   - To start all missing simulation engines, click the Start All Sim. Engines button.
   - To remap objects for a missing simulation engine, select an available simulation engine from the Remap To list for the Missing simulation engine.
   - To remap all objects to an available simulation engine, select an available simulation engine from the Map All To list.

     When you select a simulation engine in the Map All To list, it sets the same simulation engine in the lists for each specific missing simulation engine. Put another way, when you click OK to close the dialog box, VR-Forces uses the

remapping specified for each missing simulation engine. It ignores the setting for Map All To. Map All To is just a quick way to set all missing simulation engines to the same available one.

3. If you decided to remap objects to an available simulation engine, click OK. VR-Forcess remaps the objects and the scenario loads.

2.6. Running a Scenario

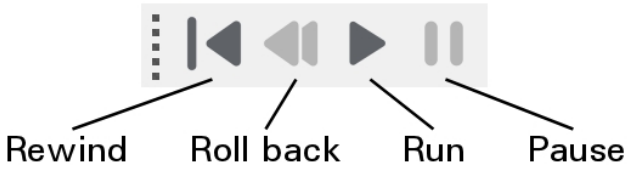
If you want to use VR-Forces to simply observe a remote simulation, you can start it, open a terrain database, and wait for it to pick up messages from the network. However, you will probably use VR-Forces to run scenarios that have locally simulated simulation objects.

When you load a scenario, it opens in the paused state unless you start with the `-L` and `-r` sim. engine command line options.

> When you run a scenario, VR-Forces saves a snapshot of the state of the scenario in memory. However, it does not save the scenario to disk. We recommend that you save new or edited scenarios before you run them. If you choose not to, it is your responsibility to remember to rewind to the initial scenario snapshot if you want to save the starting point of the scenario.

➤ To run a scenario, choose **Simulation > Run Scenario**, or click the Run button (▶) on the Simulation Control Toolbar (Figure 19).

Figure 19. Simulation Control Toolbar



2.6.1 Changing the Simulation Speed

You can set the speed of a simulation to run faster or slower than real-time. If you are running VR-Forces with time management, the Simulation Time Scale Toolbar has no effect on simulation speed.

> If you increase the speed at which a scenario runs, the frame rate is reduced and performance of models may degrade.

➤ To change the speed of a simulation, click an option on the Simulation Time Scale Toolbar (Figure 20).

Figure 20. Simulation Time Scale Toolbar

Configuring the Simulation Time Scale Toolbar

By default, the range for the Simulation Time Scale Toolbar is 1 to 15. You can change the range. To change the range, edit these lines in *./appData/settings/vrfGui/default_GuiSettings.grsx*:

```
<myTimescaleLow>1</myTimescaleLow>
<myTimescaleHigh>15</myTimescaleHigh>
```

To change the options on the time scale buttons, edit the myTimescaleButtons elements:

```
<myTimescaleButtons>1;2;5;10;15</myTimescaleButtons>
```

If you want the Simulation Time Scale Toolbar to use a slider as in VR-Forces 4.6.1 and earlier, instead of discrete buttons, set the myTimescaleButtonMode to 0:

```
<myTimescaleButtonMode>0</myTimescaleButtonMode>
```

Changing Simulation Parameters when Increasing the Time Scale

When you increase the time scale in scenarios that have many entities, particularly if they are fast moving entities such as aircraft, simulation quality may degrade. VR-Forces lets you change the dead-reckoning algorithm, frame mode, and frame time automatically to settings that may improve simulation quality.

These settings are configured in *./appData/settings/vrfSim/fastForwardSettings.mtl*. The comments in the file explain the acceptable values for each parameter. This is an example configuration:

```
(fast-forward-settings
  (fast-forward-entry
    (play-speed 3)
    (settings
      (dead-reckon-algorithm Static)
      (frame-mode 2)            ;; fixed frame
      (frame-time 1.0)
    )
  )
  (fast-forward-entry
    (play-speed 6)
    (settings
      (dead-reckon-algorithm Static)
      (frame-mode 2)            ;; fixed frame
      (frame-time 2.0)
    )
  )
)
```

Each fast-forward entry specifies a time scale speed at which it takes effect. When you change the time scale, VR-Forces finds the highest play-speed specified in this file that is less than the current speed and updates the settings accordingly. If the current speed is less than the lowest value, default settings are used.

2.6.2 Pausing a Scenario

When you pause a simulation, local simulation objects stop executing plans and tasks. Display of remote entity activity continues as it is received over the network.

➤ To pause a simulation, choose **Simulation > Pause Scenario**, or click the Pause button ( ) on the Simulation Control Toolbar (Figure 20 on page 34).

2.6.3 Rewinding a Scenario

If you are running a scenario and want to start it over again, you can rewind it. You can rewind a scenario to its initial state, which is captured as a snapshot, and you can roll back to any snapshot or checkpoint (if snapshots and checkpoints are enabled).

When you rewind a scenario, all sim. engines are set to the paused state.

> ⓘ An initial snapshot is taken of a scenario when you play it even if periodic snapshots is not enabled. The initial snapshot is used for rewinding.

➤ To rewind a scenario, choose **Simulation > Rewind to Scenario Start**, or click the Rewind Scenario to the Beginning button ( ) on the Simulation Control Toolbar (Figure 20 on page 34).

If you do not pause a scenario before you rewind it, it starts playing again immediately.

Rolling Back to a Snapshot or Checkpoint

You can roll back to any of the available snapshots or checkpoints. When you roll back to a snapshot, VR-Forces loads the scenario as it existed at the point of that snapshot. When you run the snapshot, all snapshots from a later time than the one you rolled back to are deleted. Older snapshots are kept in memory. New snapshots are taken starting from the loaded snapshot.

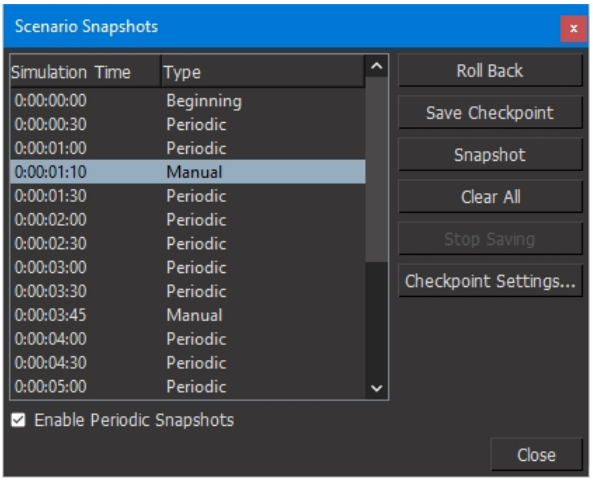
Once snapshots are deleted based on the maximum number of snapshots being reached, you cannot roll back to them. If you have saved periodic checkpoints, you can load them using the Load Scenario command on the File menu.

For details about checkpoints and snapshots, see "2.8. Checkpoints and Snapshots" on page 38.

➤ **To roll back to a snapshot or checkpoint:**

1. Choose **File > Scenario Snapshots**, or on the Checkpoint Settings page, click Show Snapshots. The Scenario Snapshots dialog box opens (Figure 21).
2. Select the snapshot or checkpoint you want to roll back to.

Figure 21. Scenario Snapshots dialog box

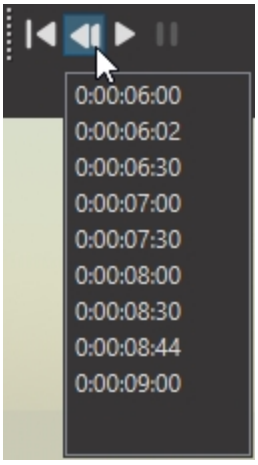


3. Click Roll Back.

Using the Snapshot and Checkpoint List

You can also display a list of snapshots and checkpoints by clicking the Snapshots button () on the Simulation Control Toolbar (Figure 22).

Figure 22. Snapshot list



2.6.4 Closing a Scenario

When you close a scenario, the terrain stays loaded. Checkpoints and snapshots remain in memory until a new scenario is loaded. Therefore, if you click Rewind after closing a scenario, the scenario opens again.

➤ To close a scenario, choose **File > Close Scenario**, or click the Close Scenario button () on the File Toolbar.

2.7. Reinitializing a Scenario

Sometimes when you load a scenario that was created in a previous version of VR-Forces, entities do not behave as they did previously due to changes to systems. Usually, the way to fix this problem is to delete the objects, create new instances of the object type, and reconstruct their plans or tasks. Unfortunately, when you create a new object, it is assigned a unique ID. Since the unique ID for the newly created objects will be different from those of the deleted objects, any plans or tasks that referenced the deleted objects will not work, because they depend on the unique ID to identify the object that they are referencing. If you need to recreate an object and maintain its unique ID, you can reinitialize a scenario.

When you reinitialize a scenario, VR-Forces deletes all simulation objects and creates them again with the same unique IDs they originally had. This preserves task and plan relationships. However, this process does not preserve the entire state of entities. If you reinitialize a checkpointed scenario, it may lose information that is saved as part of a system. Plans are restarted, so the current task might be cleared. So there may be cases where reinitializing causes more problems than it solves.

Given this consideration, you should only use this option for scenarios that are at zero simulation time and only if you are sure that it is the only option. You may want to save a backup of the scenario in case you want to revert to it and try fixing it by manually deleting the offending object and adjusting other aspects of the scenario as needed.

➤ **To reinitialize a scenario:**

1. Optionally, save a backup of the scenario in case you want to revert to the original state.
2. Choose **File > Reinitialize Scenario**.

2.8. Checkpoints and Snapshots

As noted in "2.3. Saving a Scenario" on page 22, each time you save a scenario, you are essentially creating a checkpoint. A checkpoint saves the state of a scenario to disk and allows you to run the scenario starting from the point at which it was saved. Since a checkpoint is saved to disk, you can return to it at any time. However, the time spent saving to disk may briefly interrupt scenario progress.

Sometimes VR-Forces users want to be able to quickly return to an earlier point in a simulation without the processing overhead required to save checkpoints. As an alternative or complement to checkpoints, you can save snapshots.

Snapshots save the state of a scenario to memory. You can quickly roll back to any saved snapshot and start running the scenario from that point. When you roll back to a snapshot, all snapshots that were taken at a later time than the rollback point are discarded and new snapshots are saved. When you load a scenario, any snapshots in memory are discarded.

You can configure the frequency with which snapshots are saved and how many to keep in memory at a time. You can save snapshots as checkpoints.

2.8.1 Saving Checkpoints

To save checkpoints, you can :

- Save the scenario.
- Save a snapshot as a checkpoint.
- Enable periodic checkpointing.

Checkpoints are saved to the same directory as the base scenario unless you specify otherwise. If periodic checkpoints are enabled, the Status bar displays the time until the next checkpoint.

> If simulation objects are executing scripted tasks, some information regarding the tasks is not saved. For details, see "Limitations for Checkpointing Scripts".

When you run a checkpoint, simulation objects resume the tasks that they were executing at the time the checkpoint was saved. If detonations were scheduled or if missiles were in flight, they resume their former state.

Saving an Individual Checkpoint

Whenever you save a scenario, it creates a checkpoint.

➤ To save an individual checkpoint, choose **File > Save Scenario**.

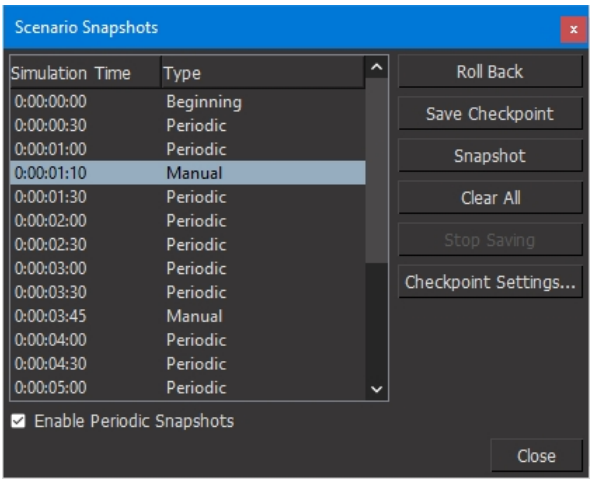
Saving Snapshots as Checkpoints

If you enable periodic snapshots, you can save the snapshots as checkpoints.

➤ **To save a snapshot as a checkpoint:**

1. Choose **File > Scenario Snapshots**. The Scenario Snapshots dialog box opens (Figure 23). It lists all of the snapshots available.

Figure 23. Scenario Snapshots dialog box



2. Select the snapshots that you want to save as checkpoints.
3. Click Save Checkpoint. The snapshots are saved to checkpoints.

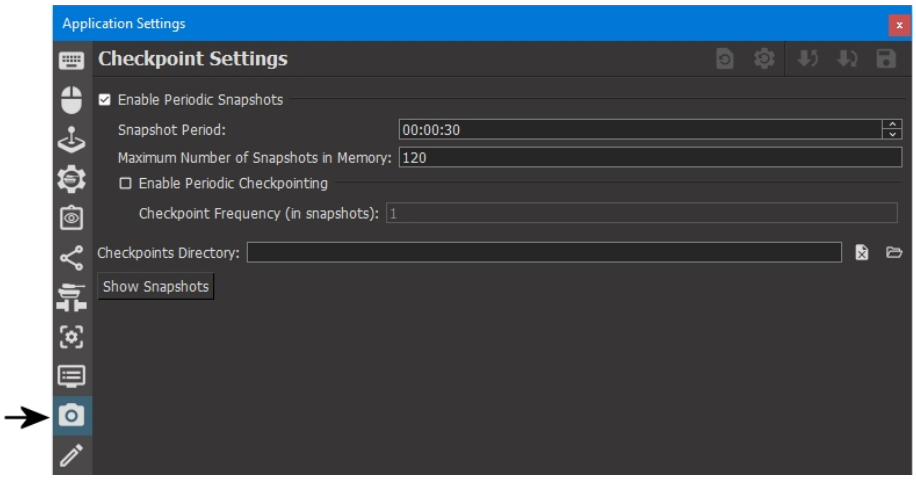
Scheduling Periodic Checkpoints

You can configure VR-Forces to save checkpoints at regular intervals. When VR-Forces saves a checkpoint, it is actually saving the snapshot taken at the checkpointing interval. Therefore, to schedule periodic checkpoints, you must enable periodic snapshots.

➤ **To schedule checkpoints:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Checkpoint Settings page (Figure 24).

Figure 24. Checkpoint Settings page



3. Select the Enable Periodic Snapshots check box.
4. In the Snapshot Period box, specify how frequently VR-Forces should take snapshots.
5. Select the Enable Periodic Checkpointing check box.
6. In the Checkpoint Frequency box, specify how frequently to save checkpoints as a function of the number of snapshots taken. For example, 4 means save every fourth snapshot as a checkpoint. If you save snapshots every 30 seconds, then you would save a checkpoint every two minutes.
7. Optionally, specify the directory in which to save the checkpoints. If you leave this box blank, they are saved to the directory where the base scenario is located.

Disabling Periodic Checkpointing

➤ **To disable periodic checkpointing:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Checkpoint Settings page (Figure 24).
3. Clear the Enable Periodic Checkpointing check box.

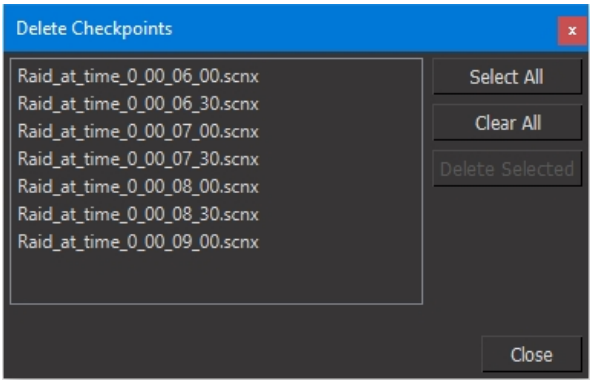
2.8.2 Deleting Checkpoints

You can delete checkpoints manually in a file browser, like you can delete any other files on your computer. You can also easily delete them from the GUI.

➤ **To delete checkpoints:**

1. Choose **File > Delete Scenario Checkpoints**. The Delete Checkpoints dialog box opens (Figure 25).

Figure 25. Delete Checkpoints dialog box



2. Select the checkpoints that you want to delete.
3. Click Delete Selected.

2.8.3 Creating Snapshots

You can create snapshots manually and you can specify that they be created automatically at a specified interval.

Creating Snapshots Manually

➤ **To create a snapshot:**

1. Choose **File > Scenario Snapshots**, or on the Checkpoint Settings page, click Show Snapshots. The Scenario Snapshots dialog box opens (Figure 23 on page 39).
2. Click Snapshot.

> You can also create a snapshot by clicking the Snapshot button ( ) on the Scenario Toolbar.

Enabling Periodic Snapshots

When you enable periodic snapshots, you specify how frequently to take them, in simulation time, and the maximum number to store in memory at a time. If the number of snapshots reaches the maximum, VR-Forces drops the oldest snapshot and adds the newest.

➤ **To enable periodic snapshots:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Checkpoint Settings page (Figure 24 on the previous page).
3. Select the Enable Periodic Snapshots check box.

4. In the Snapshot Period box, specify how frequently VR-Forces should take snapshots.
5. In the Maximum Number of Snapshots in Memory box, specify how many snapshots to keep in memory.

2.8.4 Clearing the Snapshot List

Snapshots are stored in memory and listed in the Scenario Snapshots dialog box. They are cleared automatically when you load a scenario or rewind a scenario and run it again. If you roll back to a snapshot and run the scenario, all snapshots later than the rollback point are deleted. You can also clear snapshots manually.

When you clear the snapshot list, VR-Forces keeps the initial snapshot so that you can still rewind the scenario.

➤ **To clear the snapshot list:**

1. Choose **File > Scenario Snapshots** or, on the Checkpoint Settings page, click Show Snapshots. The Scenario Snapshots dialog box opens (Figure 23 on page 39).
2. Click Clear All.

2.9. Pausing a Scenario Automatically

You can specify a scenario end time (in simulation time). As the scenario progresses, the amount of time remaining is displayed in the Simulation Time Remaining Toolbar (Figure 26). When the scenario reaches this time, it stops. You can resume the scenario by clicking the Play button.

Figure 26. Simulation Time Remaining Toolbar

You can set the scenario end time when you create a scenario or after you have loaded it.

2.9.1 Setting the Scenario End Time for a New Scenario

The default scenario end time for new scenarios is 0 – do not pause the scenario.

➤ **To set the scenario end time for a new scenario:**

1. Create a new scenario as described in "2.1. Creating a Scenario" on page 4.
2. In the New Scenario dialog box, select the Advanced tab.
3. In the Scenario End Time boxes, specify an end time in days, hours, minutes, and seconds.
4. Continue the scenario creation procedure.

2.9.2 Setting the Scenario End Time for an Open Scenario

You can set the scenario end time for a scenario after you open it. If you set a time that is sooner than the current elapsed time, the scenario stops immediately.

➤ **To set the scenario end time for an open scenario:**

1. Choose **File > Scenario Properties**. The Scenario Properties dialog box opens.
2. Select the Advanced tab
3. In the Scenario End Time boxes, specify an end time in days, hours, minutes, and seconds.
4. Click OK.
5. Optionally, save the scenario. Remember that if the scenario is already running or has advanced and then paused, the end time is saved to a checkpoint, not the original scenario.

2.10. VR-Forces and Batch Mode

Batch mode allows you to run one or more scenarios multiple times, without direct action through the graphical user interface. A scenario run in batch mode runs with the parameters in the scenario file, except that you can override the random number seed.

Batch mode is read-only. You cannot save a scenario, create simulation objects or objects, pause the scenario, or otherwise change it, while a batch is running. However, you can view the Objects List Panel, plan windows, and open informational dialog boxes. You can close a scenario that is running in batch mode and resume control of VR-Forces.

2.10.1 The Batch File

A batch mode file contains the information that VR-Forces needs to run scenarios in batch mode. Batch mode offers these options:

- Run a particular scenario multiple times.
- Run a series of different scenarios (batches), each of which can be run multiple times.
- Run from the command line using one sim. engine. If a scenario was created using multiple sim. engines, you cannot run it in batch mode using this method.
- Run from the VR-Forces GUI using one or more sim. engines.
- Generate Logger Control messages to have the MAK Logger record each simulation.

For each batch (one scenario run one or more times) within the batch file, you can:

- Specify the length of time the simulation is to run (in real time or simulation time).
- Specify the object parameter database to use.
- Specify that the random number seed used be the value specified in the scenario file, a value specified in the batch file, or be computer generated.

A batch file has the file extension *.bsn*.

2.10.2 Creating a Batch File

To run VR-Forces in batch mode, you create a batch file (*.bsn*). VR-Forces supplies a sample batch file. Use it as a model for creating your own batch files.

➤ **To create a batch file:**

1. Create scenarios as you would any scenario. (For details, see "2.1. Creating a Scenario" on page 4.)
2. In a text editor, open the sample batch file provided with VR-Forces.
3. Save the batch file under a new name, with the file extension *.bsn*.
4. Replace the sample values with your own, as described in "2.10.3 Editing a Batch File" below.
5. Save the new batch file.

2.10.3 Editing a Batch File

A batch file is written in MAK Technologies Lisp format (MTL). (For information about the MTL format, see "MAK Technologies Lisp (MTL) Files".) A batch file starts with parameters that affect the entire batch. Then it has a batch list, which contains one or more batches. Each batch specifies a scenario, the number of times it is to be run, and other parameters. Table 3 describes the parameters.

This is an example of a batch file:

```
(scenario-batch
  (use-logger-control True)
  (logger-files-path "..\logger_tapes") ;; path is relative to
       Logger executable.
  (batch-list
    (batch
      (number-of-runs  2)
      (random-number-seed 4)
      (scenario-filename  "..\data\scenarios\Raid.scnx")
      (sms-filename
        "..\data\simulationModelSets\EntityLevel.sms")
      (simulation-run-duration 0 : 0 : 0 : 30.000000)
       ;; simulation time
    )
    (batch
      (number-of-runs  3)
      (scenario-filename
        "..\data\scenarios\Apache.scnx")
        (sms-filename
        "..\data\simulationModelSets\EntityLevel.sms")
      (run-duration 0 : 0 : 0 : 20.000000) ;; real time
    )
  )
)
```

Table 3. Batch file parameters

Parameter	Description
Scenario-batch parameters	
use-logger-control	Specifies whether to generate Logger Control messages. Options: • False – Do not generate messages. • True – Generate messages to create Logger files for each run of each batch.
logger-files-path	If **use-logger-control** is True, specifies the directory in which to store Logger files. The path can be absolute or relative to the Logger executable.
Batch-list parameters	
number-of-runs	Specifies how many times to run the scenario in this batch run entry.
random-number-seed	Optional. Overrides the random number seed in the scenario file. It specifies a random number seed: • -1 – use a value generated by the computer. Values are different for each run of this batch. • Any non-negative integer – use the number specified for each run of the batch.
scenario-filename	Specifies the scenario to be run in this batch run entry. The scenario can be specified as an absolute path or a path relative to the directory in which the executable is located.
sms-filename	Optional. Overrides the simulation model set specified by the scenario and specifies the simulation model set file to use.
simulation-run-duration	Specifies the duration in simulation time that the simulation will run. It is mutually exclusive with **run-duration**. If both are specified, VR-Forces uses **simulation-run-duration**. ⚠ Specify hours, minutes, and days as integers, for example 15. Specify seconds as real numbers, for example, 15.0.
run-duration	Specifies the duration in real time that the simulation will run. For example, if a scenario has a time scale set to 2 (twice real time), and a run-duration of 10 minutes, 20 minutes of simulation time will elapse before the batch stops running this scenario. It is mutually exclusive with **simulation-run-duration**. ⚠ Specify hours, minutes, and days as integers, for example 15. Specify seconds as real numbers, for example, 15.0.

2.10.4 Running VR-Forces in Batch Mode

Running in Batch Mode in the GUI

➤ **To run VR-Forces in batch mode in the GUI:**

1. Start VR-Forces. If any of the scenarios you plan to run in batch mode require multiple sim. engines, start the sim. engines with the required site and application IDs.
2. If you are going to generate Logger control messages, start the MAK Data Logger. (Make sure to use the same protocol as you are using for VR-Forces.)
3. Choose **File > Load Batch File**. The Load Batch File dialog box opens.
4. Select the batch file you want to run.
5. Click Open.

VR-Forces begins running the first scenario in the batch file. When VR-Forces finishes running the batch file, it closes the final scenario.

ⓘ
- When VR-Forces runs a batch file, if it cannot load a scenario specified in the file, it skips the scenario and goes to the next batch entry in the file.
- Automatic remapping is always enabled for batch mode.

Running in Batch Mode from the Command Line

If your scenarios only need one sim. engine, you can run a batch file from the command line.

➤ To run a batch from the command line, use the `-B` parameter, with the sim. engine in separate mode, for example:

```
vrfSimDIS -s 1 -a 3001
  -B "../userData/scenarios/Sample/sampleScenarioBatch.bsn"
```

The scenario begins to run automatically.

2.10.5 Recording Batch Scenarios

You can generate Logger Control messages and save scenario output to a Logger file. You must start the Logger before you run the batch scenario.

If you generate Logger Control messages, each simulation is saved individually by the Logger, with a filename in the format:

```
<batch_scenario_name>_<scenario_run_number>_<random_number_seed>.lgr
```

You can specify the directory in which you want the Logger file to be saved.

The MAK Data Logger does not record batch file runs if a previous recording exists. By default, the Logger does not allow you to overwrite existing files using the remote control API. So, once you record a batch run, you cannot record a subsequent run of the same batch. You can work around this problem by editing the *lgrConfig.xml* file to permit overwriting of Logger files. Make the following edits in the configuration file for each protocol for which you want to record batch runs:

Add these lines to the <defaults> section:

```
<var name="processRemoteControl" type="bool" value="true"/>
<var name="recordRemoteControl" type="bool" value="true"/>
<var name="allowDeleteRemoteControl" type="bool" value="true"/>
```

Add these lines to the <commands name="startup"> section:

```
<command domain="Simulation" protocol="DIS"
command="SetRemoteControlSettings">
  <param name="process" var="processRemoteControl"></param>
  <param name="record" var="recordRemoteControl"></param>
  <param name="allowDelete" var="allowDeleteRemoteControl"></param>
</command>
```

Use the appropriate value for the protocol attribute. For DIS, set `listenWhenIdle` to false:

```
<var name="listenWhenIdle" type="bool" value="false"/>
```

2.11. Sending Run and Pause Messages to Simulation Participants

By default, when you run or pause a scenario, VR-Forces sends scenario control messages that are meaningful only to VR-Forces applications that are part of the current session. Sometimes VR-Forces users want all of the participants in a simulation, which may include non-VR-Forces applications, to know when VR-Forces runs or pauses a scenario. You can achieve this by telling VR-Forces to send standard Start/Resume and Stop/Freeze messages instead of the VR-Forces-specific messages. All of the applications that are listening for them will receive them and can respond as needed.

> ⓘ If you are running multiple VR-Forces sessions and you use Start/Freeze messages, all of the sessions receive them, which may defeat the purpose of having separate sessions.

➤ **To enable use of Start/Freeze messages:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the GUI Settings page (Figure 15 on page 29).
3. Select the Send Standard Start/Resume and Stop/Freeze PDUs check box.

2.12. Recording VR-Forces Simulations with the MAK Data Logger

You can use the MAK Logger to record VR-Forces simulations and then play them back for review. If you want to view a Logger file in VR-Forces (as opposed to some other MAK viewer, such as VR-Vantage), note the following considerations:

- Do not load the original scenario in VR-Forces while you view the recording. Just load the terrain database that the simulation requires.

- If you are using HLA 1516, do not record a simulation and then immediately play it back. Exit the applications and start a new federation. Trying to play back the file in the federation from which it was recorded can result in name reservation conflicts and applications may crash.

If a scenario includes plans that have view control messages, the Logger records them. When you play the recording, VR-Forces and other VR-Vantage applications will change view modes in response to them. This feature lets you easily create demonstrations that change views.

3. Scenario Events

Scenario events let you provide non-simulated input to scenario participants.

3.1. Introduction to Scenario Events

A scenario event represents an incident or condition that exercise participants can respond to, for example a political event, a natural disaster, or some other novel event that might require a participant to respond. VR-Forces does not simulate scenario events. Objects in a simulation do not interact with scenario events and scenario events do not directly affect simulation objects (plans can include conditional statements that are contingent on scenario events). Scenario events are placeholders that allow you to provide a richer simulation experience.

Events can include rich text, audio files (WAV), video (AVI) and raster files (JPG, PNG, SVG, or BMP). Use of these media allow you to expand the scope and quality of a training experience without needing to build a complicated simulation to support it. The flexibility of scenario events allows you to use them in any way that adds value to an exercise, for example, mission briefings, and video teaching points.

Unless you disable it, when an event is triggered, VR-Forces displays a dialog box that displays the event information. If a scenario has events that are triggered by simulation time or time of day, the Next Scheduled Event Toolbar displays the amount of time until the next event (Figure 27).

Figure 27. Next Scheduled Event Toolbar

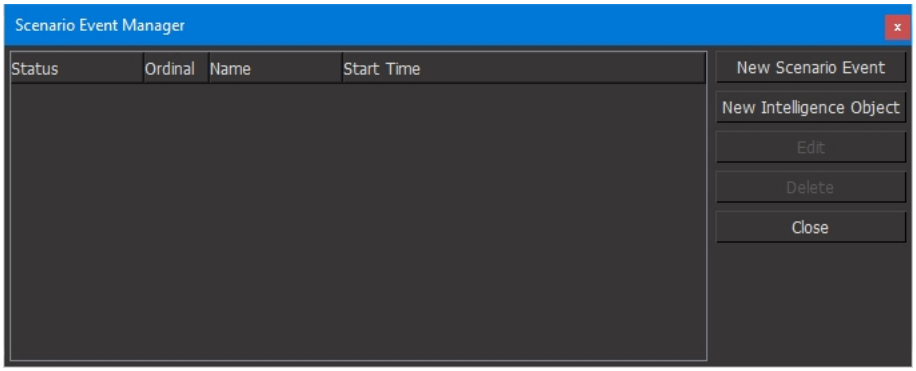
ⓘ If you want to play MP4 video for scenario events, you must install an appropriate driver on your computer.

3.2. Creating a Scenario Event

➤ **To create a scenario event:**

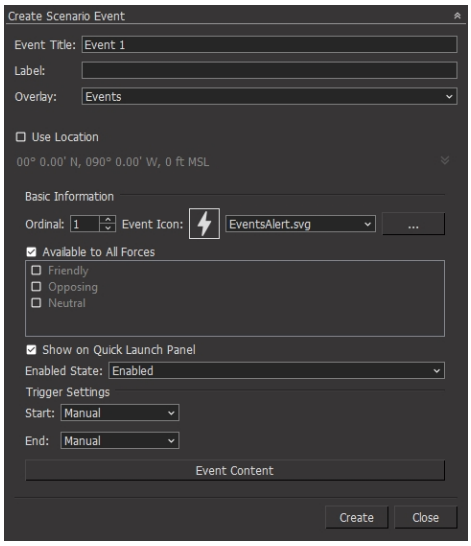
1. Choose **Simulation > Scenario Events** or click the Scenario Events button ( ) in the Scenario Components Toolbar. The Scenario Event Manager opens (Figure 28).

Figure 28. Scenario Event Manager



2. Click New Scenario Event. The Create Scenario Event dialog box opens (Figure 29).

Figure 29. Create Scenario Event dialog box



3. Set values for the parameters in the dialog box. Table 4 describes the parameters you can specify for a scenario event:

Table 4. Scenario event parameters

Parameter	Description
Event Title	A short descriptive name for the event that displays next to the icon on the Quick Launch Panel.
Label	The label provides more information about the event. This is used as the tooltip on the Quick Launch Panel. (optional)
Extended labels	If present, fill out extended labels. (optional)
Overlay	The overlay to which the event is assigned. Default: Events.
Use Location	You can specify a location for an event icon to appear when the event is triggered. You can click on the map where you want the event to occur or enter a location in any of the location formats. (optional)
Ordinal	Events are numbered in the sequence they are created. You can change the numbers to suit your purposes.
Event Icon	Select an appropriate icon to appear on the map at the location and time of the event. This icon is also used on the Quick Launch Panel. (optional)

Table 4. Scenario event parameters (continued)

Parameter	Description
Forces	You can make the event available to all or some forces. To limit it to some of the forces in the scenario, clear the Available to All Forces check box and then select the forces you want to get the event when it occurs. You can select multiple forces.
Show on Quick Launch Panel	Adds the event to the Quick Launch Panel for one-click activation.
Enabled State	Specify whether the scenario event is enabled, disabled, or linked to the enabled state of another scenario event.
Start	Specify one of the options for how the event is triggered: — Manual. Event will be triggered manually by the scenario administrator. — Date/Time. Specify the date and time of day that should trigger the event. — Linked. Specify the linking events as described in "3.2.2 Adding Linked Events" on page 54 that should trigger the event.. — Simulation Time. Specify the simulation time that should trigger the event.. — Immediate. The event starts as soon as it is created.
End	Specify how the end of the event is triggered: — Manual. Ended by the scenario administrator. — Date/Time. Specify the date and time of day at which it should end. — Duration. Specify how long the event lasts. — Permanent. The event does not end until you manually change the end type and end it. ⚠ If you trigger an event by time and the specified time is prior to the start time of the scenario, the event is triggered when the scenario starts.
Event Content	Add content as described in "3.2.1 Adding Content to a Scenario Event" on the facing page.

4. Click Create. Your new event appears in the event list. You can sort events by any of the column headings.

3.2.1 Adding Content to a Scenario Event

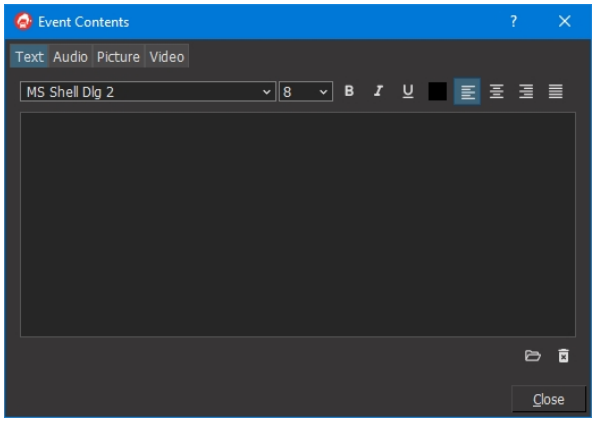
The content for a scenario event can be text, audio, video, graphics (BMP, JPG, SVG, or PNG), or some combination of them.

For sound and video files, you can specify files in any format that your computer supports.

➤ **To add content to a scenario event:**

1. Create an event as described in "3.2. Creating a Scenario Event" on page 50.
2. In the Create Scenario Event dialog box, click Event Content. The Event Contents dialog box opens (Figure 30).

Figure 30. Event Contents dialog box



3. Select the tab for the content type that you want for this event. You can have more than one type of content.
4. For video, picture, or audio:
    a. Click Choose. A file selection dialog box opens.
    b. Select the file you want to add.
    c. Click Open.

    For text:
    a. Type the text you want in the dialog box or click Load and select a text file to load.
    b. Format the text as desired.
5. Optionally, test video and audio files by clicking the Play button.
6. Click Close.

> ⚠ If the event content is too large, VR-Forces does not save the content when you click Create or Update in the Scenario Event dialog box.

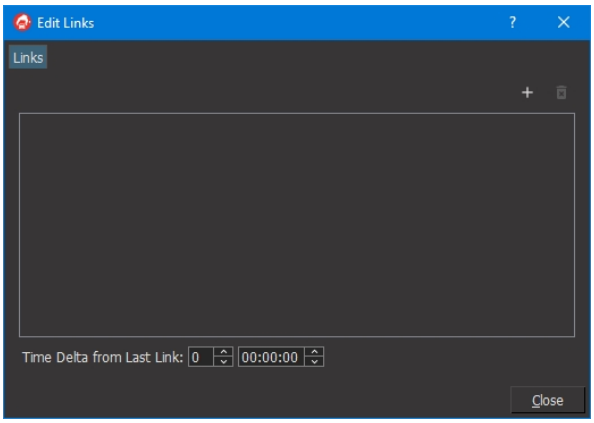
3.2.2 Adding Linked Events

You can link multiple events to a single preceding event (many to one) or link one event to another linked event if your scenario requires it (one to one). You can create a linked event to trigger at any time after the preceding event it is linked to. For example, an event of a bridge being washed away could be linked to a preceding event such as reporting heavy rains, flooding, or rising river levels.

➤ **To add a link to a scenario event:**

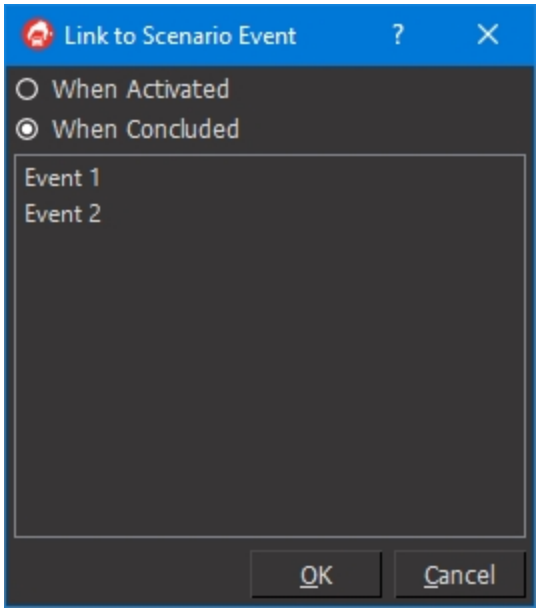
1. Create an event as described in "3.2. Creating a Scenario Event" on page 50.
2. For the Start trigger, select Linked. The Edit Links button is added to the dialog box.
3. Click Edit Links. The Edit Links dialog box opens (Figure 31).

Figure 31. Edit Links dialog box



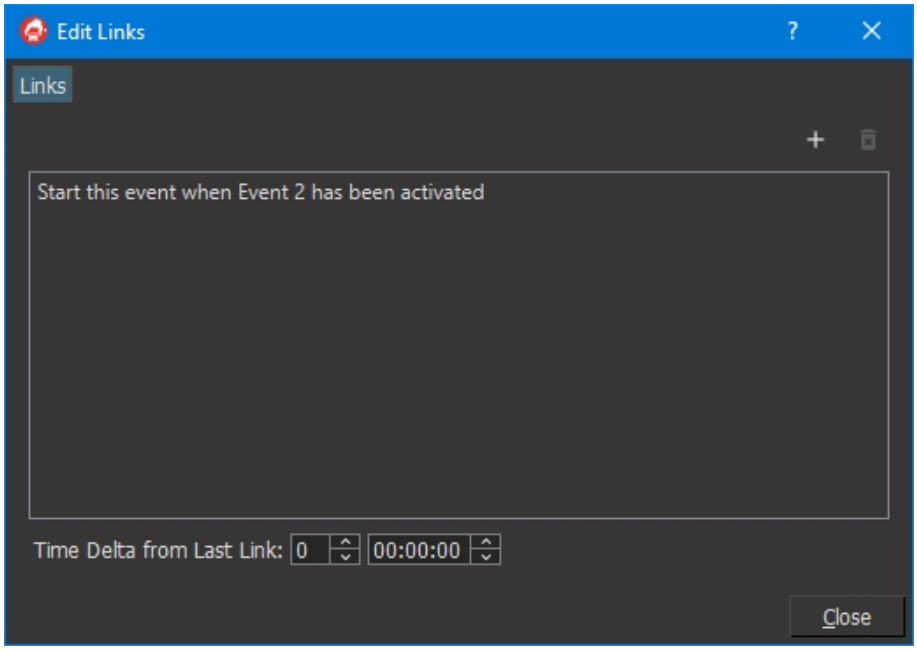
4. Click the Add button (+). The Link to Scenario Event dialog box opens (Figure 32).

Figure 32. Link to Scenario Event dialog box



5. Select When Activated or When Concluded to indicate the relationship of the event to its linked event.
6. Select the event you want to link to.
7. Click OK.
8. Optionally, add additional linked events.
9. Specify the time delta. This is the amount of time to wait after the preceding event before triggering the linked event. If you specify multiple linked events, this is the time after all event conditions have been met.

Figure 33. Edit Links dialog box with links



10. Click OK.

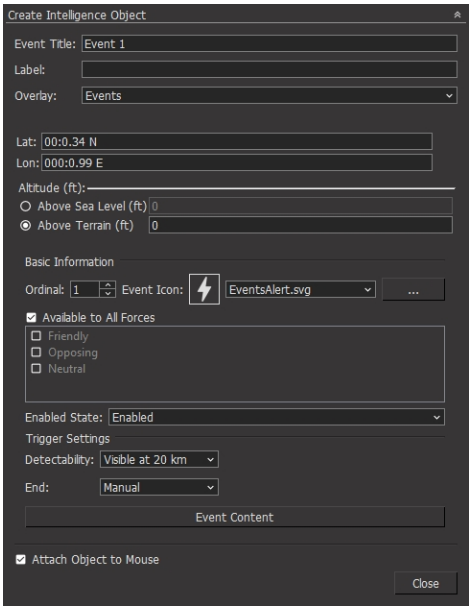
3.3. Adding Intelligence Objects

Intelligence objects are tactical graphics that trigger scenario events when they are detected by a simulation object. By default, intelligence objects are detected by visual sensors. However, you can configure them to be detectable by other sensors.

➤ **To add an intelligence object:**

1. Choose **Simulation > Scenario Events** or click the Scenario Events button ( ) in the Scenario Components Toolbar. The Scenario Event Manager opens (Figure 28 on page 50).
2. Click New Intelligence Object. The cursor changes to draw mode and the Create Intelligence Object dialog box opens (Figure 34).

Figure 34. Create Intelligence Object dialog box



3. Click where you want the intelligence object to be located, or type its location in the dialog box.
4. Specify basic information and trigger settings as you would for a scenario event ("3.2. Creating a Scenario Event" on page 50), except that instead of configuring a start event, you specify detectability.

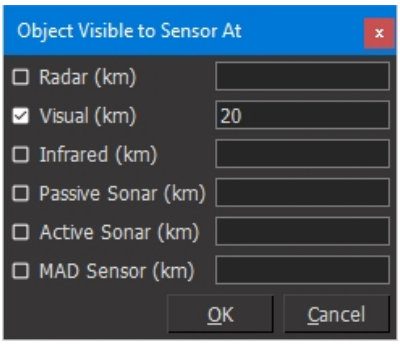
   To use only a visual sensor for detectability, select a distance in the Detectability list.

   To use other sensors, specify detectability:

   a. In the Detectability list, select User Defined. The Edit Detectability button is added to the dialog box.

   b. Click Edit Detectability. The Object Visible to Sensor At dialog box opens (Figure 35).

Figure 35. Set Sensor Signatures



   c. Select each sensor that you want to use to detect the intelligence object.

d. For each sensor specify the sensor signature. (Although sensor signatures are technically unit-less, as a practical matter the sensor signature represents the distance in kilometers at which the object can be sensed.)

e. Click OK.

f. Click Create.

3.4. Starting a Scenario Event

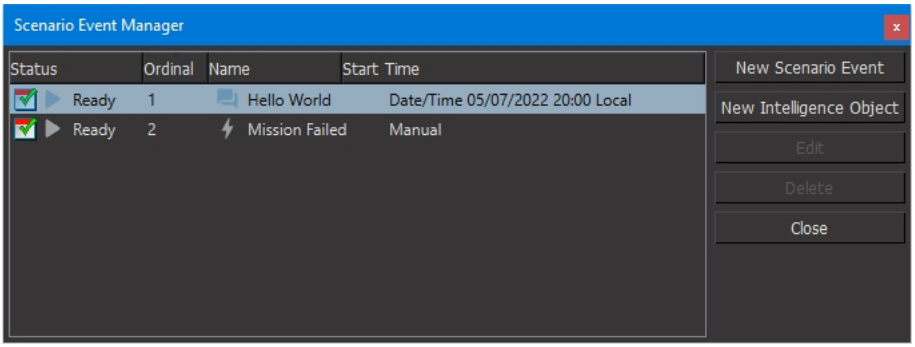
A scenario event starts based on the Start Event trigger, by a command in a plan, or manually. Regardless of how a scenario event's trigger is set up, you can manually start an event at any time.

> If an event has been added to the Quick Launch Panel, you can start it by clicking its icon and title text on the panel.

➤ **To start a scenario event manually:**

1. Choose **Simulation > Scenario Events**. The Scenario Event Manager opens (Figure 36).

Figure 36. Scenario Event Manager, Play button



2. In the Status column, click the Play button (▶) for the event you want to start (Figure 36). The Event window opens (Figure 37).

Figure 37. Event window

3.4.1 Controlling Scenario Events from a Plan

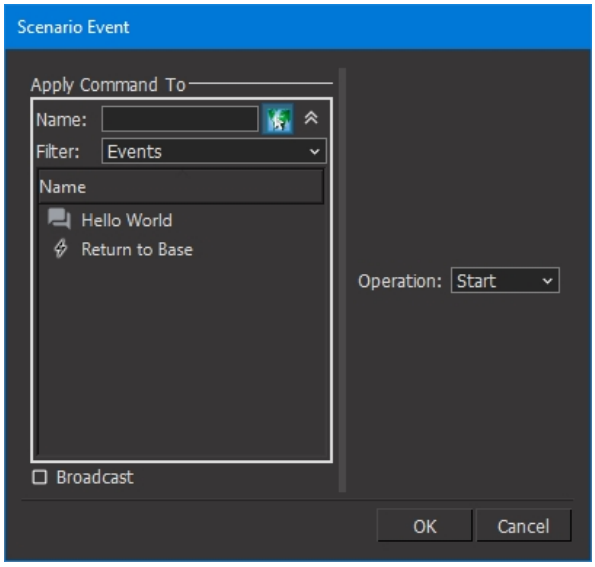
You can start, stop, enable, and disable scenario events from plans. For example, a simulation object's plan could have an Entity In Area condition and when that condition is satisfied, you could trigger an event such as, "The enemy has seized your headquarters!" or "Congratulations, you have seized the enemy headquarters!".

Scenario events can also be used as the condition in a trigger. For details, see "Plan Concepts" and "Writing Plans".

➤ **To send a scenario event in a plan:**

1. In a plan window, choose **Commands > Scenario Event**. The Scenario Event dialog box opens (Figure 38).

Figure 38. Scenario Event dialog box



2. Select the event you want to send.
3. Select an option in the Operation list.
4. Click OK.

3.5. Ending a Scenario Event

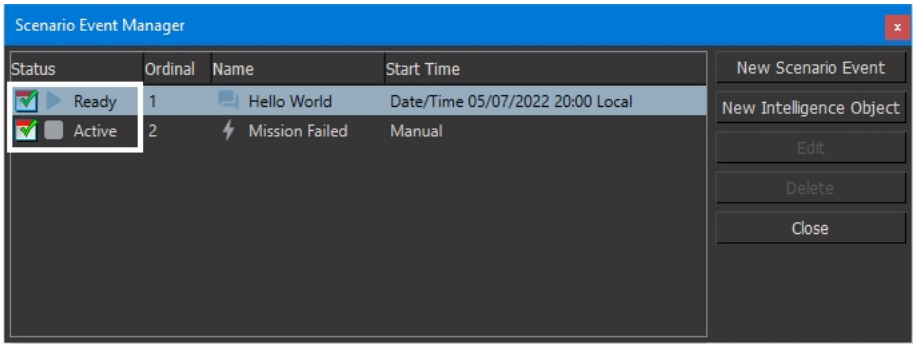
An event ends based on the End Event trigger setting, by a command in a plan, or manually.

> When a scenario event ends, the Event window does not close automatically. The title bar indicates that the event is over by changing the text from Active to Inactive. You must close the window manually.

➤ **To end a scenario event manually:**

1. Choose **Simulation > Scenario Events** or click the Scenario Events button ( ) in the Scenario Components Toolbar. The Scenario Event Manager opens.
2. In the Status column, click the Stop ( ) button for the event you want to stop (Figure 39).

Figure 39. Scenario Event Manager, Stop button



> If the Status column says Active, but does not have a Stop button, this is a permanent event. To end it, you must edit the event and change the End mode to Manual. A Stop button is then displayed.

3. Click Close.

3.6. Enabling and Disabling Scenario Events

You can enable and disable scenario events. You can change their status from the Scenario Event Manager, the Edit Scenario Event dialog box, and from plans. For details about enabling and disabling a scenario event from a plan, see "3.4.1 Controlling Scenario Events from a Plan" on page 58.

➤ **To enable or disable a scenario event in the Scenario Event Manager:**

1. Choose **Simulation > Scenario Events**. The Scenario Event Manager opens (Figure 39 above).
2. In the Status column, click the Enable/Disable ( ) button for the event you want to stop (Figure 39).

➤ **To enable or disable a scenario event in the Edit Scenario Event dialog box:**

1. Choose **Simulation > Scenario Events**. The Scenario Event Manager opens (Figure 39).
2. Select the event you want to edit.
3. Click Edit. The Edit Scenario Event dialog box opens (similar to Figure 29 on page 51).
4. In the Enabled State list, select the option you want.
5. Click Update.

> You can also click the Scenario Events button ( ) in the Scenario Components Toolbar to open the Scenario Event Manager.

3.7. Sorting the Event List

Scenario events have ID numbers. The numbers have no effect on when events are sent. They are simply a convenience for instructors and players. You can sort the list by number or by any other column listed.

➤ To change the sort order of scenario events, click the heading of the column that you want to sort on.

3.8. Deleting a Scenario Event

➤ **To delete a scenario event:**

1. Choose **Simulation > Scenario Events** or click the Scenario Events button ( ) in the Scenario Components Toolbar. The Scenario Event Manager opens (Figure 28 on page 50).
2. Select the event you want to delete.
3. Click Delete.
4. Click Yes in the confirmation message.

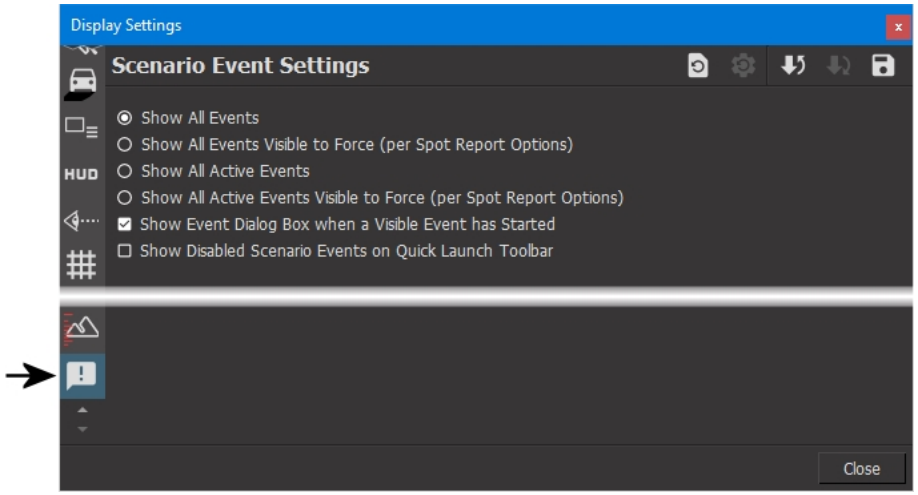
3.9. Showing and Hiding Scenario Events

You can specify which events, if any, display and whether the Event dialog box displays when an event is triggered. The visible to force settings refer to visibility according to spot report settings and are based on the force availability specified when a scenario event is created or edited.

➤ **To show or hide scenario events:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Scenario Event Settings page (Figure 40).

Figure 40. Scenario Event Settings page



3. Select an option for viewing events.

4. Target Detection and Combat Features

This chapter describes VR-Forces features related to target detection and acquisition, such as spot reports, hostility relationships, lasing, and munition damage.

4.1. Displaying Spot-Reported Simulation Objects

A spot report is a report by a simulation object to members of its force, notifying them that it has detected a member of another force.

By default, VR-Forces displays simulation objects based on complete ground truth. That is, it shows all simulation objects in their exact location. In real-world combat and training situations, however, participants rarely know where all other participants are, including those in their own force. This is called the fog of war. VR-Forces can simulate the fog of war by displaying simulation objects based on spot reports rather than ground truth.

When you display simulation objects based on spot reports, you do not see all members of forces other than your own. You only see spot-report icons, also known as tracks, for other-force simulation objects that have been detected by members of your force. Spot-report icons are similar to simulation object icons, but they do not move as simulation object icons do. Furthermore, since spot reports lose value over time as the detected entities may move to different locations, VR-Forces maintains a certainty level for the location of spot-reported entities. If a detected entity is not detected again for a period of time, its certainty level decreases and its icon becomes increasingly transparent to show that it is less certain that the entity is at the reported location.

- Generating spot reports can reduce the performance of the simulation engine.
- When spot reports are enabled, VR-Forces does not display icons for dead entities.
- When you rewind a scenario, the global spot report setting persists. Spot report settings for individual entities revert to the settings saved with the scenario.
- If you enable spot reports and you use Stealth observer mode, VR-Forces uses 2D icons for the entities that are identified by spot reports. You do not see any 3D models.

You can find links to videos that demonstrate simulation object classification and spot reports at https://www.mak.com/mak-one/support/help/tutorials, or click the camera icon next to the procedure whose video you want to view.

4.1.1 Enabling or Disabling Spot Reports

You can enable or disable spot reports globally or per simulation object. To set spot reports for a simulation object, use the Spot Reports set data request, as described in "Spot Reports".

By default, when you enable spot reports, they are sent only to the GUI (for display purposes). They are not sent to other simulation objects. This reduces the performance impact of spot reports on the sim. engine, but it means that entities do not receive that information for targeting purposes. Their spot report receivers do not show any spot reports in the Information dialog box. If you want spot reports to be sent to simulation objects, you must enable spot reports as described here and also specify that spot reports be sent to simulation objects using the Spot Report set data request. For more information, see "Spot Reports".

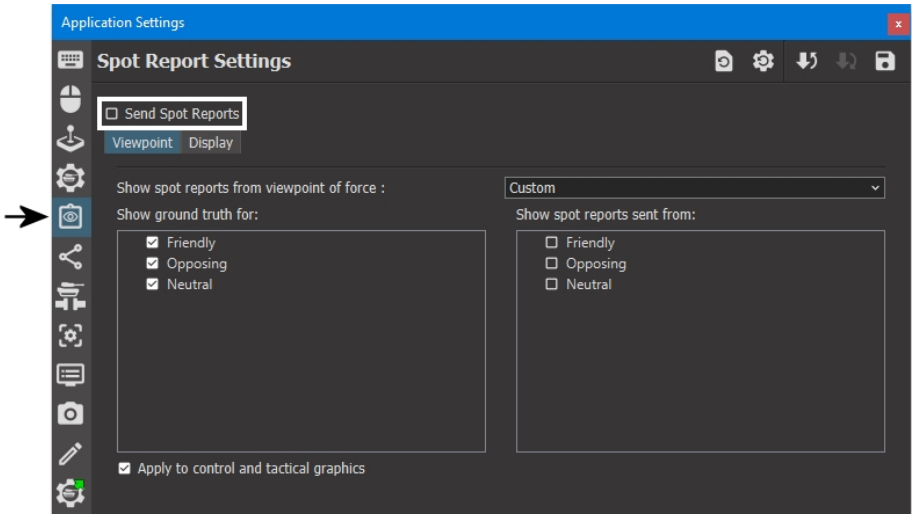
Enabling or Disabling Spot Reports Globally

When you enable or disable spot reports globally, the setting is applied to all simulation objects that have not explicitly set spot reports on or off with the Spot Reports set data request. The setting persists, which allows you to set a simulation object to use the global spot report setting.

➤ **To enable or disable spot reports for all simulation objects:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Viewpoint tab (Figure 41).

Figure 41. Spot Report Settings page, Viewpoint tab



4. Select or clear the Send Spot Reports check box. The change takes effect immediately.

4.1.2 Configuring the Spot Reports Viewpoint

The spot report viewpoint determines which simulation objects and tactical graphics are shown using ground truth and which are shown using only spot-report icons. You can select preconfigured viewpoints or create custom viewpoints.

- If you create a simulation object for a force that is not visible based on the viewpoint setting, after you create the simulation object, it disappears. Therefore, we recommend that you show ground truth for all forces when you are creating entities, tactical graphics, simulation effect objects, and so on.
- When you rewind a scenario, the viewpoint configuration stays the same as in the previous run of the scenario. The global spot report setting stays the same. Entity-specific spot report settings revert to the settings saved with the scenario.

Selecting a Preconfigured Spot Report Viewpoint

VR-Forces provides viewpoints for the friendly, opposing, and neutral forces. In each of these viewpoints, VR-Forces displays ground truth for the viewpoint force and spot-report icons for the other forces. For example, if you select the Friendly force viewpoint, all simulation objects in Force 1 are displayed showing ground truth. All other simulation objects are displayed only if they have been detected by a friendly simulation object that is configured to send spot reports.

When you select a preconfigured viewpoint, the check boxes in the windows in the Map View group box are automatically set to match the selected viewpoint.

➤ **To configure spot reports:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Viewpoint tab (Figure 41 on the previous page).
4. Select an option in the Show Spot Reports from Viewpoint of Force list.

Configuring Custom Spot Report Viewpoints

If you do not want to use a preconfigured viewpoint, you can create a custom viewpoint. This option allows you to see ground truth and spot-report icons from a mix of forces. When you create a custom viewpoint, the Show Spot Reports from Viewpoint of Force list displays the viewpoint that you have created (usually Custom).

It is possible to create a custom configuration in which no objects are displayed, so think carefully about your choices.

➤ **To configure a custom viewpoint:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Viewpoint tab (Figure 41 on the previous page).
4. In the Show Ground Truth For window, select the forces for whom you want to display the ground truth. You would usually select the same forces as the ones selected in the Show Spot Reports Sent From window.
5. In the Show Spot Reports Sent From window, select the forces whose spot-report icons you want to display. In other words, if you are not showing ground truth for the opposing force and only want to see the opposing force simulation objects that

are detected by friendly and neutral simulation objects, then select the Friendly and Neutral check boxes.

Configuring Same-Side Fog of War

There may be cases when you want to simulate same-side fog of war. That is, members of a force do not know where other members of their force are unless detected by a simulation object and reported using a spot report.

To simulate same-side fog of war, you must edit the *spot-report-generator.sysdef* file. You might want to create a customized file that you can assign only to the simulation objects that you want to simulate same-side fog of war. (You can use the Simulation Object Editor to change the spot report systems that simulation objects use.) In the system definition file change the **send-spot-reports-on-own-force** parameter to True, as shown on line 14:

```
1   (spot-report-generator-system
2     (controllers
3       (spot-report-generator
4         (component-descriptor-type
5           "spot-report-generator-controller-descriptor")
6         ...
7         (send-stop-spotting-reports False)
8         (object-types-to-spot-report
9           (object-type 1 (1 -1 -1 -1 -1 -1 -1))
10          (object-type 1 (2 -1 -1 -1 -1 -1 -1))
11          (object-type 1 (3 -1 -1 -1 -1 -1 -1))
12          (object-type 1 (5 -1 -1 -1 -1 -1 -1))
13        )
14        (send-spot-reports-on-own-force True)
15        (send-spot-reports-on-hostile-forces True)
16        (send-spot-reports-on-neutral-forces True)
17        (broadcast False)
18        ...
19        (send-spot-reports-on-scenario-load True)
20        (send-to-front-end-only True)
21        (only-one-destroyed-report False)
22        (send-only-on-data-change True)
23      )
24    )
25    ...
26  )
```

Configuring Other Options for Spot Reports

You can change other defaults in the sysdef file, including limiting the **object-types-to-spot-report**. Reducing the number of objects detected makes spot reporting less computationally expensive. Another parameter that you might want to change to reduce the number of objects is **send-spot-reports-on-neutral-forces** by setting it to False.

You can also change the values of the **send-only-to-front-end** and **broadcast** parameters to change the default behavior of spot reports and how they affect performance. Setting **send-only-to-front-end** to False sends spot reports to other sim. engines and simulation objects. Setting broadcast to true sends spot reports to all simulation objects in the same force.

The **send-stop-spotting-reports** parameter controls when to send out stop spotting reports. When it is set to false, VR-Forces sends stop spotting reports only when an entity is acting as a remote sensor. Entities receiving spot reports from a remote sensor treat them as if they had detected the spotted objects themselves. When the parameter is set to true, they are sent whenever detection is lost, not only for remote sensor receivers. You do not need to wait for a timeout to know that an entity is no longer detected.

4.1.3 Configuring the Spot Reports Certainty Level

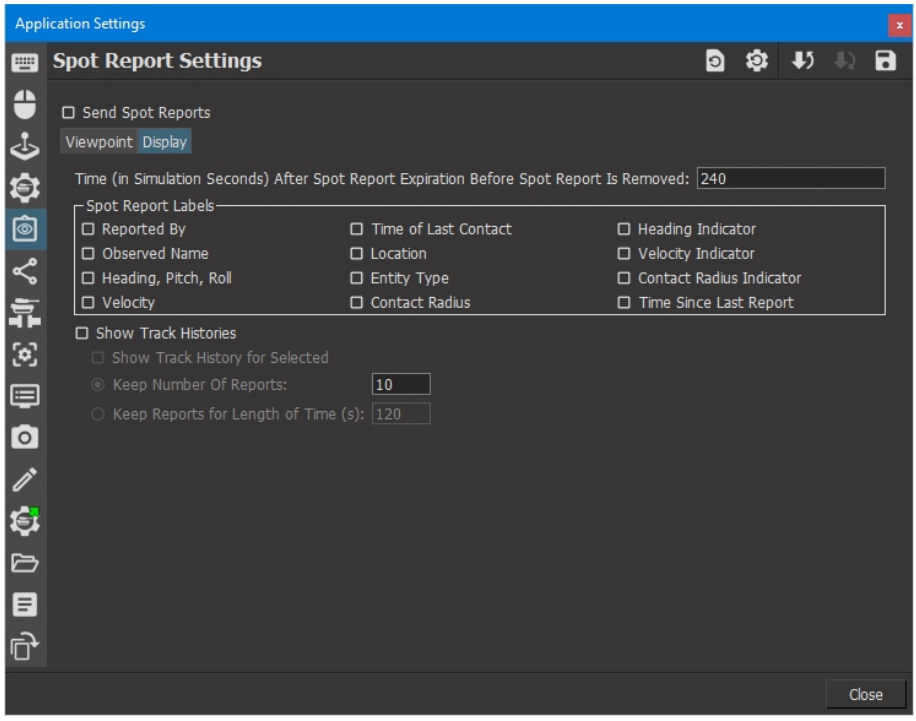
It is assumed that as time passes the validity, or certainty, of a spot report declines. To simulate this decline in certainty, VR-Forces can display the icons that represent spot-reported entities at increasing levels of transparency. Once the initial validity period has expired, the spot report degrades over a configurable amount of time. The icon's transparency decreases in four steps, with each step being 25% of the total time allotted for degradation.

> ⓘ The validity period is specified by the **report-resend-period** parameter in the *spot-report-generator.sysdef* file in the SMS you are using. The default is 60 seconds.

➤ **To configure the amount of time, in seconds, over which a spot report degrades:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Display tab (Figure 42).
4. Specify a value, in seconds, in the Time After Spot Report Expiration Before Spot Report Is Removed box.

Figure 42. Spot Reports Option, Display tab



4.1.4 Applying Spot Reports to Tactical Graphics

If a tactical graphic is assigned to a force, you can display it based on spot reports rather than ground truth. The same criteria are applied to their display as are applied to the display of simulation objects. For information about how to assign objects to a force, see "Creating Tactical Graphics".

1. To apply spot reports to tactical graphics:
2. Choose **Settings > Application**. The Application Settings dialog box opens.
3. Select the Spot Report Settings page.
4. Select the Viewpoint tab (Figure 41 on page 65).
5. Select the Apply to Control and Tactical Graphics check box.

4.1.5 Displaying Labels for Spot Reports

You can display labels (symbol decorations) for the icons of spot-reported entities with information similar to that for regular simulation object icons.

➤ **To display spot report labels:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Display tab (Figure 42 above).
4. In the Spot Report Labels group box, select the labels that you want to display.

> You can also enable labels for spot-reported entity icons on the Display Settings dialog box, Symbol Decoration Settings page.

4.1.6 Using Spot Reports in Tasks

Spot-reported simulation objects are listed in the Objects List Panel and other simulation object lists. Although the spot-report icon in the display window may not show information about a spot-reported object if it has not been identified, the entry in the list includes the object name. You can filter the list to show only spot-report icons.

You can select a spot-reported object as a parameter in tasks that require you to select a simulation object. If you use a spot-reported simulation object in a task, VR-Forces carries out the task based on the ground truth of the reported simulation object, not the location of the spot-report icon. For example, if you specify a spot-reported object in the Follow Entity task, and the target simulation object moves, the following simulation object will move toward the actual location of the spot-reported simulation object, not its location when the spot report was made.

4.1.7 Displaying Track Histories for Spot-Reported Objects

You can display track histories for spot-reported simulation objects (Figure 43). When you enable display of track histories for spot-reported objects, it does not actually cause them to display. You must also enable them for the selected spot-reported object and select a spot report.

Figure 43. Spot report track history

> The Track Histories setting on the Observer menu and the Observer toolbar do not affect the display of track histories for spot-reported objects. That feature shows entities' histories.

➤ **To display track histories for a spot-reported simulation object:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Spot Report Settings page.
3. Select the Display tab (Figure 42 on page 69).
4. Select the Show Track Histories check box.
5. Select the Show Track History for Selected check box.
6. Select the spot-reported object whose track history you want to show.

Pinning a Spot Report Track History

If you want to see more than one spot report track history at a time, you can pin track histories. If a track history is pinned, you do not have to select the spot-reported simulation object to see its track history.

➤ **To pin the track history for a spot-reported object:**

1. Select the spot-reported simulation object.
2. Choose **Objects > Decorations > Pin Track History**.

Removing Spot Report Track History Pins

➤ **To remove spot report track history pin:**

1. Select a simulation object that has its track history pinned.
2. Choose **Objects > Decorations > Pin Track History**. The track history pin is removed.

➤ To remove track history pins for all entities, choose **Objects > Decorations > Remove All Pin Track History**.

4.2. Displaying Tactical Data Link Messages

VR-Forces can display information about tactical data link (TDL) messages received over the network or sent by simulation objects that have a tactical data link message generator system. Systems are provided for each major platform type. TDL support is provided by an ITAR-controlled plug-in. However, if you do not have the plug-in, the base VR-Forces product simulates the effect of generating TDL messages.

When VR-Forces receives a TDL message, it displays:

- An icon similar to a spot report icon.
- Optionally, the time since the last report.
- Optionally, the data that accompanies the message, in up to 9 lines. The number of lines to display is configurable. There is no particular type of information or ordering information associated with the lines. It is simply based on the data in the TDL message.
- Optionally, track histories for the TDL reports.

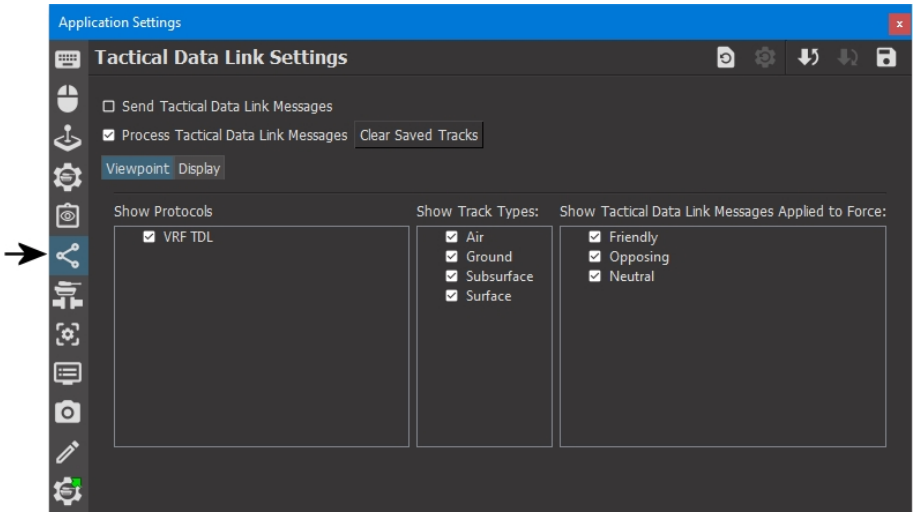
For information about configuring the display of TDL messages, see "4.2.1 Configuring the Display of TDL Messages" on the next page.

You can enable TDL messaging globally or for individual simulation objects. To enable TDL messages for a specific simulation object, use the TDL Messaging set data request. For details, see "TDL Messaging".

➤ **To enable TDL messaging globally:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Tactical Data Link Settings page (Figure 44).

Figure 44. Tactical Data Link Settings page, Viewpoint tab



3. To send TDL messages, select the Send Tactical Data Link Messages check box.
4. To display TDL messages, select the Process Tactical Data Link Messages.

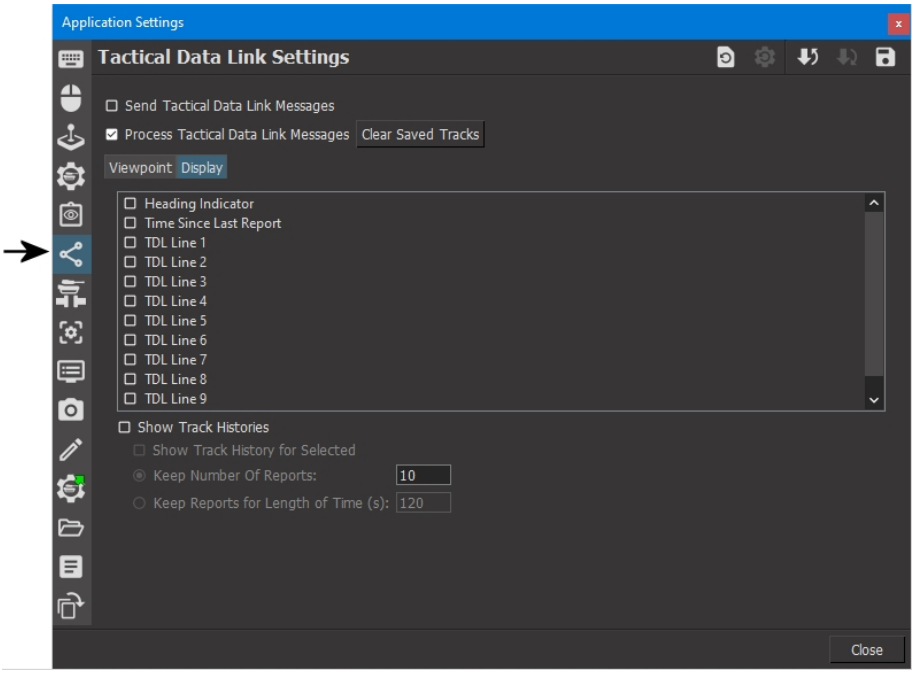
4.2.1 Configuring the Display of TDL Messages

You can configure how VR-Forces displays TDL messages.

➤ **To configure TDL messaging:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Tactical Data Link Settings page.
3. Select the Viewpoint tab (Figure 41 on page 65).
4. In the Show Protocols list, select the type of protocol for which you want to show messages. (If the ITAR plug-in is installed, the VR-Forces TDL messages are not sent.)
5. In the Show Track Types list, select the platforms for which you want to show messages.
6. In the force list, select the forces from which you want to display messages.
7. Select the Display tab (Figure 45).

Figure 45. Tactical Data Link Settings, Display page



8. Select check boxes for the data you want displayed next to the TDL message icon.
9. Optionally, select the Show Track Histories check box.
10. If you enable track histories, specify whether to show track histories only for the selected entity, the number of reports to track, and how long to display them. For information about how to display track histories, see "4.2.2 Displaying Track Histories for TDL Messages" below.

4.2.2 Displaying Track Histories for TDL Messages

You can display track histories for tactical data link (TDL) messages. When you enable display of track histories for TDL messages, it does not actually cause them to be displayed. You must also enable them for the selected message and select a message.

> The Track Histories setting on the Observer menu and the Observer toolbar do not affect the display of track histories for TDL messages.

➤ **To display track histories for a TDL message:**

1. Choose **Settings > Application**. The Application Settings dialog box opens.
2. Select the Tactical Data Link Settings page.
3. Select the Display tab (Figure 45 above).
4. Select the Show Track Histories check box.
5. Select the Show Track History for Selected check box.
6. Select the TDL message whose track history you want to show.

Pinning a TDL Message Track History

If you want to see more than one TDL message track history at a time, you can pin track histories. If a track history is pinned, you do not have to select the message to see its track history.

➤ **To pin the track history for a TDL message:**

1. Select the message.
2. Choose **Objects > Decorations > Pin Track History**.

Removing TDL Message Track History Pins

➤ **To remove TDL message track history pins:**

1. Select a simulation object that has its track history pinned.
2. Choose **Objects > Decorations > Pin Track History**. The track history pin is removed.

➤ To remove track history pins for all entities, choose **Objects > Decorations > Remove All Pin Track History**.

4.3. Managing Force Hostility Relationships

VR-Forces categorizes simulation objects as members of a force, such as friendly, opposing, or neutral. By default, friendly forces and opposing forces are hostile to each other. Neither is hostile towards neutral forces. You can change force hostility relationships dynamically. This may be particularly useful if you are simulating an environment in which civilian populations become hostile to a particular force, or in which armed militias, for example, switch allegiances to other forces.

You can add new forces in the Simulation Object Editor. For details, see "Managing the Forces List". You can change a force's hostility in a global or individual plan. For details, see "4.3.1 Changing a Force's Hostility in a Plan" on the facing page.

You can change the default hostility relationships by editing the *forceHostilty.mtl* file for the simulation model set you are using (for example, *./data/simulationModelSets/<model_set>/forceHostilty.mtl*).

When you change force hostility at runtime, the changes are saved as part of the scenario. The hostility relationships loaded with a scenario override the hostility relationships loaded by the simulation model set's hostility file.

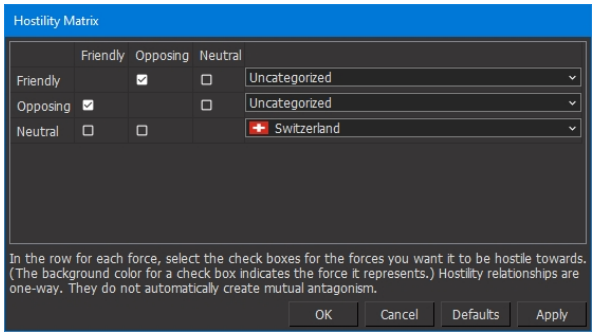
ⓘ

- Hostility relationships are not reciprocal. For example, if you set neutral force simulation objects to be hostile to friendly force simulation objects, that does not mean that friendly force simulation objects are hostile to neutral force simulation objects.
- If you click the Defaults button, it resets the Hostility Matrix to the values when you first opened the dialog box in the VR-Forces GUI during the current session. If the you have not made any changes, clicking the Defaults button has no effect.

➤ **To change the force hostility matrix:**

1. Choose **Simulation > Hostility Matrix**. The Hostility Matrix dialog box opens (Figure 46). Each row in the matrix lists the hostility relationships between that row's force and all of the other forces (represented by color-coded columns).

Figure 46. Hostility Matrix



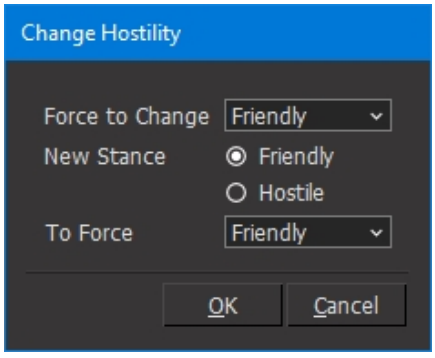
2. For each row in the matrix, select the check boxes in the columns for the forces that you want this force to be hostile towards. Clear the check boxes for forces that you do not want it to be hostile towards.
3. If desired, assign a country to a force by choosing it from the drop-down. When you create an entity for that force and the entity has markings defined for that country, then the entity will automatically display those markings.
4. Click OK.

4.3.1 Changing a Force's Hostility in a Plan

➤ **To change a force's hostility in a global or local plan:**

1. Open a global plan or an individual plan.
2. Choose **Commands > Change Hostility**. The Change Hostility dialog box opens (Figure 47).

Figure 47. Change Hostility dialog box



3. In the Force to Change list, select the force whose hostility you want to change.
4. Select the new force relationship (New Stance), either Friendly or Hostile.

5. In the To Force list, select the force towards which you want to change hostility.
6. Click OK. The command is added to the plan.

4.4. Detecting Targets

Simulation objects can have one or more sensors, for example, visual, sonar, or radar, for detecting and identifying other simulation objects. If a simulation object has a Remote Sensor Receiver system, it can use another entity's sensors to discover sensor contacts. (For more information about remote sensors, see "Remote Sensor".)

When a simulation object detects another simulation object, it assigns it a combat identification (CID) level. If it continues to maintain contact with the simulation object, it upgrades the CID level:

- CID Level 0 – not detected.
- CID Level 1 – Detected: the simulation object is detected, but only its platform is known, for example, ground.
- CID Level 2 – Classified: the simulation object's category is known, for example, tank.
- CID Level 3 – Identified: the simulation object's category and subcategory are known, for example M1A2.
- CID Level 4 – Full Knowledge: all visible information about the simulation object is known.

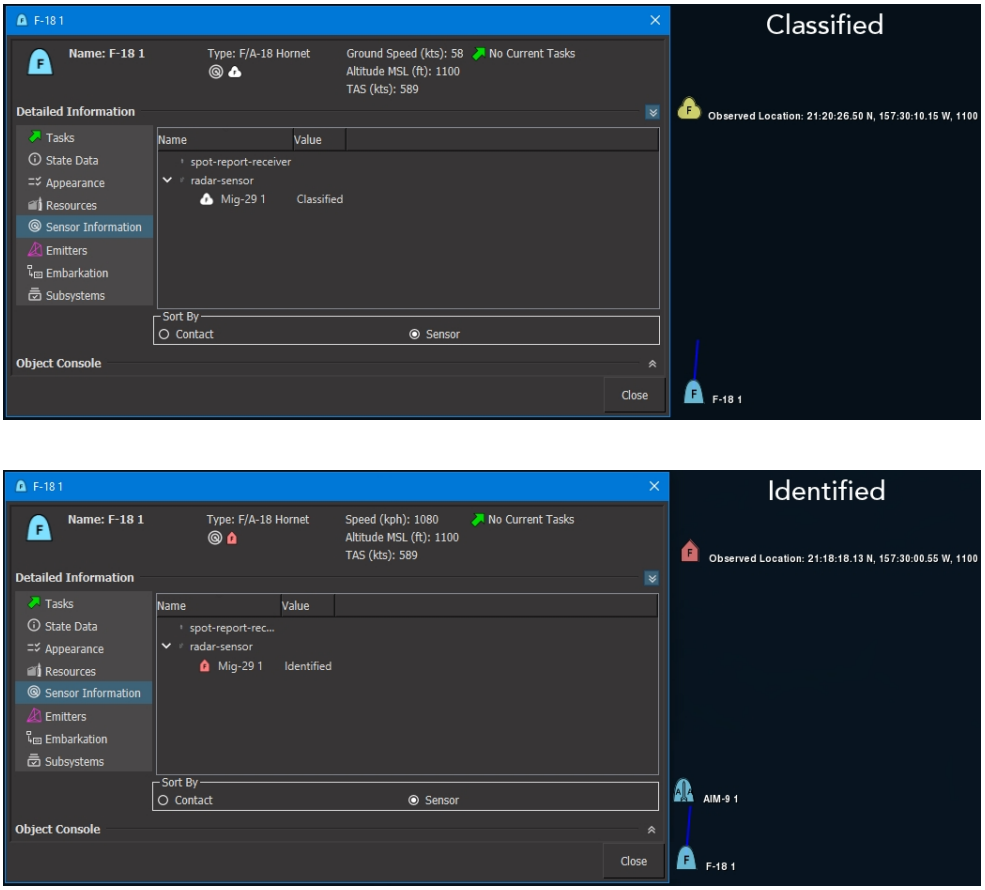
CID levels are assigned based on the distance to the detected simulation object and the length of time that the simulation object has been in contact. You can configure these values in the detection tables (*./data/simulationModelSets/<model_set>/vrfSim/detection/*.csv*). For details about these configuration files, see "Detection Tables".

By default, simulation objects do not fire at a detected hostile simulation object until it achieves the Identified level. You can configure this behavior by changing the **detection-level-to-set-hostility** parameter in the simulation object's sensor system.

Figure 48 shows screen captures of a simulation object as its CID level is advanced from Classified through Identified. In this scenario, spot reports are enabled. Therefore, when a simulation object is detected, it displays with a yellow icon, rather than the normal color for the force. It is not given the correct force color until the detected simulation object is CID level Identified.

You can find links to videos that demonstrate simulation object classification and spot reports at https://www.mak.com/mak-one/support/help/tutorials, or click the camera icon next to the procedure whose video you want to view.

Figure 48. Simulation object classification and display when spot reports are enabled



4.4.1 Target Detection and Spot Reports

Simulation object detection and classification can be enhanced by enabling spot reports. Detected simulation objects are assigned CID levels based on length of contact and distance from the detecting simulation object. In some cases, such as a fixed-wing aircraft flying quickly over a battlefield, or a simulation object that is at the farthest point of its range from a target, contact might not be sufficient to advance to the Identified CID level. In such cases, even though the simulation object is technically capable of firing on the target, it might not do so because the CID level is not high enough.

If spot reports are enabled, the classification of simulation objects in the spot reports that a simulation object receives is combined with the directly sensed information. Therefore, if a simulation object classifies a target at a low CID level, but it receives a spot report that classifies the target at a high CID level, it can fire on the target based on the CID level in the spot report. For example, as noted in the previous paragraph, a fixed-wing aircraft is unlikely to maintain contact with a ground-based simulation object long enough to elevate its CID level to Identified. However, if a member of its force on the ground identifies the target and sends out a spot report indicating CID level 3 or 4, when the fixed-wing aircraft receives the spot report, it can add the target to its target list.

4.4.2  Lasing Targets

Some entities, such as DI Lasing, have laser-targeting capability. Others, such as attack helicopters, have laser targeting capability and carry laser-guided missiles. Laser-guided missiles use the standard VR-Forces targeting mechanisms. They fire on targets that are identified by laser beams that have the same laser code as the laser-guided missile launcher. The laser beam could be generated by a laser designator on the same entity (autonomous lasing), or could be generated by another entity. Entities with both a laser designator and a missile launcher use their laser and missile when autonomously engaging targets.

Laser codes are numbers in the range of 1111-1788 using the digits 1 through 8. Three-digit codes in the range 111-788 are also supported with an implied initial 1 in the code. By default, VR-Forces calculates a unique laser code for each entity when it is created. When you assign a laser code to an entity, any laser beams sent by that entity use the assigned laser code and the missiles carried by that entity (if applicable) use the same code. Therefore, for example, the default behavior for attack helicopters would be to fire their laser-targeted missiles only at targets that they have lased.

> ⓘ A laser-guided missile launcher must be able to see a laser spot and an appropriate target before it fires. If the missile loses the laser spot before it hits the target, it tracks to the last known laser spot.

Autonomous Lasing

By default, entities that have laser-guided missiles are configured for autonomous lasing. This means that they can lase targets and fire at them without outside intervention (for example, using the Lase Target task). Entities with autonomous lasing enabled identify and fire at targets independently, just like entities that use conventional munitions.

Synchronizing Laser Codes

There are cases in which you will want more than one entity to use the same laser code. For example, you might want one entity to lase targets (the designator), and another one to fire on them (the shooter). In this case you would assign both entities the same laser code. The designator would be given the hold-fire rule of engagement or not have any missile resources. The shooter would be configured to not lase targets (Lase Autonomous would be set to Off). It would just fire on the targets identified by the designator's laser.

You can synchronize the laser codes of the lasing entity and the firing entity by setting them for each entity with the Laser Code set data request, or you can let VR-Forces synchronize the codes with the Synchronize Laser Code set data request. For details, see "Laser Code" and "Synchronize Laser Code".

4.5. Engagement Zones

Engagement zones are areas associated with ground-based simulation objects and air units that affect their rules of engagement. VR-Forces supports two types of engagement zones: missile engagement zones (MEZ) and joint engagement zones (JEZ). Engagement zones can have infinite height, like most VR-Forces areas, or can have a floor and ceiling, like extruded areas.

MEZs usually refer to zones associated with a ground-based air defense system. The system is permitted to engage targets within this defined area. Multiple engagement zones can be associated with a single air defense system and multiple systems can be associated with an engagement zone. To associate simulation objects with an engagement zone, you use the Engagement Zone set data request. For details, see "Engagement Zones".

JEZs are zones in which multiple systems can engage targets, usually both air units and ground based units. Successful management of fire in a JEZ may require a mechanism to manage the various systems and to help prevent friendly fire.

When a simulation object that is associated with an engagement zone detects a target, VR-Forces checks its rules of engagement. If the ROE is set to Fire When in Engagement Zone and if the target is in the engagement zone, it can fire on the target. If the target is not in the engagement zone, it cannot fire at it.

Ordinarily, simulation objects do not fire on other simulation objects unless they have a combat identification level of Identified or Full Knowledge. An engagement zone can be set to assume that unknown objects are hostile. In this case, if a simulation object has the Fire When in Engagement Zone rule of engagement, it can fire on unknown objects. For more information, see "Assume Unknown is Hostile".

Radars detect the position and type of potential targets, but not the force. Therefore, when a simulation object is set to fire on unknown objects, it will fire on objects in its own force if they do not identify themselves as being of the same force. The force of a target is known only if it has an active transponder (IFF is enabled) and it is of the same force as the radar.

Like other areas, engagement zones can be activated and deactivated. If an engagement zone is inactive, simulation objects whose ROE is set to Fire When in Engagement Zone cannot fire at targets when they are in the zone.

You can create engagement zones just like you would other areal objects. For details, see "Creating and Placing Objects" and "Creating Volumetric (Extruded) Tactical Graphics".

4.6.  Electronic Warfare

In entity-level scenarios, VR-Forces supports radar jamming by entities that have a tactical radar jammer system. The jamming entity must have line-of-sight to the jammed entity and the jammed entity must be within the cone of the electromagnetic emissions from the jamming entity. (For information about electromagnetic emissions, see "Displaying Electromagnetic Emissions".)

Radar jamming is enabled and set up using the set data requests:

- "Add Jamming Target".
- "Jammer State".
- "Jamming Mode".
- "Remove Jamming Target".

> There is also a model for jamming at the aggregate level. See "Electronic Warfare (Aggregate-Level)".

4.7. Radar Coverage

VR-Forces can display the area in which a simulation object's radar can detect objects. The radar coverage area is color coded based on the altitude at which it is testing intersections. Each color shown indicates full visibility at that altitude and above. A sensor might be able to detect simulation objects at a lower altitude, but this is not certain. You can specify the altitudes at which you want to test intersections and can specify the colors used to display the radar coverage.

In 2D, the coverage area is depicted as bands of color (Figure 49). The color indicates the minimum altitude layer at which the radar can see in that direction.

Figure 49. Radar coverage, 2D and color key

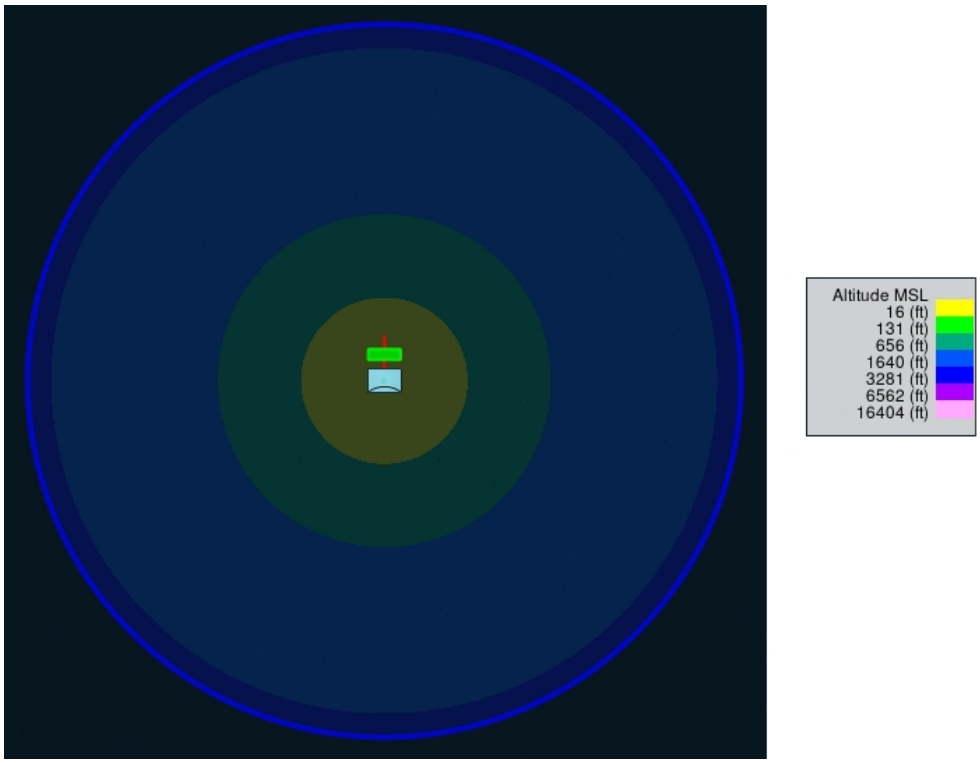
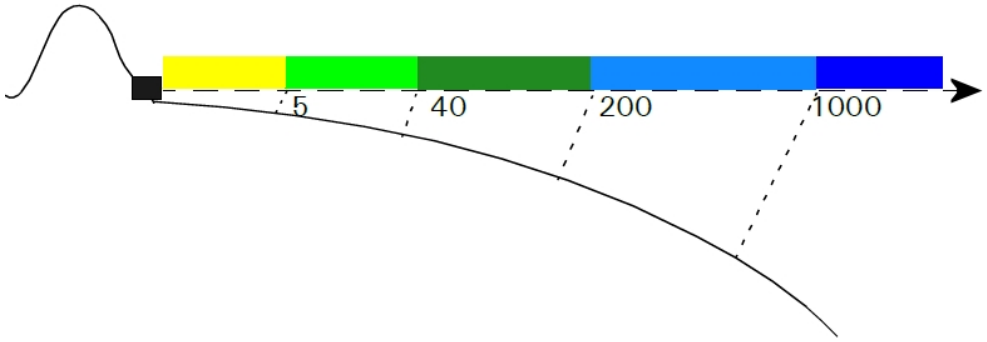


Figure 50 illustrates how the radar coverage display is calculated. Imagine a radar facility at sea level looking out over the ocean (similar to Figure 49). The dotted lines represent the height above sea level of the radar beam as the earth curves (the illustration is not to scale). In the yellow band, the radar can detect all objects that are 5 meters above sea level or higher. In the light green band, the radar can detect all objects that are 40 m above sea level or higher. It might also detect objects between 5 m and 40 m. For each specified intersection height that you want to test, VR-Forces adds another band of color.

Figure 50. Radar coverage



In 3D, the coverage area is depicted as a shaded section of a sphere surrounding the radar. Areas covered by the radar are shaded. Areas that are obscured have no shading. A color gradient is used to show varying altitudes.

You can configure altitude layers and colors on the Display Settings dialog box, Radar Coverage Settings page.

- Radar coverage displays can take some time to calculate, and fill in gradually as the calculation proceeds.
- Because the coverage area is computationally expensive to display, this feature is only usable on units and entities that are stationary.
- Radar coverage ignores vegetation (procedural trees) and structures (buildings and SpeedTrees).

4.7.1 Enabling the Display of Radar Coverage

Radar coverage is enabled or disabled globally per observer. It is displayed per simulation object. For details about displaying radar coverage, see "4.7.2 Displaying Radar Coverage" below.

➤ **To enable the display of radar coverage, use one of the following methods:**

- Choose **Observer > Radar Coverage**.
- On the Observer Toolbar, click the Radar Coverage button ( ).
- On the Display Settings dialog box, Observer Settings page, select the Radar Coverage check box.

4.7.2 Displaying Radar Coverage

To display radar coverage, you must enable it for the observer and then display it for a specific simulation object. For details about enabling radar coverage, see "4.7.1 Enabling the Display of Radar Coverage" above.

➤ **To display radar coverage:**

1. Select a simulation object that has a ground-based radar, such as the Patriot Radar AN/MPQ-65 (entity-level) or Air Defense Radar (aggregate-level).
2. Choose **Objects > Decorations > Pin Radar Coverage**. A radar coverage display is added, centered on the selected simulation object.

Removing Radar Coverage Pins

➤ **To remove the radar coverage pin:**

1. Select a simulation object that has radar coverage pinned.
2. Choose **Objects > Decorations > Pin Radar Coverage**. The radar coverage display is removed.

➤ To remove radar coverage pins for all entities, choose **Objects > Decorations > Remove All Pin Radar Coverage**.

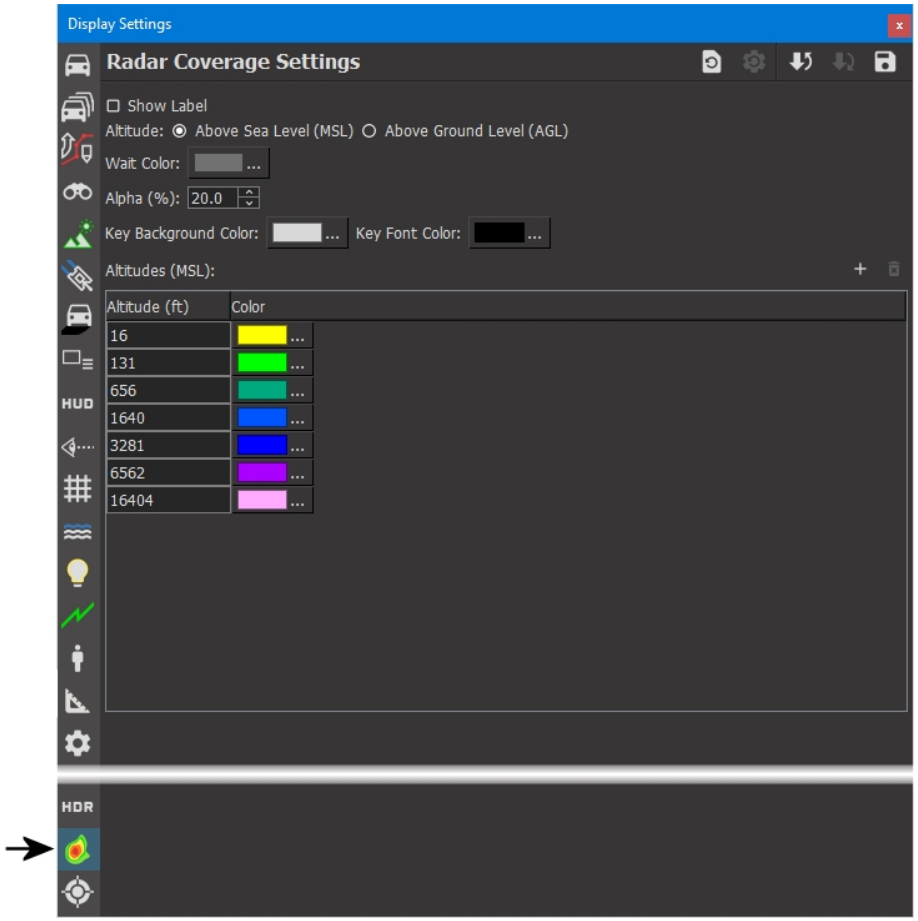
4.7.3 Configuring Radar Coverage Displays

You can configure the colors used for radar coverage displays, the altitude at which colors change, and other aspects of the display.

➤ **To configure radar coverage:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Radar Coverage Settings page (Figure 51).

Figure 51. Radar Coverage Settings page



3. Change the settings as desired. Table 5 describes the settings.

Table 5. Radar coverage settings

Option	Description
Show Label	Display the name of the radar coverage area.
Altitude	Calculate altitude above sea level or above ground level.
Wait Color	The color to use for the radar coverage fan while the GUI waits for results from the sim. engine.
Alpha	The amount of transparency for the coverage areas.
Key Background Color	The background color to use for the altitude key.
Key Font Color	The color to use for the text on the altitude key.
Altitudes	The altitudes to test for intersections and their associated colors.

4.8. Using Sonar

VR-Forces simulates active and passive sonar. Entities can be configured with three types of sonar sensor systems: Active Sonar, Passive Sonar, and Sonar. The Sonar system includes both active and passive sonar. Both types of sonar use the VR-Forces sensor signature mechanism. The signatures are different, reflecting the different chances of detecting entities using the two types of sonar.

Passive sonar has these characteristics:

- Sonar can be dipped at a depth other than that of the entity.
- If an entity is moving faster than a specified speed, passive sonar does not work.
- Sonar takes into account thermocline data. Thermoclines are layers of water that have different temperatures and, therefore, different density. Sound transmission varies as it passes through the thermocline. In VR-Forces sonar effectiveness is reduced as it passes through thermocline boundaries. For more information, see "6.9. Setting the Thermocline" on page 120.
- Passive sonar ignores line-of-sight. It does not take into account underwater terrain or the presence of islands.
- If an entity is publishing propulsion noise, it can affect its detectability by passive sonar.

Active sonar has these characteristics:

- Active sonar should acquire targets more quickly than passive sonar.
- If an entity has an active sonar system enabled, it is more easily detected by passive sonar systems.
- Entities with an active sonar that is enabled publish Underwater Acoustics PDUs.
- Entities can have multiple active sonar modes, similar to radar modes, using emitter beams. For details about configuring emitters, see "Configuring Emitters".
- Active sonar detects targets within the range of its emitter beams, but without regard to the direction of the beams.
- Active sonar ignores propulsion noise and the presence of active sonar on the target.

Sonar only works in water. Therefore rotary-wing entities with sonar systems must set their depth in order to sense anything.

4.8.1 Propulsion Noise and Sonar

If an entity is generating propulsion noise, it is easier to detect using passive sonar. VR-Forces publishes propulsion noise for surface and subsurface entities. If the power plant for an entity is turned off, it does not publish propulsion noise. To turn propulsion noise on or off, change the power plant appearance (for details, see "Appearance") or set the **power-plant-active** parameter in the propulsion-noise-generator component of the entity's movement system.

VR-Forces publishes three propulsion noise values:

- Current Shaft RPM, calculated as **current speed * speed-to-rpm-factor**.
- Ordered Shaft RPM, calculated as **ordered speed * speed-to-rpm-factor**.
- Shaft RPM Rate of Change, calculated as **current acceleration * acceleration-to-rpm-factor**.

The values for current speed, ordered speed, and current acceleration are the current values for the entity. The values for speed-to-rpm-factor and acceleration-to-rpm-factor are taken from the **speed-to-rpm-factor** and **acceleration-to-rpm-factor** parameters in the propulsion-noise-generator component of the entity's movement system.

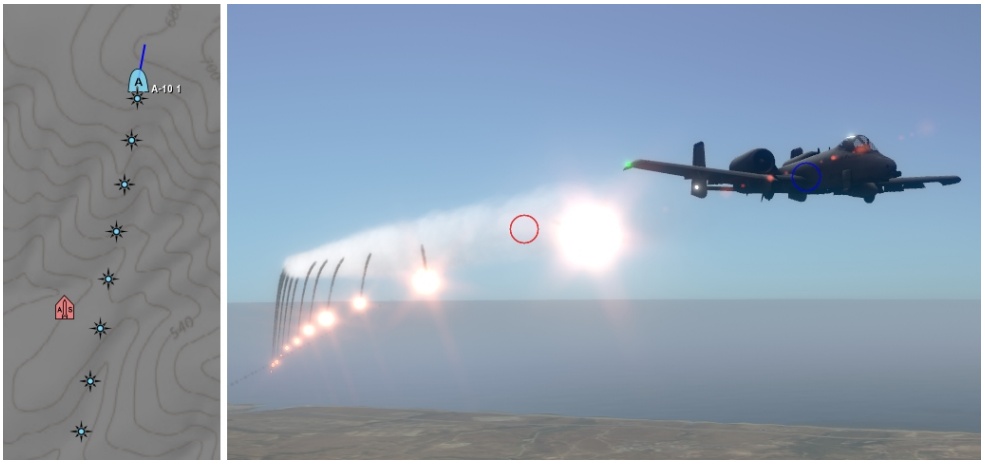
When a target is being detected by passive sonar, if it is publishing propulsion noise, its calculated RPM is compared to a nominal RPM (default 100) to generate a sensor signature modifier. For example, if an entity is publishing an RPM of 80, its signature is scaled by 0.8. If its RPM is 150, the signature is scaled by 1.5.

4.9. Launching Countermeasures (Chaff and Flare)

Fixed-wing and rotary-wing entities can launch countermeasures (chaff and flare) when they are targeted by air-to-air and ground-to-air missiles. By default, entities launch countermeasures automatically based on settings in the *countermeasures.sysdef* file. When an aircraft detects a hostile missile within a specified range, it determines the type of missile and looks up the type of countermeasures to use in the *USCounterMeasuresTable.asl* or *CISCounterMeasuresTable.asl* table.

Figure 52 shows a friendly force fighter launching countermeasures against a sidewinder missile.

Figure 52. Countermeasures (flare)



You can disable automatic countermeasures with the Countermeasures Auto Launch set data request. You can also launch countermeasures with the Launch Countermeasures task. The Launch Countermeasures task is not an exclusive task, so it does not interfere with the entity's current task. For details about the task and set data request, see "Launch Countermeasures" and "Countermeasures Auto Launch".

You can configure the behavior of automatic countermeasures in the *counter-measures-launcher.sysdef* and *cis-counter-measures-launcher.sysdef* files (in *./data/simulationModelSets/EntityLevel/vrfSim/systems/other*). Table 6 describes the parameters that you might want to change.

Table 6. Countermeasure launcher parameters

Parameter	Description
auto-launch-enabled True	Enables or disables automatic launching of countermeasures.
auto-launch-range	The range within which an entity will detect missiles against which to launch countermeasures.
auto-launch-reset-time	The amount of time, in seconds, an entity will wait before launching additional countermeasures.
auto-launch-default-resource	The default countermeasure to use.
auto-launch-number-of-resources	The number of countermeasures to launch for each launch sequence.
counter-measures-select-table-file	The countermeasures selection table to use to determine which countermeasures to fire for a particular missile.

4.10. Modeling Artillery Munitions

In entity-level scenarios, VR-Forces models artillery in two ways: it models an artillery entity that you have tasked to fire, or it models an indirect fire event in an area you specify. When you create a task directing a weapon such as a howitzer to fire munitions, VR-Forces models both the artillery entity and the munitions it fires. Indirect fire events simply model artillery barrages within an area, not the artillery entities or munitions. For details about indirect fire events, see "Chapter 5. Indirect Fire, Ballistic Missiles, and Munition Detonations" on page 87.

By default, when an artillery entity fires, VR-Forces models the munition. Munitions are not affected by wind speed or friction. VR-Forces can also model artillery without modeling the munition. In this case, it just publishes a fire event and detonation event. Modeling munitions requires more computer resources than not doing so. Therefore, in large simulations you may want to disable modeling of munitions.

To disable modeling munitions, in the indirect-fire-system, set the **simulate-munition** parameter to false for each weapon system by editing the sysdef file.

5. Indirect Fire, Ballistic Missiles, and Munition Detonations

This chapter explains how to create and edit indirect fire events and ballistic missiles. These features are only supported in entity-level scenarios.

5.1. Introduction to Indirect Fire

Indirect fire events allow you to create, configure, and schedule indirect fire. Indirect fire events are not associated with a particular simulation object such as a howitzer. VR-Forces does not model the launcher or trajectory of the munitions. Indirect fire results in one or more salvos of one or more rounds in or near a designated area of the terrain. Figure 53 illustrates indirect fire within an indirect fire area.

Figure 53. Indirect fire event

Indirect fire types are configured in *./data/simulationModelSets/EntityLevel/indirectArtilleryTypes.mtl*.

ⓘ Indirect fire events, which are area-based, are not supported in aggregate-level scenarios. However, there is an indirect fire task available to artillery entities in aggregate scenarios; see "Indirect Fire" for details.

5.1.1 Creating an Indirect Fire Event

When you create an indirect fire event, you specify an area in which the event is to take place, the parameters of the event, and a starting time. The area in which an indirect fire event takes place is a tactical graphic and is listed on the Objects List View of the Objects List Panel.

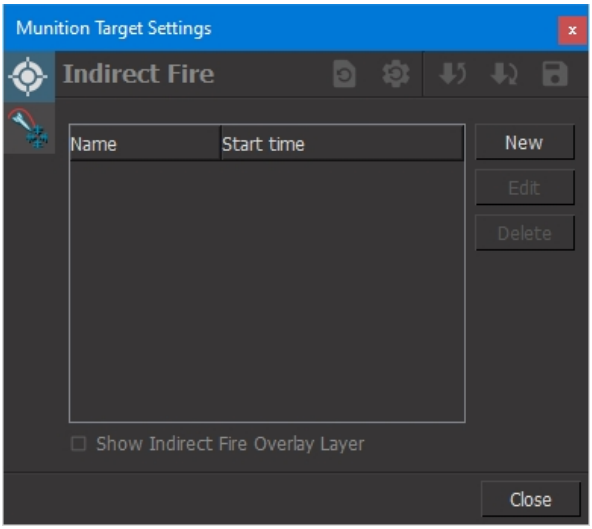
By default, indirect fire areas are placed on the Indirect Fire layer (which is created for the first indirect fire object). (For information about overlays, see "Overlays".)

You can activate and deactivate indirect fire areas; see "Active and Inactive Tactical Graphics" for details.

➤ **To create an indirect fire event:**

1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens (Figure 54).

Figure 54. Munition Target Settings dialog box, Indirect Fire page



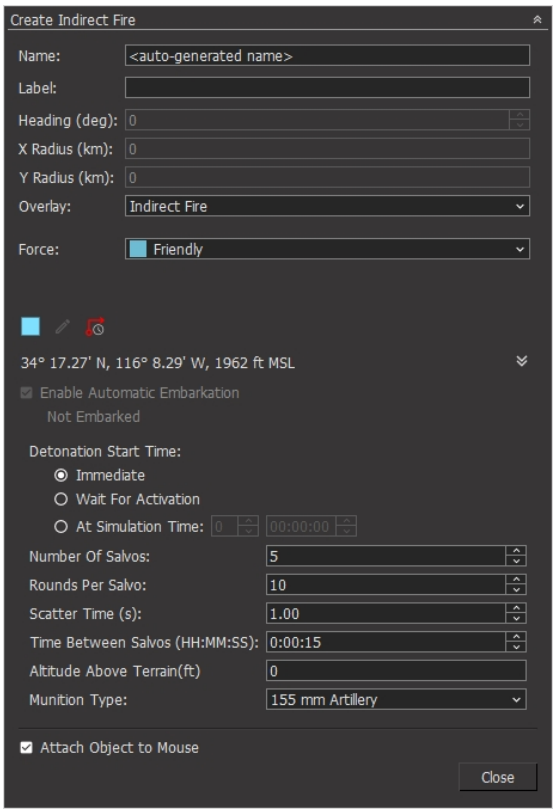
2. Select the Indirect Fire page.
3. Click New. The cursor changes to create mode and a tab for the Create Indirect Fire dialog box is added to the window.
4. Draw an ellipse on the map to specify the area of the fire event:
   a. Click to set the center of the ellipse. A center point is set and a small circle is displayed with a vertex indicator at the bottom of the circle.
   b. Drag the mouse up or down and then click to set the height of the ellipse. A vertex indicator is added to the left side of the circle.
   c. Drag the mouse left or right and then click to set the width of the ellipse (Figure 55).

   Optionally, you can specify the properties of the ellipse in the Create Indirect Fire dialog box.

Figure 55. Indirect fire event area

5. Click the tab to display the Create Indirect Fire dialog box (Figure 56).

Figure 56. Create Indirect Fire dialog box

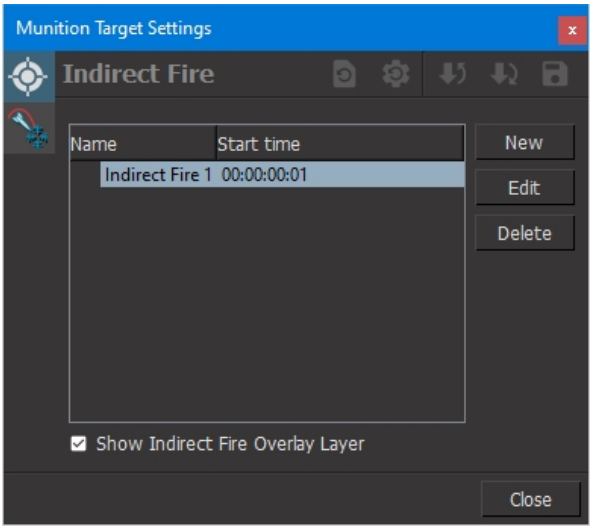


6. Enter a name for the event in the Name text box, or accept the default name (Indirect Fire *n*).
7. Optionally, specify an overlay layer for the event. (If there are no layers in the scenario, one is created when you click Create to create this indirect fire event.)
8. Configure the parameters of the fire event:

- Select the detonation start time. The options are:
  - Immediate. Indirect fire begins immediately.
  - Wait for Activation. The indirect fire area is created in the inactive state. Indirect fire begins when an Activation set data request activates the area. This allows you to activate indirect fire manually or from a plan.
  - At Simulation Time. The simulation time at which you want the munition to start firing.
- In the Number of Salvos box, enter the number of salvos you want to fire.
- In the Rounds Per Salvo box, enter the number of rounds you want to fire for each salvo.
- In the Scatter Time box, enter the length of time for each salvo. The rounds in each salvo will be detonated at random times within this time period.
- If you specified more than one salvo, in the Time Between Salvos box, enter the amount of time you want to pass between each salvo.
- Optionally, specify an altitude above the terrain at which the rounds should detonate.
- Select the munition type in the Munition Type list.

9. Click Create. The Indirect Fire event is added to the list (Figure 57). The area is added as an object on the Indirect Fire layer on the Overlays View of the Objects List Panel (unless you chose a different layer).

Figure 57. Indirect Fire page with indirect fire event



5.1.2 Editing Indirect Fire Events

You can edit the properties of an indirect fire event. You can also directly manipulate the area that specifies the event.

➤ **To edit an indirect fire event:**

1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens.
2. Select the Indirect Fire page (Figure 57 on the previous page).
3. Select the indirect fire event that you want to edit.
4. Click Edit. The Edit Indirect Fire dialog box opens.
5. Change the properties that you want to change.
6. Click OK.

5.1.3  Deleting Indirect Fire Events

➤ **To delete an indirect fire event:**

1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens (Figure 54 on page 89).
2. Select the Indirect Fire page (Figure 57 on the previous page).
3. Select the indirect fire event that you want to delete.
4. Click Delete.
5. Click Close.

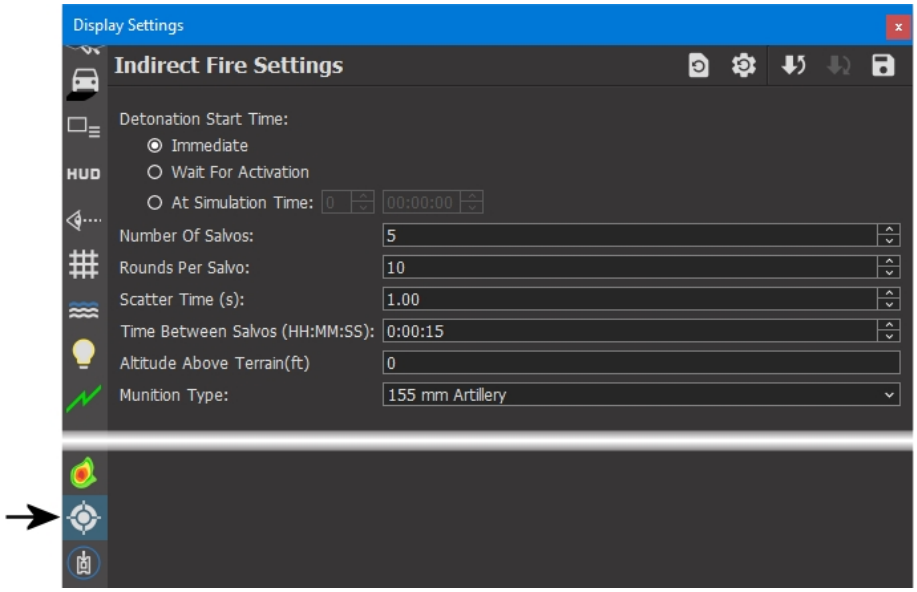
5.1.4  Configuring Indirect Fire Event Default Values

You can configure a set of default values that are applied to the Create Indirect Fire dialog box.

➤ **To configure default values for indirect fire events:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Indirect Fire Settings page (Figure 58).

Figure 58. Indirect Fire Settings page



3. Set the parameters to the desired default values. Most are self-explanatory. The Add Detonations if Missing check box causes VR-Forces to map a detonation to a detonation graphic if there are no mappings for that detonation event in the Detonation Mapping Settings page of the Simulation Object Type Mappings dialog box. If you do not want a detonation displayed for detonation events that are not mapped, clear this check box.
4. Click Close.

5.2. Ballistic Missiles

VR-Forces supports two types of ballistic missiles:

- Strategic missile weapon systems. Entities with strategic missile weapon systems can fire at targets using Launch Ballistic Missile tasks.
- Fixed trajectory ballistic missiles. The trajectory of a missile is determined by a series of coordinates in a comma-separated values (CSV) file. VR-Forces includes some sample CSV files. VR-Forces users are responsible for creating their own files if they want missiles to fly with different characteristics.

Strategic missile systems calculate a trajectory based on a maximum altitude and the distance to the target. They attempt to accurately represent the speed of the missile and they can take into account the curvature of the earth. (For information about strategic missile launch tasks, see "Launch Ballistic Missile at Location" and "Launch Ballistic Missile at Target Object".)

> The Ballistic Missile tasks do not use physics-based modeling. They calculate a simple parabola for the trajectory and move the missile along the trajectory ignoring atmospherics, thrust stages, cargo weight, gravity, and the rotation of the earth.

You fire a fixed trajectory ballistic missile by specifying a target and choosing a missile type. The characteristics of the missile type are specified in the Simulation Object Editor. A missile launches at the same altitude as the target point at a distance determined by the CSV file. Therefore, if a target point is placed at or near the ground, it is possible that due to variations in the terrain, the launch location could be underground. Once the missile launches, it follows the trajectory contained in the CSV file that it is configured with. It detonates when it reaches the target point. For details about configuring missiles, see "Configuring Ballistic Missile Movement".

This remainder of this section describes how to fire fixed trajectory ballistic missiles.

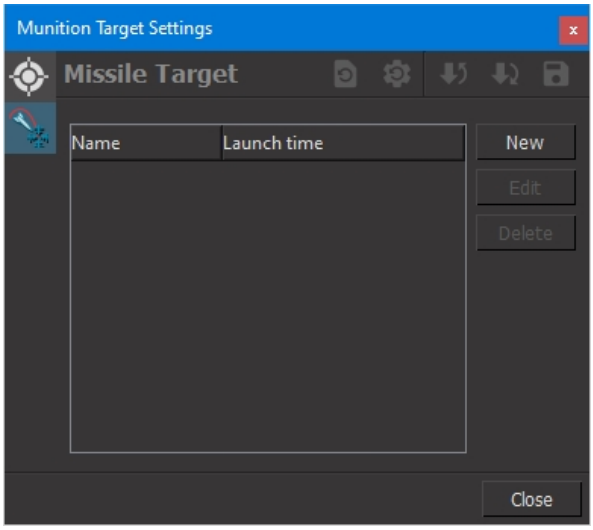
> ⓘ Ballistic missiles are not supported in aggregate-level scenarios.

5.2.1 Firing Ballistic Missiles

➤ **To fire a ballistic missile:**

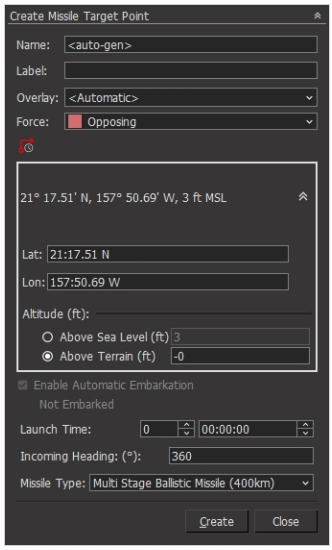
1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens.
2. Select the Missile Target page (Figure 59).

Figure 59. Munition Target Settings dialog box, Missile Target page



3. Click New. The cursor changes to cross-hairs.
4. Click the location where you want the missile to hit. The Create Missile Target dialog box opens (Figure 60).

Figure 60. Create Missile Target dialog box



5. Optionally, specify the parameters for the target. In particular, set these parameters:
   - Name. The name of the target point.
   - Launch Time. The simulation time at which the missile will be launched.
   - Incoming Heading. The heading, relative to the target, from which the missile will be launched.
   - Missile Type. An option from the list.
6. Click Create. The missile event is listed in the Missile Target window.

5.2.2 Editing Missile Target Events

You can edit the properties of a missile target event. You can also directly manipulate the target point.

➤ **To edit a missile target event:**

1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens (Figure 54 on page 89).
2. Select the Missile Target page (Figure 60 above).
3. Select the missile fire event that you want to edit.
4. Click Edit. The Edit Missile Target dialog box opens.
5. Change the properties that you want to change.
6. Click Update.

5.2.3 Deleting Missile Target Events

➤ **To delete a missile target event:**

1. Choose **Simulation > Munition Targets**. The Munition Target Settings dialog box opens (Figure 54 on page 89).
2. Select the Missile Target page (Figure 60 on the previous page).
3. Select the missile fire event that you want to delete.
4. Click Delete.
5. Click Close.

ⓘ You can also delete a target event by deleting the target point from the terrain.

5.3. Creating Munition Detonations

When weapons fire munitions, they detonate and cause damage to simulation objects, terrain, or both. You can also manually create munition detonations. They have the same effect as the simulated munitions they represent.

ⓘ Munition detonations are not supported in aggregate-level scenarios.

➤ **To create a munition detonation:**

1. Choose **Create > Munition Detonation**. The Create Detonation dialog box opens, as if you were creating a simulation object or tactical graphic. The cursor changes to red cross-hairs.
2. If you want to use the currently configured detonation type and result, click on the terrain where you want the detonation to occur. You can continue to click to create more detonations while in this mode.
3. If you want to change the munition type or result:
    a. In the Munition Type list, select the type of munition you want to detonate.
    b. In the Result list, select the type of detonation that you want to occur.
    c. Optionally, specify the location for the detonation.
    d. Click on the terrain where you want the detonation to occur. Click as many times as you want.
4. To exit detonation mode, close the Create Detonation dialog box or press **Esc**.

6. Setting Environment Conditions

This chapter explains how to set the time of day and weather conditions.

> ⓘ If you want environment conditions in different applications or instances to look the same, use the command-line option (`-O | --hasNTPSync`). This enables the synchronization of all time-based features between different VR-Vantage applications, such as instances of VR-Vantage or the VR-Forces GUI. Note that system clocks must be synchronized.

6.1. Introduction to Environment Conditions

You can set the date, time of day, weather, and marine conditions from the GUI and in plans. The date, time of day, and weather conditions are sim. engine settings, not merely GUI display effects. They affect simulation object sensors and line-of-sight. For example, if visibility is set to 4 km, a simulation object will not see a simulation object that is 5 km away. If the time of day changes from daytime to nighttime, lighting conditions in the 3D view change accordingly.

Date and Time and Weather are options on the Commands menu in plans.

6.2. Setting Weather Conditions

You can set these weather conditions:

- Cloud cover
- Visibility
- Precipitation
- Wind speed and direction
- Ambient air temperature

When you specify weather, the sim. engine sends out state updates with these conditions. Simulation objects respond to them only if their model has the ability to do so.

6.2.1 Setting the Visibility, Wind Direction and Speed

You can set atmospheric conditions, including wind speed and direction, visibility, obscurant (such as fog, smoke, or dust), cloud types, and sky coverage. If you do not specify the sea state manually, it is set based on the wind conditions. (The maximum wind speed that you can set is 360 km/h.) The wind direction affects the angle at which precipitation falls. Wind speed and direction also affect SpeedTrees, grass, and particle systems (dust clouds, smoke, and so on).

ⓘ Local weather zones are not affected by global wind speed. They use the wind speed and direction specified in the area's Weather settings.

> The arrow in the dial indicating the wind direction points in the direction that the wind is moving. For example, if the wind direction is set to 180 degrees, the arrow points up to show that the wind is moving from the south to the north.

On the Weather page, you can specify different wind speeds and directions, visibility, obscurants, and cloud characteristics at various altitudes. This allows for a more realistic simulation for aircraft and simulation objects with parachutes. You can create up to ten levels of atmospheric configurations.

For information on setting the color of ground-level fog, see "Configuring Ground Fog" on page 101.

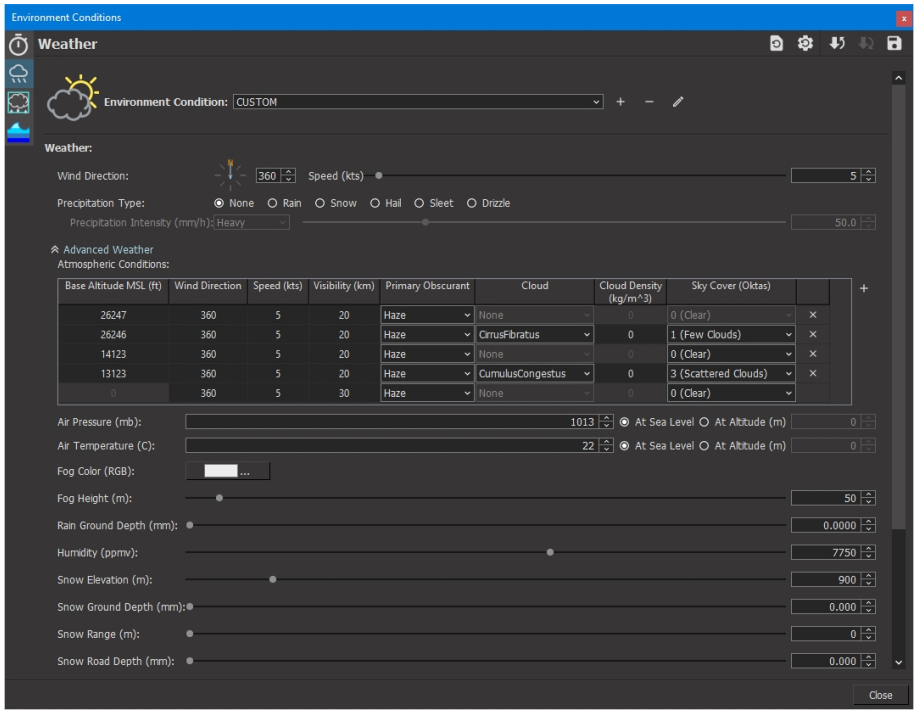
The highest elevation in the Atmospheric Conditions table (under Advanced Weather) is automatically configured with a Sky Cover value of 0 and no clouds. This elevation's Base Altitude MSL value provides an effective ceiling for the clouds at the elevation immediately underneath it.

If you set Visibility to a large value, you might need to adjust the value of the channel's Z far attribute (also known as the far clipping plane). If the Z far attribute is set to a value smaller than the Visibility value, objects beyond the Z far distance are not visible in the channel. For details, see "Setting the Clipping Planes".

➤ **To set wind, visibility, and other elevation-based atmospheric conditions:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61).

Figure 61. Weather page



3. Click Advanced Weather to expose the Atmospheric Conditions table.
4. Set the atmospheric conditions for one or more levels of elevation:
    a. Select an environment condition from the Environment Conditions list, or click the Add button (+) to create a new environment condition.

       The lowest elevational configuration in the Atmospheric Conditions table begins at 0 feet above mean sea level (MSL). You can specify the base of each subsequent elevation.
    b. Use the Wind Direction dial in the Weather section of the page to enter a Wind Direction heading between 001 and 360 degrees.
    c. Drag the Speed slider to set the wind's speed.

       The wind heading and speed you entered appear in the Atmospheric Conditions table.
5. Set the visibility, obscurant, and sky cover for one or more levels in the Atmospheric Conditions table:
    a. Select Visibility in a row in the table or click the Add button (+) to the right of the table to create a new row.
    b. Enter a Visibility distance.
    c. If visibility is limited, choose the Primary Obscurant.
    d. Set the Sky Cover.
    e. If you set Sky Cover to a nonzero value, you activate the Cloud column. Choose a cloud type.
    f. If you set Visibility to a high value, you might need to change the channel's Z far attribute value — it must be higher than the Visibility value. Z far defaults to 200 kilometers. For details on changing Z far, see "Setting the Clipping Planes".

6.2.2 Setting the Air Pressure and Temperature

You can specify the air pressure and temperature at any altitude, including sea level. When the air temperature is below freezing, the ground simulates freezing and affects ground vehicle mobility. VR-Forces does not do anything with the air pressure information. It is sent over the network for the use of applications that need to model air pressure.

➤ **To set the ambient air temperature:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61 on the previous page).
3. In the Weather section of the page, click Advanced Weather. The dialog box expands to show more options.
4. In the Air Pressure field, set the air pressure in millibars.
5. Select At Sea Level or At Altitude.

6. If you selected At Altitude, set the altitude in meters for the new air pressure. (Setting an altitude of 0 has the same effect as selecting At Sea Level.)
7. In the Air Temperature field, set air temperature in degrees Celsius.
8. Select At Sea Level or At Altitude.
9. If you selected At Altitude, set the altitude in meters for the new air temperature. (Setting an altitude of 0 has the same effect as selecting At Sea Level.)

6.2.3 Setting Visibility

The visibility setting determines the denseness of obscurants in the atmosphere (fog, smoke, or dust, for example), which determines how far you can see.

➤ **To set visibility:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61 on page 99).
3. In the Atmospheric Conditions table, change the value in the Visibility column.
4. If visibility is limited, choose a Primary Obscurant.
5. If you set Visibility to a large value, you might need to adjust the value of the channel's Z far attribute (also known as the far clipping plane). If the Z far attribute is set to a value smaller than the Visibility value, objects beyond the Z far distance are not visible in the channel. For details, see "Setting the Clipping Planes".
6. To change visibility at a different elevation, click the Add button (+) and then specify the values for that elevation.

Configuring Ground Fog

You can set the height and color of ground-level fog. The fog height affects how much sky the observer can see when looking up from ground level. For example, if you set the fog height to 50m and look up, you will see some amount of sky above you, and to the sides it will be very gray. If you then set it to 10m and look up, you will see mostly blue sky above you, but still quite a bit of gray to the sides.

These steps create ground-based fog and let you choose the fog's color. To configure fog at higher elevations (or other obscurants like smoke or dust), see "6.2.3 Setting Visibility" above. To configure fog (or other obscurants) that's confined to a limited area, see "6.6. Using Weather Simulation Objects" on page 112.

> ⓘ Fog settings are not modeled in the sim. engine. Therefore, they are not available when you set weather conditions in a plan.

➤ **To configure fog:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61 on page 99).

3. In the Weather section of the page, click Advanced Weather. The dialog box expands to show more options.
4. To set the fog height, adjust the slider or enter a value in the box.
5. To set the fog color:
    a. Click the color swatch. A color picker dialog box opens.
    b. Select the color you want to apply to the fog. It is applied immediately, so you can easily see the effect of the new color and change it if you want to.
    c. Click OK.

6.2.4 Specifying Precipitation Type and Intensity

VR-Forces supports display of rain, snow, hail, sleet, and drizzle. You can configure the intensity of precipitation.

> ⓘ Precipitation has no effect on the simulation. It is only a visual effect. Snow does not accumulate.

➤ **To specify the precipitation type and intensity:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61 on page 99).
3. Select a Precipitation Type. Once you select a type other than none, the Precipitation Intensity setting becomes editable.
4. Adjust the Precipitation Intensity slider or enter a value in the box. The range is 0 through 200 mm per hour. The intensities list updates based on the rate you specify. You can also choose one of the preset intensities from the list:
    — Light – 1.2 mm per hour.
    — Moderate – 6.2 mm per hour.
    — Heavy – 30 mm per hour.
    — Extreme – 125 mm per hour.

6.2.5 Setting Rain Accumulation

You can specify the accumulation of rain on the terrain. When you have rain accumulation, the terrain appears slightly shinier and specular.

You can also specify that puddles should appear in areas of the terrain. If it is raining, ripples appear in the puddles. To define where puddles should accumulate, use a Rain Texture map. See "Using Shader-based Effect Maps" in *Adding Content to MAK ONE Applications Reference Manual*.

➤ **To set the rain accumulation:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.

2. Select the Weather page (Figure 61 on page 99).
3. Click Advanced Weather.
4. Adjust the Rain Ground Depth slider or enter a value in the box. This specifies the depth of rain on the ground and other terrain features. The depth specifies the level of specular reflection on the ground and the appearance of puddles.

6.2.6 Setting Snow Depth and Elevation

You can specify the depth of snow on the ground and roads, and you can also specify the elevation for the snow line. Snow can also display on the roofs of buildings.

If appropriate, you can mask out the snow for certain parts of the terrain using a Snow Texture map. You can also use the SNO effect texture or the snow mask effect map layer to have snow drift across the roads when the wind is blowing. See "Using Shader-based Effect Maps" in *Adding Content to MAK ONE Applications Reference Manual*.

➤ **To set the snow depth and minimum elevation:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 61 on page 99).
3. Click Advanced Weather.
4. Adjust the Snow Ground Depth slider or enter a value in the box. This specifies the depth of the snow on the ground and other terrain features. The range is 0.000 through 10,000.000 mm.
5. Adjust the Snow Road Depth slider or enter a value in the box. This specifies the depth of the snow on roads. The range is 0.000 through 10,000.000 mm.
6. Adjust the Snow Elevation slider or enter a value in the box. This specifies the minimum elevation (in meters) where the snow cover appears.
7. If desired, use the Snow Range slider and box to specify the range in elevation at which the snow transitions from no snow to the defined snow depth. Adjust the slider or enter a value.

   For example, if the Snow Elevation is set to 100 m and the range is set to 50 m, there will be no snow at 100 m, half the current snow coverage at 125 m, and full coverage at 150 m.
8. If you want to see snow blowing in the wind, create a Weather tactical graphic and specify the Blowing Snow environmental condition. See "6.5. Creating Local Weather Zones" on page 109.
9. Click Close.

6.3. Setting the Date and Time of Day

You can set the date, time zone, and time of day.

- Date: When you create a scenario, you specify a date. However, you can change the date at any time while you are creating or running a scenario. The date at a location is advanced when the time at that location advances through midnight. When this happens, all locations to the east, up to the time zones at UTC + 12 hours, are in the advanced date; all locations to the west, up to the time zones at UTC -12 hours, are in the previous date. Thus there is effectively a date line at a longitude 12 hours from UTC.
- Format: You can specify whether the time and date should be display in standard format or military format. You can also choose a display format for the date and time, which will be one of a variety of options relevant to either standard or military.
- Time zone: You can change the time zone displayed in the Environmental Settings toolbar to any zone that you want. If you change the time zone in the Environment Conditions dialog box, the time displayed in the Environment Settings Toolbar changes to the current time for that time zone. However, the time at a given location on the earth does not change. The illumination at a location therefore does not change when the time zone displayed in the toolbar changes. Since the time zone at a location may not be the same as the time zone displayed in the toolbar, the illumination at that location may not match the time displayed in the toolbar.
- Time of day: The time of day advances as simulation time advances. However, they are not directly tied to each other. For example, if at the start of a simulation (simulation time 0:00:00:00), you set the time of day to 10:00 AM, after eight hours of simulation time passes, the time of day will be 6:00 PM. However, you could just as easily set the initial time of day to some other time or change it at will during the course of the simulation. These changes would not affect the elapsed simulation time in any way. They would only change the starting point for the advance of the time of day.

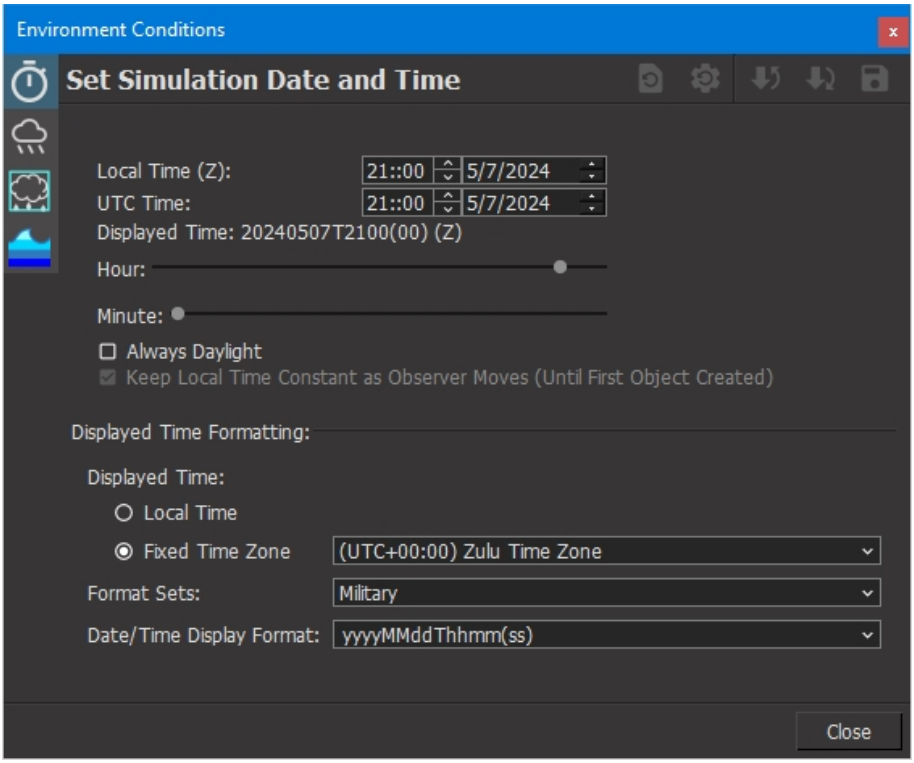
  VR-Forces uses an ephemeris model to calculate illumination based on the date and time of day. The amount of light affects visual sensors and scene lighting in Stealth view mode. Illumination at a location is based on the time of day at that location.

6.3.1 Setting the Date

➤ **To set the date:**

1. Choose **Simulation > Environment Conditions**, or click the date on the Environment Settings Toolbar. The Environment Conditions dialog box opens.
2. Select the Date and Time page (Figure 62).

Figure 62. Date and Time page



3. Place the cursor in either the Local Time or UTC Time box for the date.
4. To change the month, click the right or left arrows, or click the month and select a month from the list.
5. To change the year, click the right or left arrows, or click the year and type in a year or use the spin controls to select a value.
6. To specify the date, click on the date in the calendar.
7. To specify the format, select the desired options from the Formats Set and Date/Time Display Formats lists.

6.3.2 Setting the Time of Day

As time advances, the time displayed on the Environment Settings Toolbar changes. The time of day setting is independent of simulation time. The default time zone is Coordinated Universal Time (UTC, also called Zulu time). You can change the time zone to the local zone or any of the zones in the Fixed Time Zone list.

ⓘ If you want to set the time of day and time zone, set the time zone first.

➤ **To set the time of day:**

1. Choose **Simulation > Environment Conditions**, or click the date on the Environment Settings Toolbar. The Environment Conditions dialog box opens.
2. Select the Date and Time page (Figure 62 above).

3. Set the time of day by changing the value in the Current Time box or by adjusting the Hour and Minute sliders.
4. Select the time zone in one of the following ways:
   - Select a time zone in the Fixed Time Zone list.
   - Select Local Time. The time zone set for your computer is used.
5. In the Formats Set box, specify the format for the time and date by selecting Military or Standard.
6. Optionally, select a date/time display format. This changes the format used in the Environment Settings Toolbar. The available formats are related to the option the in Formats Set box.

6.3.3 Choosing the Illumination Model

Illumination affects visual sensors. Illumination can be on all of the time or it can be affected by the time of day. The illumination setting does not affect the GUI. It only affects sim. engine calculations. If the day/night illumination model is active (Always Daylight is not selected), thesim. engine calculates illumination based on a simulation object's location given current Greenwich Mean Time, offset by the time zone where the simulation object is located (based on longitude). This calculation does not take into account daylight saving time or localized time zone variations.

➤ **To specify the illumination model:**

1. Choose **Simulation > Environment Conditions**, or click the date on the Environment Settings Toolbar. The Environment Conditions dialog box opens.
2. Select the Date and Time page (Figure 62 on the previous page).
3. Select or clear the Always Daylight check box.

6.4. Specifying the Environment Conditions

VR-Forces has a set of preconfigured environment conditions. Each environment condition has settings for:

- Weather:
  - Wind speed and direction.
  - Precipitation type and intensity.
  - Visibility.
  - Primary obscurant (for example, haze, dust or smoke).
  - Cloud type and sky cover.
  - Air pressure.
  - Ambient air temperature.
  - Fog height and color.

  - Rain depth on the ground.
  - Humidity
  - Snow depth on the ground and roads and starting elevation for accumulation.
- Marine:
  - Sea state.
  - Sea swell.
  - Underwater visibility.
  - Choppiness.
  - Depth offset.
  - Surface transparency.
  - Surge depth.
  - Tidal current direction and speed.
  - Wave fetch.

Conditions range from clear to stormy and allow you to quickly set the weather without needing to adjust all the various weather parameters. If you want to, however, you can change any of the individual weather parameters. Changing a weather or marine setting does not affect the saved settings unless you specifically overwrite them.

You can save and import environment settings. Environment settings support the standard VR-Forces settings behaviors, as described in "Managing VR-Forces Settings".

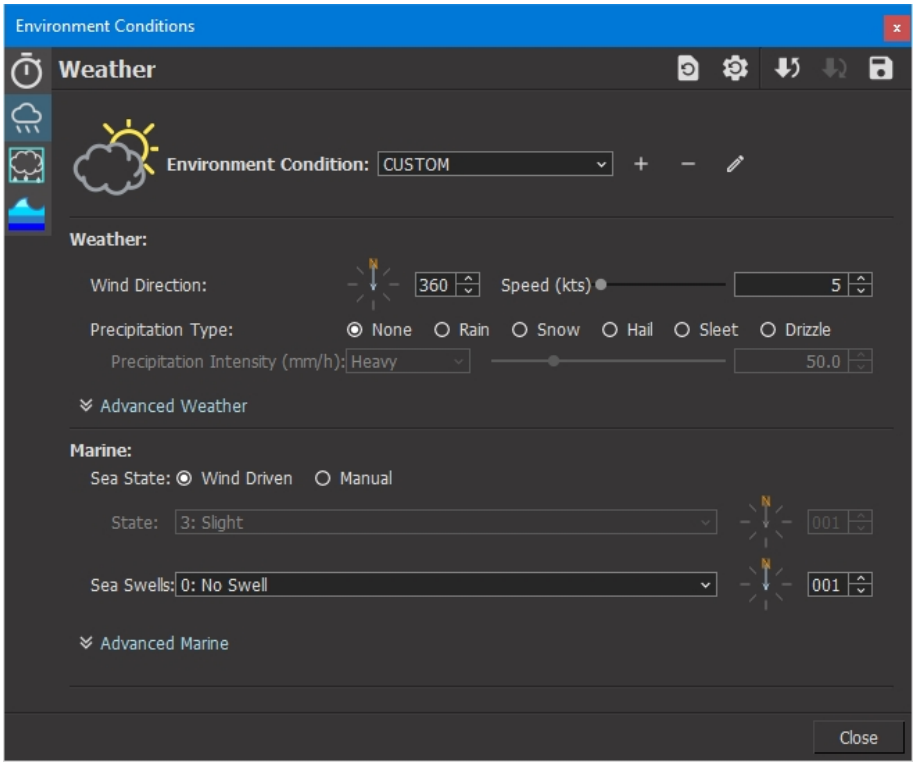
You can add your own environment conditions.

> The settings on theWeather page always reflect the current settings for environment conditions. They do not necessarily match any of the preconfigured environment conditions as listed in the Apply Environment Conditions list except immediately after a preconfigured environment condition is applied. In other words, the Apply Environment Conditions list lets you apply environment conditions, but is not intended to summarize the status of the current environment conditions.

➤ **To specify an environment condition:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 63).

Figure 63. Weather page



3. In the Apply Environment Condition list, select the condition you want.

> ⓘ After you select a preconfigured environment condition, if you change any of the settings, the Apply Environment Condition list reads CUSTOM instead of the condition that you chose.

6.4.1 Adding an Environment Condition

If you add an environment condition, it is added to the Apply Environment Condition list.

➤ **To add an environment condition:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 63 above).
3. Change the weather settings to the ones you want for the new condition.
4. Click the Add button (+). The Name an Environment Condition dialog box opens.
5. Type a name for the new condition.
6. Click OK.

6.4.2 Editing an Environment Condition

You can change the settings for an environment condition. Editing a condition essentially means overwriting a condition with the current set of weather and marine conditions.

➤ **To edit an environment condition:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 63 on the previous page).
3. Optionally, in the Apply Environment Condition list, select a condition you want to use as your starting point.
4. Change whatever conditions you want.
5. Click the Edit button ( ). The Select an Environment Condition dialog box opens.
6. In the Select Name list, select the condition you want to overwrite.
7. Click OK.

6.4.3 Deleting an Environment Condition

You can delete any defined environment conditions. If you delete one of the built-in environment conditions, you can restore it from the factory settings.

➤ **To delete an environment condition:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 63 on the previous page).
3. Click the Delete button ( ). The Select an Environment Condition dialog box opens.
4. In the Select Name list, select the condition you want to delete.
5. Click OK.

6.5. Creating Local Weather Zones

You can create local weather zones. Each area can have different weather conditions—for example, rainy, clear, or overcast, with different wind and visibility conditions. Simulation objects can respond to those conditions as they move from one area to another.

Weather zones can be boxes, free-form areas, circles, and ellipses. You can create them from the Create Object Palette just as you would create the comparable tactical graphics. You can also create box weather zones from the Environment Conditions dialog box on the Local Weather Zones page.

Weather zones have a priority. If two areas overlap and one has a higher priority than the other, a simulation object in the overlapping area responds to the weather conditions in the higher priority area. If overlapping areas have the same priority, the one that is discovered first on the network is used.

By default, weather zones are placed on the Weather overlay. Weather zones display a graphic that represents the weather conditions. Figure 64 illustrates a weather zone set to the Ground Fog environmental condition. You can move and resize weather zones just as you can move and resize other tactical graphics.

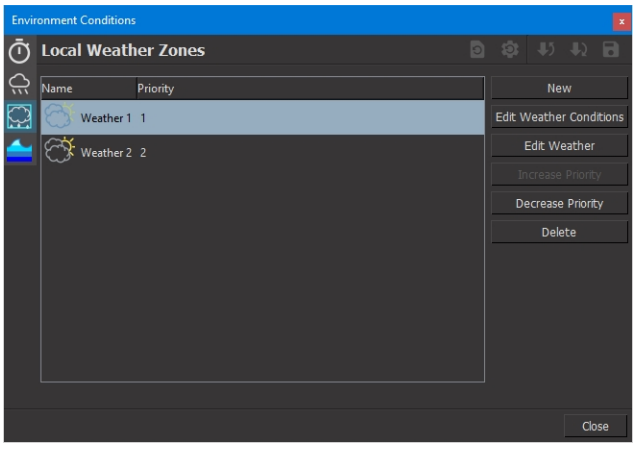
Weather zones are used by the simulation engine and affect simulation objects in the scenario.

Figure 64. Weather (Area) zone - 2D and 3D

➤ **To create a local weather zone from the Local Weather Zones page:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Local Weather Zones page (Figure 65).

Figure 65. Local Weather Zones page



3. Click New. A weather zone box appears in the scenario.
4. Click on the terrain to place the lower left corner of the weather zone. The Create Weather dialog box opens.

The default size for a weather zone is 100,000 meters per side, so the zone might entirely cover small terrains. If this is a problem, edit the Length and Width in the Create Weather dialog box to reduce it to a manageable size.

5. Optionally, edit the parameters for the weather zone. In particular, select an environmental condition or edit the environment to specify the conditions you want to apply to this weather zone. For details about editing the environment, see "6.5.2 Editing a Weather Zone's Environment" below.
6. Click Create.
7. On the Local Weather Zones page, select a weather zone and use the Edit Weather Conditions or Edit Weather button to alter the zone's parameters.
8. When weather zones overlap, use the Increase Priority and Decrease Priority buttons to change the priority of the selected weather zone.

6.5.1 Creating a Weather Zone Simulation Graphic

You can create all types of weather zones from the Create Object Palette. Weather zones create effects such as rain, fog, snow, and blowing dust or sandstorms. Precipitation and the Visibility setting in the Atmospheric Conditions affect line of sight because they reduce the apparent signature of entities, thereby reducing the ability of sensors to detect entities. Other effects are not modeled in the sim. engine, therefore they do not affect line of sight.

The weather zones are listed as:

- Weather (Area). A free-form shape—you determine the number of sides.
- Weather (Box). The same weather zone you can create from the Local Weather Zones page.
- Weather (Circle).
- Weather (Ellipse).

The procedures for drawing them are the same as for other similarly shaped graphics, as described in "Placing Objects", "Drawing Circles and Ellipses", and "Drawing Boxes". The main difference is that when you create a weather zone, the Edit dialog box always opens to allow you to specify the weather-specific parameters.

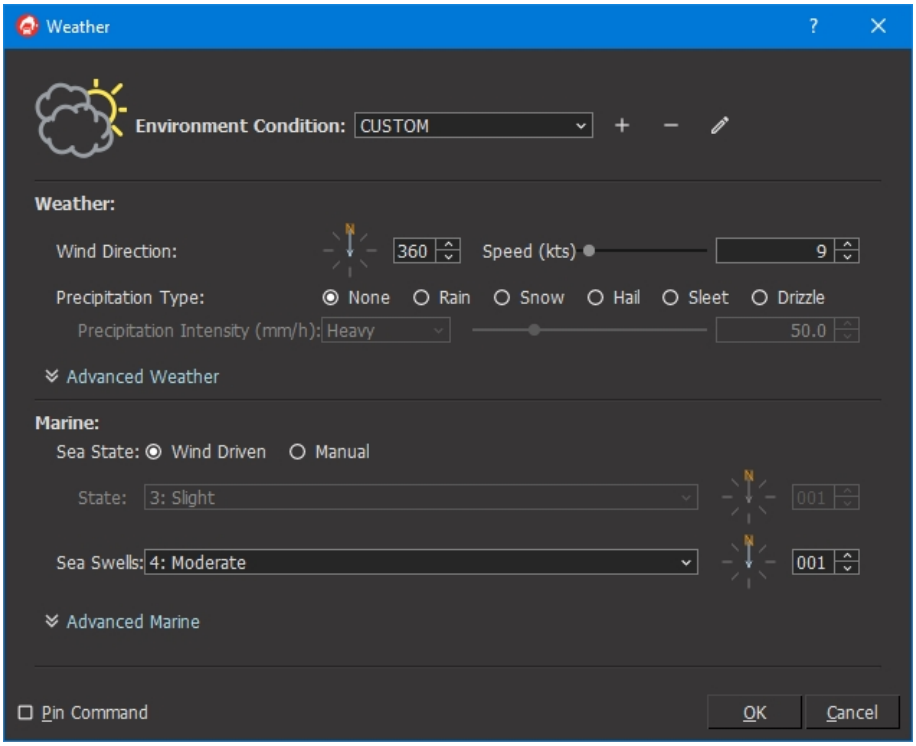
6.5.2 Editing a Weather Zone's Environment

You can edit a weather zone's environment when you create it or after you create it. You can assign it one of the preconfigured environmental conditions or change individual environment settings. You can edit all types of weather zones from the Local Weather Zones page. It does not matter how you created them. For details about preconfigured environmental conditions, see "6.4. Specifying the Environment Conditions" on page 106.

➤ **To edit a weather zone's environment:**

1. Do one of the following:
   - In the Create Weather dialog box, click Edit Environment. The Weather dialog box opens (Figure 66).
   - On the Local Weather Zones page of the Environment Conditions dialog box, select the weather zone you want to edit and then click Edit Weather Conditions. The Weather dialog box opens.

Figure 66. Set Weather dialog box



2. Change weather conditions as desired. They are the same as those on the Weather page of the Environment Conditions dialog box. For details, see "6.2. Setting Weather Conditions" on page 98.
3. Click OK.

6.6. Using Weather Simulation Objects

Weather zones, wind corridors, and air turbulence can affect simulation objects. You can also use weather zones for the Entity In Area condition in plans. VR-Forces publishes weather details and VR-Vantage applications can use that information to display them.

Weather tactical graphics include:

- Weather zones (Area, Box, Circle, Ellipse). Areal graphics that visualize localized weather, as described in "6.5. Creating Local Weather Zones" on page 109.

- Wind Corridor. A linear graphic that affects aircraft. See "6.6.1 Creating a Wind Corridor" below.
- Air turbulence. An area that publishes air turbulence to the network. For details, see "6.6.2 Creating an Air Turbulence Area" below.

6.6.1 Creating a Wind Corridor

Wind corridors are linear tactical graphics that specify a directional wind that aircraft can move through or along like a route. You can use this to represent an area where the wind is blowing at a different speed than elsewhere. For example, the wind in a valley might move at a different speed than the wind across the mountains. A wind corridor affects aircraft with either a tailwind or headwind, increasing or decreasing the true airspeed.

As with other linear tactical graphics, wind corridors have a direction. The wind direction starts at the first point you draw and moves to the final point.

➤ **To create a wind corridor:**

1. Open the Create Object Palette.
2. Optionally, click the Weather filter ( ).
3. Select Wind Corridor.
4. Click to place the vertices of the wind corridor. Double-click when you place the final vertex. The Create Wind Corridor dialog box appears.
5. Specify the parameters for the wind corridor. In addition to the typical parameters, wind corridors have these parameters:
   - Width and height. The dimensions in the X and Z planes.
   - Altitude. This is the altitude of the vertical center of the wind corridor above sea level or terrain. Each vertex in the wind corridor can have a specific altitude. See "Specifying an Object's Altitude" to learn the different ways you can define the altitude.
   - Wind speed. The speed of the wind in the corridor.
   - Priority. This wind corridor's place in the order of precedence determines which corridor prevails when corridors overlap. The highest priority is 1.
6. Click Create.

6.6.2 Creating an Air Turbulence Area

You can create areas of air turbulence. Air Turbulence Areas allow you designate an area on the network where aircraft may experience turbulence. VR-Forces does not simulate the turbulence, but it is part of the comprehensive weather model simulated by VR-Forces. The information is sent over the network for the use of applications and other simulators that need to model air turbulence.

➤ **To create an air turbulence area:**

1. Open the Create Object Palette.
2. Optionally, click the Weather filter ( ).

3. Select Air Turbulence Area.
4. Click to place the vertices of the area. Double-click when you place the final vertex. The Create Air Turbulence Area dialog box displays.
5. Specify the parameters for the air turbulence area. In addition to the typical parameters, air turbulence areas have these parameters:
   - Floor and Ceiling. The altitude of the bottom and the top of the area.
   - Air Turbulence Level. Select the level of turbulence in the area: Light, Moderate, Severe, or Extreme.
6. Click Create.

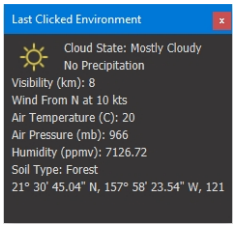
6.7. Displaying Weather Conditions at a Location

VR-Forces can display the weather conditions at a terrain location or the location of an entity in the Last Clicked Environment Panel (Figure 67). If the entity is not on the ground, the panel displays the weather at the altitude of the entity. For subsurface entities, the weather at sea level is reported. The panel shows:

- Cloud cover
- Precipitation
- Visibility
- Wind speed and direction
- Air Temperature
- Air Pressure
- Humidity
- Soil type
- Coordinates and elevation of the location or entity

ⓘ As the entity moves, the environment information updates every two seconds.

Figure 67. Last Clicked Environment Panel



➤ **To view weather conditions at a location:**

1. Choose **View > Last Clicked Environment Panel**. The Last Clicked Environment Panel opens.
2. Click a location on the map or an entity.

6.8. Marine Conditions

VR-Forces supports a variety of dynamic ocean effects (Figure 68), including:

- Douglas Sea State. This includes wind waves (wind sea), swell character, and the directions of each. VR-Forces allows each to be separately configured.
- Wave chop.
- Surface transparency. The ability to see through the water from above sea level.
- Underwater visibility. The ability to see underwater.
- Swell.
- Surge depth. When bathymetry data is present, this is the depth of the water at which full storm effects take place. Where the water is shallower, it is calmer.
- Tidal current speed and direction. If tidal stream wakes are enabled, they are automatically applied to streamed buoys and beacons.

Figure 68. Ocean effects

- Enabling dynamic ocean effects can affect performance.
- To synchronize the rendering of ocean waves between different products or instances, use a command-line option (-O | --hasNTPSync). Without using that option, waves may not match.

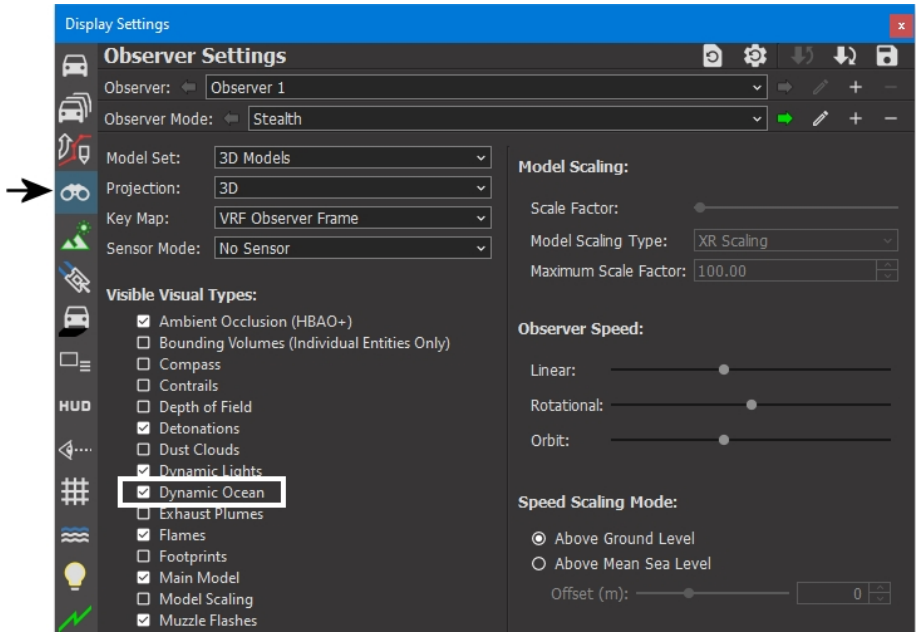
6.8.1 Enabling Marine Effects

Marine effects can reduce performance.

➤ **To enable marine effects (dynamic ocean):**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Observer Settings page (Figure 69).

Figure 69. Observer Settings page



3. Select or clear the Dynamic Ocean check box.

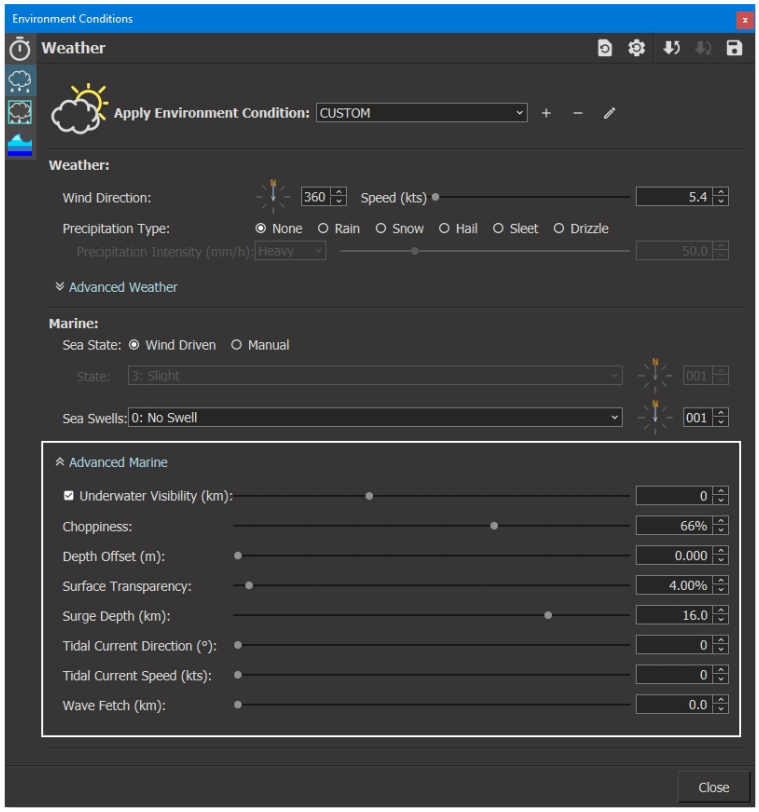
6.8.2 Configuring Advanced Marine Conditions

The preconfigured environment conditions include marine conditions. You can also customize the marine settings at any time.

➤ **To configure marine effects:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Weather page (Figure 70).

Figure 70. Weather page



3. If the Marine section is not expanded, click Advanced Marine. The dialog box expands to show additional configuration options.
4. Change settings as desired. Table 7 describes the marine settings.

Table 7. Advanced marine settings

Parameter	Description
Sea State	Determined by the wind direction and speed or set manually. If manual, select one of the swell states for the Douglas sea state scale. You can also set the compass direction from which sea state originates.
Sea Swells	The amount of rise and fall of waves independent of wind. Choose from 10 fixed swell states. You can also set the compass direction from which swells originate.
Advanced parameters	
Underwater visibility	The distance you can see underwater if the observer is underwater.
Choppiness	The percentage of turbulence on the water.
Depth Offset	The offset of ocean depth.

Table 7. Advanced marine settings (continued)

Parameter	Description
Surface Transparency	The depth at which VR-Forces begins to apply transparency to the water surface if the terrain has bathymetry or underwater polygons. This setting helps reduce Z-fighting along the shoreline. It does not mean that you can see some distance into the water when the observer is over deep water. The maximum value represents transparency to 500 meters.
Surge Depth	When bathymetry data is present, this is the depth of the water at which full storm effects take place. As the water is shallower, it is calmer.
Tidal Current Direction	The direction of the tidal current flow. This affects the display of tidal wakes on buoys and beacons.
Tidal Current Speed	The speed of the tidal current. This affects the display of tidal wakes on buoys and beacons. Default: 0.
Wave Fetch	The wind-fetch distance is the distance that wind can travel over water in a constant direction.

6.8.3 Enabling Tidal Stream Wakes

Tidal stream wakes create wake effects around buoys and beacons based on the movement of the tides. Tidal stream wakes are supported as standalone models. They are automatically attached to streamed buoys and beacons. VR-Forces includes wake models for each of the supported buoy and beacon models. They are named for the buoy or beacon type they support, for example, BuoysPillarWake or BeaconsLatticeWake. You may want to edit the **wakeLength** parameter to provide the best visuals for your terrain.

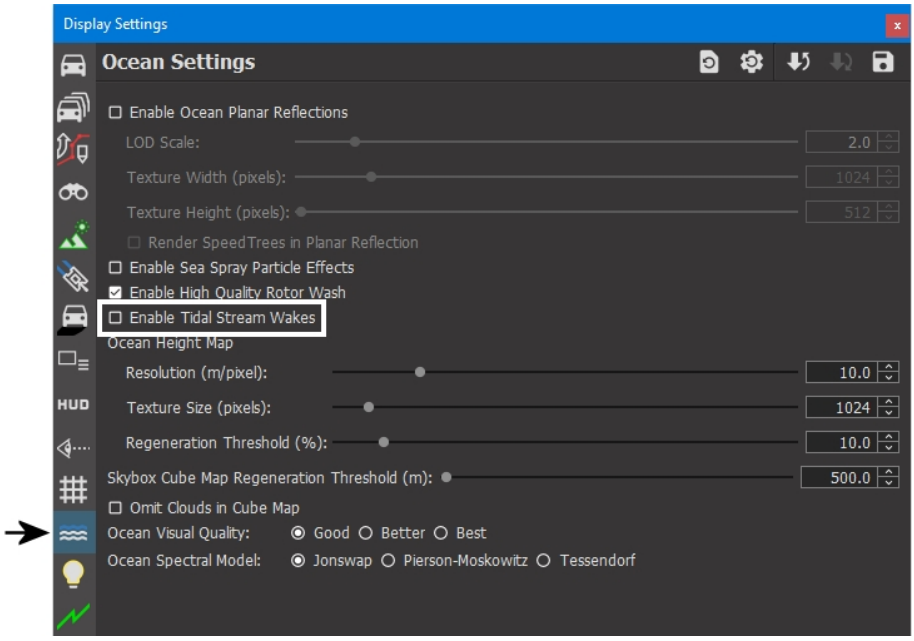
Tidal stream wakes are disabled by default.

ⓘ Even if tidal stream wakes are enabled, you will not see any wakes unless you also specify a tidal current, which is 0 by default. For details about setting tidal current, see "6.8. Marine Conditions" on page 115.

➤ **To enable or disable tidal stream wakes:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Ocean Settings page (Figure 71).

Figure 71. Ocean Settings



3. Select or clear Enable Tidal Stream Wakes.

6.8.4 Setting Ocean Quality

You can adjust the overall quality of visualization of the ocean. Increasing ocean quality decreases performance. Best quality may affect overall performance of your computer. We suggest using this setting primarily for demonstrations rather than large scenarios.

➤ **To set ocean quality:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Ocean Settings page (Figure 71 above).
3. Select one of the Ocean Visual Quality options.

6.8.5 Specifying an Ocean Wave Spectrum

In Ocean Settings, you can specify the wave spectrum to use for the ocean: JONSWAP, Pierson-Moskowitz, or Tessendorf. These are part of the Triton Ocean SDK. JONSWAP is the default spectrum.

The JONSWAP (Joint North Sea Wave Project) wave spectrum model is useful for maritime training applications and buoyancy in simulations where you require realistic ship motion. It is an extension of the Pierson-Moskowitz model, therefore using JONSWAP can have slower performance than Pierson-Moskowitz. Tessendorf is the least physically accurate, but it offers improved performance. For more information on each of these models, consult Sundog Software's documentation on the Triton Ocean SDK.

➤ **To set the ocean spectrum model:**

1. Choose **Settings > Display**. The Display Settings dialog box opens.
2. Select the Ocean Settings page (Figure 71 above).

3. Select one of the Ocean Spectral Model options.

6.8.6 Configuring Breaking Waves

VR-Forces can simulate the effect of waves breaking along the shore line. This feature uses shapefile data that defines the direction of the breaking waves. The *MAK Earth (online).mtf* terrain includes sample data for breaking waves. To see it, view the island of Oahu, Hawaii.

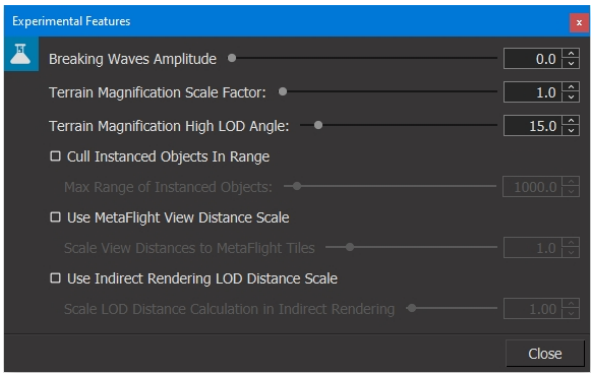
> ⓘ You can set the amplitude of breaking waves. By default, the amplitude is 0.

This is considered an experimental feature. It may be useful, but may affect performance.

➤ **To configure breaking waves:**

1. Choose **Settings > Experimental Features**. The Experimental Features dialog box opens (Figure 72).

Figure 72. Experimental Features dialog box



2. Adjust the Breaking Waves Amplitude slider or type a value in the box. The maximum value is 10.0.

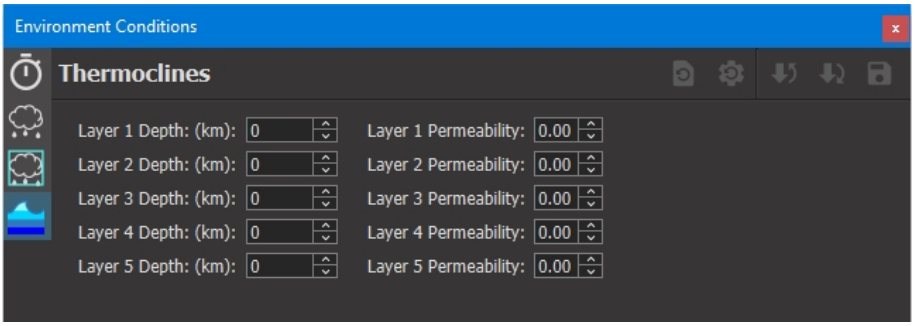
6.9. Setting the Thermocline

Thermoclines are layers of water that have different temperatures and, therefore, different density. Sound transmission varies as it passes through the thermocline. In VR-Forces, sonar effectiveness is reduced as it passes through thermocline boundaries. By default, VR-Forces does not set any thermoclines. You can specify the depth for up to five thermoclines.

➤ **To specify a thermocline:**

1. Choose **Simulation > Environment Conditions**. The Environment Conditions dialog box opens.
2. Select the Thermoclines page (Figure 73).

Figure 73. Thermoclines page



3. For each thermocline that you want to specify, enter or select a value in the depth box and the permeability box.

6.10. METOC Object Classes that Can Be Published

The published METOC object classes are based on both the global weather and any weather zones in the scenario. For details, see the *MAK ONE Products Interoperability Guide*.

7. Scenario Files

This chapter describes the files that are created when you create a scenario.

7.1. Introduction

When you create and save a scenario, VR-Forces stores information about the scenario in these files:

- The scenario file, named *scenario_name.scn*, lists the locations of the other scenario files, the terrain database, and the simulation model set file, and it also sets several parameters for the scenario.
- The scenario objects file, named *scenario_name.oob*, stores identification and state information for simulation objects, control objects, and published tactical graphics.
- The object map file (*scenario_name.omp*) maps objects to the sim. engines on which they were created.
- The plan file (*scenario_name.pln*) stores individual plans and global plans.
- The scenario extras file (*scenario_name.xtr*) lists the forces that each force is hostile to and it stores spawn pattern templates.
- The scripts file (*scenario_name.spt*) stores scenario-specific scripts.
- The overlay file (*scenario_name.ovl*) lists tactical graphics that are not published. Published tactical graphics are in the scenario objects file.
- The selection groups file (*scenario_name.sgr*) stores the selection groups in the scenario and their members.
- The saved views file (*scenario_name.osrx*) stores observer views, if any. It can also specify a startup view.
- The scenario-specific GUI settings file (*scenario_name.gui_settings*) contains GUI settings that only apply to this scenario.

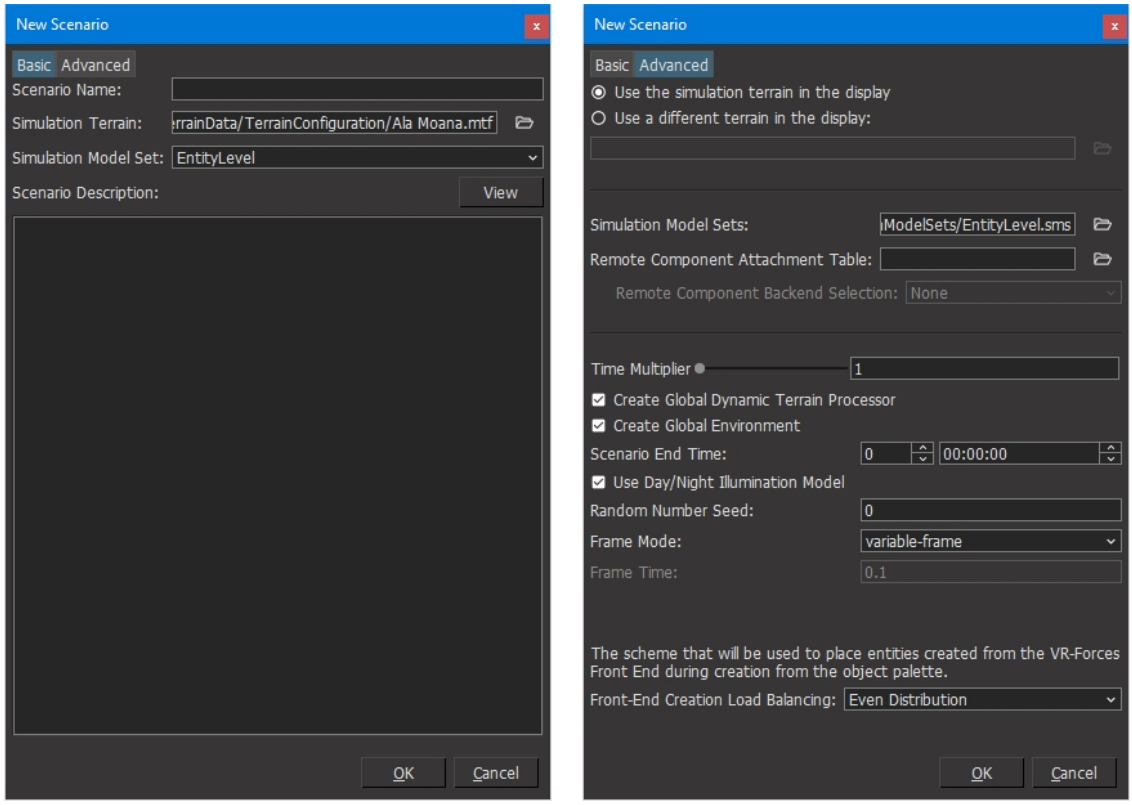
By default, scenarios are saved in a compressed zip archive with the extension *.scnx*. If you examine a compressed scenario file in an unzipping application, you will see that it contains the individual files described in the previous paragraphs.

For the most part, you will not want to edit scenario files by hand. In some cases, we recommend that you not edit them. However, this chapter explains how to edit them if you need to.

7.2. The Scenario File

When you create a scenario, you can set its parameters in the New Scenario dialog box (Figure 74).

Figure 74. New Scenario dialog box



The default values for scenario settings in the New Scenario dialog box are drawn from the file *./appData/settings/vrfGui/default-scenario.tmpl*. If you want to change the default settings, you can edit that file.

7.2.1 Scenario Parameters

A typical scenario file looks like the following (assume a scenario file called *myscenario.scn*):

```
(Scenario
  (Component-Attachment-Table  "")
  (Terrain-Database  "$(SHARED_DATA_DIR)\TerrainData\
      TerrainConfiguration\Ground_DB II.mtf")
  (Gui-Terrain-Database  "$(SHARED_DATA_DIR)\TerrainData\
      TerrainConfiguration\Ground_DB II.mtf")
  (Order-Of-Battle  "myscenario.oob")
  (Scenario-Scripts  "myscenario.spt")
  (Orbat "myscenario.orb"
  (Plan  "myscenario.pln")
  (Overlay  "myscenario.ovl")
  (SelectionGroups  "myscenario.sgr")
  (Object-Map  "myscenario.omp")
  (Scenario-extras  "myscenario.xtr")
  (time-multiplier  1.000000)
  (auto-reorganize False)
  (random-number-seed  0)
```

```
    (frame-mode  "variable-frame")
    (frame-time  0.100000)
    (run-duration-time 0.000000)
    (sim-time  0.000000) //no longer used
    (exercise-start-time  0.000000) // no longer used
    (additional-opds )
    (Simulation-Model-Set-Files "$(DATA_DIR)\simulationModelSets\
        EntityLevel.sms")
    (gui-runtime-scheme "DtRuntimeEvenDistributionLoadBalancer")
    (gui-runtime-scheme-data "")
    (remote-attachment-scheme "DtNoneBalancer")
    (remote-attachment-scheme-data "")
    (scenario-name  "myscenario"
    (scenario-data
      (Create_Global_Dynamic_Terrain  "1")
      (Create_Global_Environment  "1")
      (Set_Scenario_Start_Time_At_Local_On_First_Object_Creation
          "1692180000")
      (UseDayNightModel  "1")
      (DefaultObserverView  "1 - Absolute")
      (GuiObserverViews  "myscenario.osrx")
      (GuiScenarioSettings  "myscenario.gui_settings")
      (Language  "en")
      (ScenarioDescription  "Some text that describes the scenario.")
      (ScenarioExtentInformation  "-2.81258e+006,-4.33259e+006,
          3.72903e+006,938.957")
    )
    (version  3.200000)
  )
```

Paths in the scenario files are relative to the application directory, for example, *./bin64*.

The Starting Time of Day is stored in the scenario objects file, not the scenario file.

Table 8 describes the parameters in a scenario file.

Table 8. Scenario file parameters

Parameter	Description
Component-Attachment	Specifies whether attachment of components to remote simulation objects is allowed in the scenario. The options are: • 0: no component attachment. • 1: the first sim. engine manages all remote components. • 2: remote component attachment is load-balanced on all sim. engines. Although you can set this value in the New Scenario and Load Scenario dialog boxes, it is not saved in the scenario file. To force a specific value, you must edit the scenario file.
Component-Attachment-Table	Specifies the component attachment table to use if attachment of components to remote simulation objects is allowed.
Terrain-Database	Specifies the terrain database to be loaded by the sim. engine.
Gui-Terrain-Database	Specifies the terrain database loaded in the GUI.
Order-Of-Battle	Specifies the scenario objects file.
Scenario-Scripts	Specifies the file that has scenario-specific Lua scripts.
Orbat	Specifies the order of battle file.
Plan	Specifies the plan file.
Overlay	Specifies the overlay file.
SelectionGroups	Specifies the selection groups file.
Object-Map	Specifies the object mapping file.
Scenario-Extras	Specifies the extras file, which records force hostility relationships, and stores spawn pattern templates. It may be used for other purposes in the future.
time-multiplier	Specifies whether a scenario runs in real time or faster than real time. A value greater than 1 runs the simulation faster than real time. You can set this value dynamically using the Time Multiplier toolbar. If you are running a scenario using HLA Time Management, it is strongly recommended that you set **time-multiplier** to 1.
auto-reorganize	Alloows you to specify the default behavior for unit reorganization. Set it to True or False. For information about reorganization, see "How Units Are Organized".
random-number-seed	Sets the default random number seed. It provides a starting point for the generation of random numbers needed by VR-Forces. Enter an integer between 0 and the largest integer permitted on your system.

Table 8. Scenario file parameters (continued)

Parameter	Description
frame-mode	Sets the frame rate mode to one of: • Variable-Frame Run-To-Complete (variable-frame) • Fixed-Frame Best-Effort (fixed-frame) • Fixed-Frame Run-To-Complete (fixed-frame-run-to-complete). These frame rate modes are described in "Exercise Clock Modes".
frame-time	Specifies the length of a frame, in seconds. If **frame-mode** is set to **fixed-frame** or **fixed-frame-run-to-complete**, you must set the frame time to a non-zero value. A value of zero for frame time prevents simulation time from advancing in either of these modes.
run-duration-time	Specifies how long the scenario should run before automatically stopping.
sim-time	This parameter is not supported in scenarios created after VR-Forces 4.0.3 and is only for backwards compatibility
exercise-start-time	This parameter is not supported in scenarios created after VR-Forces 4.0.3 and is only for backwards compatibility
additional-opds	Specifies custom OPD files to load (for backwards compatibility only).
Simulation-Model-Set-Files	Specifies a list of SMS files to load. The path is relative to the default data directory (*./data*) or a user-specified data directory.
gui-runtime-scheme	Internal name used to look up a code-based assignment scheme to determine on which sim. engine to create new objects when they are created from the Create Object Palette.
gui-runtime-scheme-data	Data specific to the **gui-runtime-scheme** parameter.
remote-attachment-scheme	Internal name used to look up a code-based assignment scheme to determine which sim. engine will host remotely attached entities.
remote-attachment-scheme-data	Data specific to the **remote-attachment-scheme** parameter.
scenario-name	An optional parameter that assigns a name to the scenario. It is independent of the file name and is only used in some message dialog boxes.

Table 8. Scenario file parameters (continued)

Parameter	Description
scenario-data	A block of data with these additional parameters: • **Create_Global_Dynamic_Terrain** specifies whether the global dynamic terrain processor should be enabled. If this is disabled, dynamic terrain does not work unless you create a dynamic terrain area. • **Create_Global_Environment** specifies whether the scenario supports environmental conditions. • **DefaultObserverView** specifies the observer view to use at startup. • **GuiObserverViews** specifies the saved views file for the scenario. • **GuiScenarioSettings** specifies the *.gui_settings* file. It records scenario-specific GUI settings. • **HiddenTerrainViews** lists views associated with the scenario terrain that are not displayed in the Observer Views Panel because they were cleared from the Observer Views Panel. • **ScenarioDescription** is the text of the scenario description. • **ScenarioExtentInformation** lists the extents of the scenario for the zoom to extents option. • **Set_Scenario_Start_Time_At_Local_On_First_Object_Creation** specifies the start time for the scenario, in seconds. • **UseDayNightModel** specifies whether to use the day/night illumination model.
version	Specifies versioning information for the scenario file. Do not edit this parameter.

7.2.2 Specifying Pathnames for Scenario Files

Pathnames in a scenario file can be absolute or relative. When VR-Forces saves a new scenario, it creates pathnames that are relative to the application directory. If you think you might need to move a scenario to another computer or a new directory structure, change the paths to be relative to the *./bin64* directory. To change pathnames, edit the scenario file in a text editor. VR-Forces preserves the new paths when it saves the scenario.

If you plan to run multiple GUIs and sim. engines on multiple machines, particularly if they use different operating systems, you must change the pathnames for the terrain database and object parameter database to relative paths or simple pathnames.

Configuring How Paths are Interpreted

These parameters in *vrfSim.mtl* determine how pathnames for the object parameter database and terrain database are resolved:

- **defaultParameterDatabasePath** specifies the path to search for OPD files when an explicit path is not specified in a scenario file.
- **defaultTerrainDatabasePath** specifies the path to search for terrain database files when an explicit path is not specified in a scenario file.

When the sim. engine loads a scenario file, it looks for the object parameter database using this search order:

1. If the path is absolute, load the file in the specified path.
2. If the path is relative, try to load the file relative to the application path (*./bin64/vrfSimprotocol*).
3. Look in the directory specified by **defaultParameterDatabasePath**.

When the sim. engine loads a scenario file, it looks for the terrain database using this search order:

1. If the path is absolute, load the file in the specified path.
2. If the path is relative, try to load the file relative to the application path.
3. Look in the directory specified by **defaultTerrainDatabasePath**.
4. Use the path in the user-defined environmental variable VRF_TERRAIN_PATH, if it is specified.

7.2.3 Changing the Terrain Database for a Scenario

When you create a new scenario, you must choose terrain databases for the GUI and sim. engine. By default, the sim. engine uses the same database as the GUI.

You can edit the pathname of a terrain database in the scenario file or in the Scenario Properties dialog box. However, unless the new terrain database covers the same geographic area as the old one and is the same type of terrain (for example, UTM or geocentric), the scenario objects and overlay information associated with the scenario may be confusing or useless. (For details about editing Scenario Properties, see "2.4.3 Editing Scenario Properties" on page 27.)

7.3. The Plan File

VR-Forces saves all individual and global plans to a plan file (*.pln*).

Plan files are created interactively in the VR-Forces Plan window. The Plan window ensures that the plan is written using the correct syntax. We recommend that you do not try to create plan files or edit plan files by hand.

7.4. The Object Map File

When you start VR-Forces with multiple sim. engines, you can specify which sim. engines simulate the various objects you create. The object map file records the sim. engines on which objects are simulated. The next time you open the scenario, VR-Forces looks for the sim. engines specified in the object map file. If any of the specified sim. engines is not available, the GUI asks if you want to remap objects to the sim. engines that are present. You can run the simulation without remapping. However, the behavior of a simulation that is missing simulation objects is unpredictable.

Mappings between objects and sim. engines are saved to the object map file. If you have a scenario that uses multiple sim. engines and you are uncertain about the site:application values you need to use to run the scenario, you can view the object map file to refresh your memory. This example shows two entities and a route mapped to two different sim. engines:

```
(map-entry
  (address  1 3201)
  (name  "M1A2 1")
)
(map-entry
  (address  2 3400)
  (name  "T80 2")
)
(map-entry
  (address  1 3201)
  (name  "route_1")
)
```

The **address** parameter lists the site and application number used for the sim. engine on which the object was simulated.

If you want to run an existing scenario using sim. engines with different site and application values, you could edit the object map file and change the address field to match the new site and application values. However, if you make any mistakes in this process, your scenario might not run correctly. As an alternative to editing the object map file by hand, you could use the load balancing option to remap objects to sim. engines. For details about load balancing, see "2.5. Load Balancing Scenarios" on page 31.

7.5. Editing the Scenario Objects File

When you create a scenario, you populate it with simulation objects and tactical graphics. When you save or checkpoint a scenario, VR-Forces saves a record of each simulation object and tactical graphic, including its state information and geographic location, to the scenario objects file. The next time you load this scenario, VR-Forces creates the simulation objects using the settings in the object parameter database, but then overrides those settings with the information in the scenario objects file. Therefore, changes you make to the object parameter database might not be picked up by existing scenarios. If you want to change a basic value, you might need to edit the scenario objects file.

7.5.1 Saving Default Parameters to the Scenario Objects File

When a simulation object in a scenario is saved to the scenario objects file, parameters that are unchanged from the initial values in the object parameter database entry are written out in the form:

```
(parameter-name USE-DEFAULT)
```

The USE-DEFAULT keyword indicates that the next time this file is loaded (as part of a scenario) it should be initialized from data in the simulation object's OPD entry. The practical effect of the USE-DEFAULT keyword in scenario objects files is that if you modify the object parameter database, the next time you load a scenario, those changes can affect the simulation objects in the scenario.

This allows you to experiment with changes to parameters and view the effects of the changes to existing scenarios, without modifying the scenario objects file.

7.6. The Scenario Extras File

VR-Forces permits you to configure hostility relationships: for any force you can specify which forces it is hostile towards. By default, Friendly and Opposing forces are hostile to each other. If you change the hostility relationships, they are saved in the scenario extras file (*.xtr*).

This file also stores spawn patterns.

7.7. Temporary Scenario Directories

When you load a scenario, VR-Forces creates temporary directories in which it stores files that it requires to maintain and update the scenario. To prevent conflicts in the temporary directories when running multiple copies of VR-Forces from the same directory, VR-Forces creates unique temporary directories. The directories are created in the path specified by the TMP environment variable. If this variable is not specified on Linux, the */tmp* directory is used. The TMP variable must be specified on Windows. If VR-Forces cannot create a temporary directory, an error message is printed and the application exits. In the GUI, the temporary directory is created by *DtVrfRemoteController* and starts with the prefix *vrffe*. In the sim. engine, the temporary directory is created by *DtCgfDispatcher* and has the prefix *vrfbe*. In both cases, the temporary directory is removed when VR-Forces shuts down.