# Contents

## Basic - common

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow Chart</td>
<td>4</td>
</tr>
<tr>
<td>Main Menu</td>
<td>7</td>
</tr>
<tr>
<td>Ribbon Menu</td>
<td>9</td>
</tr>
<tr>
<td>Pop-up Menu</td>
<td>42</td>
</tr>
<tr>
<td>Default Settings</td>
<td>44</td>
</tr>
<tr>
<td>Operation method</td>
<td>57</td>
</tr>
<tr>
<td>Navigation Options</td>
<td>59</td>
</tr>
<tr>
<td>RoadDataViewer Plug-in</td>
<td>62</td>
</tr>
<tr>
<td>Before and After View for Model Visibility and Group Display</td>
<td>64</td>
</tr>
<tr>
<td>Context</td>
<td>69</td>
</tr>
</tbody>
</table>

## Basic - Road and terrain

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. 3D Elevation Data Selection, Reading and Editing</td>
<td>71</td>
</tr>
<tr>
<td>2. 2D Image Data Selection / Loading</td>
<td>88</td>
</tr>
<tr>
<td>3. Horizontal Road Alignment</td>
<td>97</td>
</tr>
<tr>
<td>4. Vertical curve road</td>
<td>104</td>
</tr>
<tr>
<td>5. Setting of road attribute</td>
<td>109</td>
</tr>
<tr>
<td>6. Road bump</td>
<td>113</td>
</tr>
<tr>
<td>7. On/Off ramp</td>
<td>114</td>
</tr>
<tr>
<td>8. Intersection</td>
<td>117</td>
</tr>
<tr>
<td>9. Road Sign, Markings</td>
<td>131</td>
</tr>
</tbody>
</table>

## Basic - Landscape

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>River</td>
<td>135</td>
</tr>
<tr>
<td>Backdrop</td>
<td>136</td>
</tr>
<tr>
<td>Lake</td>
<td>137</td>
</tr>
<tr>
<td>Flight Path</td>
<td>139</td>
</tr>
<tr>
<td>3D Model (Registration, Editing, Arrangement)</td>
<td>141</td>
</tr>
<tr>
<td>Models Panel</td>
<td>170</td>
</tr>
<tr>
<td>Parametric model</td>
<td>172</td>
</tr>
<tr>
<td>FBX Scene</td>
<td>174</td>
</tr>
<tr>
<td>Texture Tree</td>
<td>177</td>
</tr>
<tr>
<td>3D Tree</td>
<td>179</td>
</tr>
<tr>
<td>Flag</td>
<td>182</td>
</tr>
<tr>
<td>Fire and smoke</td>
<td>183</td>
</tr>
<tr>
<td>3D Text</td>
<td>188</td>
</tr>
<tr>
<td>Video Wall</td>
<td>189</td>
</tr>
<tr>
<td>Changing View Display And Viewpoint Position</td>
<td>190</td>
</tr>
<tr>
<td>Lighting function</td>
<td>193</td>
</tr>
</tbody>
</table>

## Basic - Railway

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Definition of horizontal line for Railway</td>
<td>198</td>
</tr>
</tbody>
</table>

## Simulation

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Waypoint (The motion-control point)</td>
<td>208</td>
</tr>
<tr>
<td>Vehicle Settings</td>
<td>211</td>
</tr>
<tr>
<td>Traffic Flow Settings</td>
<td>226</td>
</tr>
<tr>
<td>Traffic Light Control Settings</td>
<td>243</td>
</tr>
<tr>
<td>Traffic Connector</td>
<td>246</td>
</tr>
<tr>
<td>Road Obstructions</td>
<td>248</td>
</tr>
<tr>
<td>Movable Model Settings</td>
<td>249</td>
</tr>
<tr>
<td>Driving Simulation</td>
<td>250</td>
</tr>
<tr>
<td>Pedestrian Simulation</td>
<td>254</td>
</tr>
<tr>
<td>Flight Simulation</td>
<td>256</td>
</tr>
<tr>
<td>Simulation using Visual Options Tool</td>
<td>257</td>
</tr>
<tr>
<td>Scenario settings</td>
<td>262</td>
</tr>
<tr>
<td>Micro Simulation Player</td>
<td>291</td>
</tr>
<tr>
<td>Replay Plug-in</td>
<td>296</td>
</tr>
<tr>
<td>Log Export Plug-in</td>
<td>299</td>
</tr>
<tr>
<td>VISSIM Link Plug-in</td>
<td>306</td>
</tr>
</tbody>
</table>

## Simulator

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECO Drive Plug-in</td>
<td>312</td>
</tr>
<tr>
<td>Cluster Option Plug-in</td>
<td>316</td>
</tr>
<tr>
<td>Virtual Display Plug-in</td>
<td>326</td>
</tr>
<tr>
<td>SimLink Linkage Plugin</td>
<td>333</td>
</tr>
<tr>
<td>Gaze Tracking Plug-in</td>
<td>352</td>
</tr>
<tr>
<td>Object Detection Plug-in</td>
<td>357</td>
</tr>
<tr>
<td>Camera Sensor Base Plug-in</td>
<td>362</td>
</tr>
<tr>
<td>DS Course Converter plug-in</td>
<td>364</td>
</tr>
<tr>
<td>Speed meter display plug-in</td>
<td>367</td>
</tr>
<tr>
<td>Simulink Connection Plug-in</td>
<td>368</td>
</tr>
</tbody>
</table>

## BIM/CIM

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>DWG Tool</td>
<td>371</td>
</tr>
<tr>
<td>Import / Export LandXML</td>
<td>373</td>
</tr>
<tr>
<td>IFC Plug-in</td>
<td>377</td>
</tr>
<tr>
<td>Soil calculation</td>
<td>380</td>
</tr>
<tr>
<td>4D Simulation</td>
<td>382</td>
</tr>
<tr>
<td>Data exchange with AutoCAD Civil 3D</td>
<td>395</td>
</tr>
</tbody>
</table>

## Crowd

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Character settings</td>
<td>402</td>
</tr>
<tr>
<td>Pedestrian settings</td>
<td>406</td>
</tr>
<tr>
<td>EXODUSPlug-in</td>
<td>410</td>
</tr>
</tbody>
</table>

## Presentation

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Script (auto presentation)</td>
<td>415</td>
</tr>
<tr>
<td>Movie Options</td>
<td>417</td>
</tr>
</tbody>
</table>

## Import / Export

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shapefile Plug-in</td>
<td>420</td>
</tr>
<tr>
<td>GIS View</td>
<td>428</td>
</tr>
<tr>
<td>Parking Model Plug-in</td>
<td>441</td>
</tr>
<tr>
<td>OSM (OpenStreetMap) Plug-in</td>
<td>444</td>
</tr>
<tr>
<td>OpenFlight Converter plug-in</td>
<td>445</td>
</tr>
<tr>
<td>Export Scene Plug-in</td>
<td>446</td>
</tr>
<tr>
<td>FORUM8 Products (3D Model Creation)</td>
<td>448</td>
</tr>
<tr>
<td>3D model creating software</td>
<td>453</td>
</tr>
<tr>
<td>Create MD3 model</td>
<td>460</td>
</tr>
<tr>
<td>Texture Creating And Editing</td>
<td>464</td>
</tr>
</tbody>
</table>

## Practical

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point cloud modeling plugin</td>
<td>470</td>
</tr>
<tr>
<td>VR-Cloud® Plug-in</td>
<td>480</td>
</tr>
<tr>
<td>Xpswmm Plug-in</td>
<td>483</td>
</tr>
<tr>
<td>Tsunami Plug-in</td>
<td>490</td>
</tr>
<tr>
<td>Noise Simulation</td>
<td>495</td>
</tr>
<tr>
<td>Fluid Analysis Plug-in</td>
<td>501</td>
</tr>
<tr>
<td>Munsell Color Space Export Plug-in</td>
<td>505</td>
</tr>
<tr>
<td>SfM (Structure from Motion) Plug-in</td>
<td>508</td>
</tr>
<tr>
<td>Assessment Plug-in</td>
<td>511</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>UAV Plug-in</td>
<td>518</td>
</tr>
<tr>
<td>Oculus Rift Plugin</td>
<td>534</td>
</tr>
<tr>
<td>HTC Vive Plug-in</td>
<td>537</td>
</tr>
<tr>
<td>Custom Shader Sample Plug-in</td>
<td>543</td>
</tr>
<tr>
<td>Rendering of 360-degree image</td>
<td>547</td>
</tr>
</tbody>
</table>
Basic- common
Flow Chart

Terrain Information
- Load Terrain Information
- Input Terrain Information
- Edit Terrain Information

Road Definition
- Horizontal line, Vertical Line, Section Definition
- Spatial Intersection, Plane Intersection
- Banking Interval, Cutting Interval
- Bridge Interval, Tunnel Interval
- On-Off Ramp Settings

Editing
- Set Models, Markings and Tree
- Set Texture
- Generation of traffic flow
- Set Environment, Generation of landscape

Simulation
- Visual Options, Switching landscape
- Walking, Driving and Flight Simulation
- Lightning, Whether, Effect
- Auto-Demonstration by Scripting
- Simulation by Scenario
Main Menu

■ Default Screen

The Main screen is opened after loading the data.

![Main Screen Image]

Load default geographical features.
Load road data.
Make new project.
Download the sample data from RoadDB.
Open the last file (Up to the latest four files). The thumbnail changes depending on the data.

■ Main Screen

The Main screen is opened after loading the data.

Ribbon Menu

On the top of the screen, there are ribbon menus, which help to find commands instantaneously. Refer to “Ribbon Menus” for details.
● Terrain and Coordinate System

Japan JGD2000 / Japan Plane Rectangular CS 3

This shows the terrain and coordinate system in use.

● Save

Save the current data. This is the same process as File ribbon – [Save as…].

● Help

Open the help.

● Event Console

This shows error messages or information.

● 3D Drawing Area

This shows the landscape currently being edited. Changes of the "Visual Options" settings will be reflected immediately.

● Status Bar

Status bar shows the conditions during editing or simulation.

1) Normal

| Frame Rate (fps) | 60.9 | (12583.8, 7594.2) | 34° 59' 15.3" N, 135° 46' 53.5" E | (-112165.0, -21539.4) |

2) During driving simulation

| Travelled 0.19 of 0.40 km | Road "Road 1"+ Lane 1 Speed 50 km/h | Height 1.00 m |

Position on the road, "road being driven"+lane-speed-height of point of view

3) During flight simulation

| Travelled 0.49 of 1.39 km | 3D Flightpath 1 Speed 250 km/h |

Position on the flight path, the flight path, speed
Ribbon Menu
Ribbon menu will be shown on the top of the screen.

UC-win/Road Ribbon
UC-win//Road has the ribbons below on each feature. Click it to display the feature icons on the ribbon.

- File
- Home
- Edit
- Views
- Driving Sim
- Record / Play
- Simulation Link
- Analysis
- Point Cloud
- Device
- Server

*Ribbons depend on installed Plug-in.

Ribbon Group
Items are grouped by the features in each ribbon, which is called a ribbon group. Ex. In the "Home" ribbon, a group surrounded by a red frame is called a "Simulation" group.

Common Operation
Use each function
Click an icon to run the operation.
Ex. Click the walk icon to change the mode to the walk mode.

Display pull down menus
Click ▼ mark to show pull down menus.

Click ▼ mark under the walk icon.

Click "Drive" from the pull down menu.
Traveling starts and the icon will change. Click it again to change to the travel mode.

Common icons

- Open the option form.
- Execute.
- Stop a running function.
- Edit parameters.
- Add information.
- Open an overall function edit from.

Reduced Display

The size of ribbon group can be adjusted.

▼ Normal display

▼ Reduced display

Click the ▼ mark to show the menus in pull down.
### File Ribbon

Click a file ribbon to open the following pull down menus.

#### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>
| **New Project** | **[Default Terrain]** This option loads a new project using pre-defined terrain. Application default settings will be applied automatically. It is possible to change location settings via an option window.  
|               | **[Japan]** Create a new project using the actual Japanese terrain data.                                                                                           |
|               | **[New Zealand]** Create a new project using the actual New Zealand terrain data.                                                                                  |
|               | **[Other country]** Create a new project using an actual terrain relevant to a specific location in the supported countries.                                         |
|               | **[User Defined]** Create a new project for locations other than Japan and New Zealand.                                                                           |
|               | **[Load from file]** Create terrain data supported by GDAL RASTER FORMAT.                                                                                         |
|               | **[Download]** Download the UC-win/Road data from RoadDB. It is downloaded and unzipped in the specified folder <user data path>¥Save or any folder.                      |
|               | **[Import GSI tiles]** Go to [File]-[New Project]-[Importing GSI] to open the [GSI tile setting] screen.                                                         |
| **Open**      | Open an existing file. It is possible to open a file of the following types.  
|               | * .rd : Non-compressed UC-win/Road data file  
|               | * .rdc : Compressed UC-win/Road data file  
|               | * .rdf : UC-win/Road data file output for Free Viewer version                                                                                                    |
| **Reopen**    | Open an existing UC-win/Road project from a list of the four most recently used.                                                                               |
| **Merge**     | Combine two or more existing UC-win/Road projects into one project.                                                                                               |
|               | * It is possible to use "Merge" only when the data being added has the same area and size as the data currently being edited.                                    |
| **Import**    | Import external files from plug-in.                                                                                                                            |
|               | **[Civil 3D data exchange]** Exchange data during using Civil3D by AutoDesk at the same time.  
|               | * Displayed if "Civil3D plug-in" is installed.                                                                                                                  |
|               | **[Import DWG file]** Import DWG file data. DWG files of 2D/3D can be imported.  
|               | * This menu is displayed if "UC-win/Road DWG tool option" (non-free plug-in) is available.  
|               | * In case of 3D, DWG file, which became blocked by 3DFACE and exported by wblock, can be imported. 3D files are imported as standard models.  
|               | * In case of 2D, data must be written by a single stroke line because data is imported as cross section. Polyline is not supported. |
[Import EXODUS Data]
Import simulation results of building EXODUS by EXODUS plug-in. Refer to "EXODUS plug-in".
*This menu is shown if "EXODUS plug-in" is available.

[Import GIS VIEW Data]
Open GIS Viewer.
*This menu is shown if "GIS plug-in" is available.

[Import IFC file]
Import IFC data. UC-win/Road imports the terrain defined by IfcSite within IFC.
*This menu is shown if "IFC plug-in" is available.

[Import InRoads]
Import InRoads data.
*This menu is shown if "InRoads plug-in" is available.

[Import LandXML]
Import LandXML data.
*This menu is shown if "LandXML plug-in" is available.

[OHPASS Convert]
Convert OHPASS data into LandXML data, and then load it in UC-win/Road.
*This menu is shown if "OHPASS plug-in" is loaded.

[Load OpenStreetMap]
Load OpenStreetMap data.
*This menu is shown if "OSM plug-in" (Option in additional cost) is loaded.

[Load Parking Lot]
Load data output by the parking drawing system of UC-1 product into UC-win/Road.

[Import Shapefile]
Import Shapefile data.
*This menu is shown if "Shapefile plug-in" is in usable state.

[Load Tsunami]
Load tsunami data.
*This menu is shown if "Tsunami plug-in" is in usable state.

[Import 12d model data]
Import 12d Model data.
*This menu is shown if "12d Model plug-in" is installed and in unable state.

[Import Universal UI Data]
Load data of Universal UI plug-in.
*This menu is shown if "Universal UI plug-in" is in usable state.

[Import xpswmm data]
Import xpswmm data.
*This menu is shown if "xpswmm plug-in" is in usable state.

[Load RailRoad CSV Curve]
Load Railroad CSV curves.
<table>
<thead>
<tr>
<th>Import FLT Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import FLT format data.</td>
</tr>
<tr>
<td>*This menu is shown if &quot;Export Scene Plug-in&quot; and &quot;OpenFlight Plug-in&quot; are installed and in usable state.</td>
</tr>
</tbody>
</table>

**Export**

Use Export menu to export data.

**Export for Free viewer version**

Save a copy of the UC-win/Road project for Free Viewer Version. The file is saved as *.rdf file. Free Viewer can open only the data file output by this format. *This menu is shown if "Export For Free Viewer Plug-in" (not-free but attached to Ultimate version by default) is installed and activated.

**Civil 3D data exchange**

Exchange the data with Civil3D. *This menu is shown if "Civil3D plug-in" is installed and activated.

**Export 3DS**

Export objects as 3D models. *This menu is shown if "3D Model Output Plug-in" is installed and activated.

**Export DWG**

Export the current project in DWG file. *This menu is shown if "DWG tool plug-in" is installed and activated.

**Export IFC file**

Export IFC file. *This menu is shown if "IFC plug-in" is installed and activated.

**Export InRoads**

Export current data as a LandXML file of InRoads (extension: xml). *This menu is shown if "InRoads plug-in" is installed and activated.

**Export LandXML**

Export current data as LandXML (extension: xml).

**Save to Munsell Color File**

Save a landscape as Munsell Color file (extension: *.mcs). *This menu is shown if "Munsell Color Space output plug-in" is in installed and activated.

**Report**

Output information of the current project in HTML format. *This menu is shown if "Road data viewer plug-in" is installed and activated.

**Export 12d model data**

Export in 12d Ascii file format. *This menu is shown if "12d Model plug-in" is installed and activated.

**Export Universal UI Data**

Output contents data used in Universal UI plug-in. XML file is created where to be output.
<table>
<thead>
<tr>
<th>Menu Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Export FLT</td>
<td>Export in the OpenFlight format.</td>
</tr>
<tr>
<td>Export DS Courses</td>
<td>Export data of roads.</td>
</tr>
<tr>
<td>Save</td>
<td>Save a current UC-win/Road project under the same name.</td>
</tr>
<tr>
<td>Save As…</td>
<td>Save a copy of the UC-win/Road project under a new name.</td>
</tr>
<tr>
<td>Memory Watch</td>
<td>Watch the memory usage for UC-win/Road.</td>
</tr>
<tr>
<td>Project Options</td>
<td>Change the settings such as a logo addition, default basic traffic flow based on the local setting, and the size of intersections.</td>
</tr>
<tr>
<td>License Manager</td>
<td>Add or delete plug-ins and set up types of protect keys.</td>
</tr>
<tr>
<td>Save Current Image</td>
<td>![Save To File] Output an image of the current scene.</td>
</tr>
<tr>
<td></td>
<td>![Copy To Clipboard] Copy a current scene to clipboard.</td>
</tr>
<tr>
<td>Print...</td>
<td>Print a scene displayed in main window.</td>
</tr>
<tr>
<td>Application Options</td>
<td>Set up UC-win/Road system.</td>
</tr>
<tr>
<td></td>
<td>![Default Settings] Set up basic UC-win/Road system</td>
</tr>
<tr>
<td></td>
<td>![Audio Settings] Set up audio settings.</td>
</tr>
<tr>
<td></td>
<td>![Colour Options] Set up display colors for parts without textures...</td>
</tr>
<tr>
<td></td>
<td>![Game Controller Options] Define a game controller for UC-win/Road.</td>
</tr>
<tr>
<td></td>
<td>*This menu is available only if the game controller is connected before activating UC-win/Road.</td>
</tr>
<tr>
<td></td>
<td>![Location Settings] Manage location setting files.</td>
</tr>
<tr>
<td></td>
<td>![RoadDB Proxy Settings] Download models and textures from UC-win/Road browser using HTTP.</td>
</tr>
<tr>
<td></td>
<td>![Civil 3D options] Choose an installed &quot;Autodesk Civil 3D&quot; product to link it with UC-win/Road.</td>
</tr>
<tr>
<td></td>
<td>*This window can be operated if Civil3D plug-in is installed and activated. The installation of &quot;Autodesk Civil 3D 2006&quot; or newer version of Autodesk Civil 3D is required.</td>
</tr>
<tr>
<td>Information</td>
<td>Detailed information.</td>
</tr>
<tr>
<td></td>
<td>![Inquiry] Sends an e-mail to FORUM8's product development department.</td>
</tr>
<tr>
<td></td>
<td>*To use this feature, the &quot;Inquiry Support Tool&quot; must be installed separately.</td>
</tr>
</tbody>
</table>
<Reference>

● Loading data files
If the total size of textures used in data is over 1GB, they will automatically resize to reduce memory consumption in the following order. The resizing state will be shown as “level #”.

● Importing terrain patch data
Use terrain patch data to apply terrain data more detailed than the terrain mesh (50 square meters).

● License Manager
- Click Road version.
- Use “Batch setting” button to set the usage status and authentication mode of the plug-ins in a lump.
Home Ribbon

Home Ribbon has commands to change camera mode and environment shortly after opening or editing the data and to execute simulations like scripts and scenarios.

Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Navigation</td>
<td></td>
<td>Change camera mode and navigation mode. It is possible to set up camera mode such as driving and flying.</td>
</tr>
<tr>
<td>Camera mode</td>
<td></td>
<td>Look: Pivot around current camera position</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Zoom: Move back or forward.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Move: Move up, down, left or right</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fly: Change to “Fly”.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Spin (model): Pivot around a model.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Satellite: Change to “Satellite”.</td>
</tr>
<tr>
<td>Free</td>
<td></td>
<td>Change a navigation mode to “Free”.</td>
</tr>
<tr>
<td>Navigation mode</td>
<td></td>
<td>Change a navigation mode. Click ▼ mark and select a navigation mode.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Drive: Start driving on road.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Travel: Start simulation driving on road.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rail: Start running on rail.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fly: Change to “Fly” mode.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Walk: Change to “pedestrian” mode.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Follow: Change to “Follow” mode. Select a vehicle to ride in the passenger’s seat. It is possible to follow a car from the view of outside the vehicle by using a mouse wheel or from any angles by drag.</td>
</tr>
<tr>
<td>Pause/Continue Driving</td>
<td></td>
<td>Pause or restart walking/driving/flying simulation.</td>
</tr>
<tr>
<td>Change the vehicle</td>
<td>Change the vehicle during running.</td>
<td></td>
</tr>
<tr>
<td>--------------------</td>
<td>-----------------------------------</td>
<td></td>
</tr>
<tr>
<td>Navigation Options</td>
<td>Open navigation option form to set up moving speed or various effects.</td>
<td></td>
</tr>
</tbody>
</table>

### Simulation

Use this group to display the traffic flow and environments.

<table>
<thead>
<tr>
<th>Start Traffic</th>
<th>Start/Finish displaying traffic movement on 3D screen.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pause Traffic</td>
<td>Pause traffic movement in the window. Click this again to resume.</td>
</tr>
<tr>
<td>Stop Traffics</td>
<td>Reset traffic movement.</td>
</tr>
<tr>
<td>Run Traffic</td>
<td>Create traffic movement at high speed.</td>
</tr>
<tr>
<td>Traffic Step</td>
<td>Display traffic step.</td>
</tr>
</tbody>
</table>

**Traffic Status**

Save and load traffic movement.

- **Save traffic status**
  A traffic simulation on the whole project at the time is exported to a file.

- **Restore traffic status**
  The traffic simulation preserved beforehand is loaded.

- **Register traffic status**
  Register traffic status file saved by [Save Traffic Status].

**Environment**

Display movements of environment, character on movable models, and weather, and water huddle, movement of background and flight path.

### Environment

Set up environment such as time and weather on this group.

<table>
<thead>
<tr>
<th>Visual Options</th>
<th>Set up settings related to main window display.</th>
</tr>
</thead>
</table>

**Weather setting**

Set up weather in the scene related to environment.

- Fine
- Rain
- Snow

**Time setting**

Set up display time. The sun position changes depending on it.

**Visibility**

Select a model group to display on current environment. Click on a pencil mark and open Visible Models display.
**Context**

Select a context to apply to the current scene. Click on a pencil mark to create or edit contents on Context editor.

**Sign settings**

Switch signs which are created and arranged by parametric model.

**Script / Animation**

Use this function to edit and execute scripts.

Start a selected script animation. Select an animation and click Play to start it automatically.

Click on a pencil button to open Script editor to create and edit scripts.

**Scenario**

Use this function to edit and start scenario.

Start a selected scenario.

*Scenario Plugin is needed to edit scenario.

**Camera Position**

Use this function to save and change camera position.

Save a scene on current main display as camera position.

Create a new camera position on displayed editor.

Edit a current camera position.

Open a list window of saved scenes. Set up the keyboard shortcut keys or each view setting.

**Edit**

Frequently used editing functions are put together.

Create and edit roads and rails.

Open a registration window to create and edit road sections.

Models can be edited and located.

**Cluster**

Use this menu to switch to standard mode, cluster master and cluster client.
**Stand-alone**
- Stop the cluster mode and show as standard single display.

**Master**
- Switch to cluster master.

**Client display**
- Switch to cluster client (display mode).

**Client free**
- Switch to cluster client (free operation mode).

**<Reference>**

Mouse operation 3 (scroll)
Scroll a mouse wheel to go forward or backward.

Mouse operation 2 (holding down right click)
Hold down right click and drag on the main screen to revolve a camera around a white balloon the 3D space.

Jump (J key)
Press [J] key one time or right click on the main screen and select [Jump To] - [Click]. Then Click any place to jump to.

● **[Save Camera Position]**
When the data is loaded, it automatically displays a first saved camera position.
It is possible to freely rearrange the saved camera positions.
The number keys from 1 to 0 on the keyboard can be allotted to 10 saved camera positions. By pressing one of the keys, it is possible to instantly move to the assigned position. Allocation of the numbered keys can be set up in "Camera Positions" window.
*Numeric keys on the right side of the keyboard are for switching viewing locations. So they cannot have camera positions.
*It is possible to display plural camera views on a sub screen.

● **[Camera initial position]**
A landscape position selected by the [Default] button on the [Camera Positions] screen is a position to be displayed first when the project is loaded. The [Load position] letters will be displayed on the right side of the camera position name specified by [Default] button.
## Edit Ribbon

Edit Ribbon has commands required for the data editing.

### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Scene</strong></td>
<td>![Scene icon]</td>
<td>Use Scene group to create road, register and arrange models and setup draft mode.</td>
</tr>
<tr>
<td><strong>Plan View</strong></td>
<td>![Plan View icon]</td>
<td>Create roads / flight paths / rivers / backgrounds / lakes / cross sections on the current terrain.</td>
</tr>
<tr>
<td><strong>Library</strong></td>
<td>![Library icon]</td>
<td>Manage all models in the project by using the model panel. Register, edit, and arrange models.</td>
</tr>
<tr>
<td><strong>Draft Mode</strong></td>
<td>![Draft Mode icon]</td>
<td>Switch the normal mode and the draft mode. Click to open the draft mode options window. Use the draft mode to reduce the time of road creation and check the road shape easily.</td>
</tr>
<tr>
<td><strong>Terrain</strong></td>
<td>![Terrain icon]</td>
<td>Load terrain patch data for detail settings of the terrain.</td>
</tr>
<tr>
<td><strong>Patch</strong></td>
<td>![Patch icon]</td>
<td>Load terrain patch data (extension: .lem) to the terrain being edited.</td>
</tr>
<tr>
<td>[LEM file]</td>
<td></td>
<td>Apply [Numerical map 5m mesh (height)] data (extension: .lem) to the terrain. It is possible to load directly [5m mesh of numerical value map (Elevation)] data provided by Geospatial Information Authority of Japan as 5m mesh data. To load [Base map information] from website of Geospatial Information Authority of Japan, convert it to Shape file.</td>
</tr>
<tr>
<td><strong>XML file</strong></td>
<td></td>
<td>Apply terrain patch data (extension: .xml) to the terrain.</td>
</tr>
<tr>
<td><strong>Street Map</strong></td>
<td>![Street Map icon]</td>
<td>Paste a street map on the terrain.</td>
</tr>
<tr>
<td><strong>Dice streetmap</strong></td>
<td>![Dice streetmap icon]</td>
<td>Split large sized picture like satellite image.</td>
</tr>
<tr>
<td><strong>Road</strong></td>
<td>![Road icon]</td>
<td>Settings for roads. It is possible to add obstructions, edit road surfaces, add side objects or edit sections.</td>
</tr>
<tr>
<td><strong>Section</strong></td>
<td>![Section icon]</td>
<td>Edit road sections used for roads and railways.</td>
</tr>
<tr>
<td><strong>Side Objects</strong></td>
<td>![Side Objects icon]</td>
<td>Arrange signs, 3D models, and trees on the side of roads or markings on the lane.</td>
</tr>
<tr>
<td><strong>Obstructions</strong></td>
<td>![Obstructions icon]</td>
<td>Edit existing road obstructions. Modify the obstructing without allocation due to an error.</td>
</tr>
<tr>
<td><strong>Bumps</strong></td>
<td>![Bumps icon]</td>
<td>Edit road bumps.</td>
</tr>
<tr>
<td><strong>Surfaces</strong></td>
<td>![Surfaces icon]</td>
<td>Edit road surface.</td>
</tr>
</tbody>
</table>
### OffRoad
- Create and edit starting point of off road.

### Pedestrian
#### Network
- Make settings for the network to perform a crowd movement.

### Flightpath
#### Record Flightpath
- Set a flight.

#### Edit Flightpath
- Open the Flightpath List Editor to edit, delete, display and hide what is selected.

#### Export Flightpath
- Export the selected flight path on the main to the formatted text file.

#### Import Flightpath
- Import the formatted flight path text file.

#### Delete Flightpath
- Delete the selected flight path.

### Headlights
#### Headlight Options
- Edit headlights of vehicle in UC-win/Road.

#### Street lights
- Add streetlights on the project and edit added streetlights.

### Traffic
- Functions to generate and connect traffic flows and manage vehicle groups in this group.

#### Generator
- Set traffic flows on the road.

#### Connections
- Edit traffic connection of road network.

#### Vehicle Groups
- Manage vehicle groups.

### Smoke Tunnel
#### Edit
- Edit smoke tunnels for smoke simulation.

### Road data
- Functions to obtain the object information of current project.
- This menu is shown if "Road Data Viewer Plugin" is installed and authenticated.

#### Viewer
- Open main window of Road data viewer. List of objects arranged on current project are displayed by a tree format.
Zone Editing
Use this to create the zone in the main form and add the building models and the forests.

<table>
<thead>
<tr>
<th>Create Zone</th>
<th>Change to zone creating mode. Add an arbitrary shape of zone model.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create Building</td>
<td>Change to shape creating mode. Add an arbitrary shape of building model.</td>
</tr>
<tr>
<td>Create Forest</td>
<td>Change to area creating mode. Add an arbitrary shape of forest model.</td>
</tr>
<tr>
<td>Download Object</td>
<td>Change to object download mode. Download building models and forest models from the specified data source and add them to the zone.</td>
</tr>
</tbody>
</table>

Spread meter display
Use this to control the speed meter of driving vehicle. Spread meter plug-in is required.

<table>
<thead>
<tr>
<th>Open</th>
<th>Open a panel of speed meter.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Settings</td>
<td>Open the setting form.</td>
</tr>
</tbody>
</table>

FLT Road alignment
Use this to generate transparent roads on imported OpenFlight models. Open Flight Plug-in is required.

<table>
<thead>
<tr>
<th>Open</th>
<th>Open the panel of speed meter.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Settings</td>
<td>Open the setting form.</td>
</tr>
</tbody>
</table>

<Reference>

● [Draft mode] Is a design mode for easily checking design conditions.
In draft mode (when the button is pressed), the physical relationship between the road and the terrain is calculated only the first time, and cutting, banking, and intersections will not be regenerated. As a result, the time required to generate the road is decreased.

● [Traffic Movement]
- It's possible to board a moving vehicle or ride in a stream of traffic by clicking on the vehicle or stream of traffic. Switching between passenger seat and driver seat is possible by clicking [Enter] key after getting in a car.
- Vehicles can be deleted individually by clicking on them whilst holding [Ctrl], [Alt] and [D].

● [Display traffic movement]
Registered traffic movement can be edited on the ribbon, Home – Simulation – Start Traffic/ Stop Traffic.
* For roads that have grade intersections, right turn and left turn can be created at the intersection. Set up the drive information on the “Drive Paths” tab in the “Intersection Editor” form.
## Views Ribbon

Use View ribbon to set up screen.

### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows</td>
<td></td>
<td>Display views of each direction with the main screen.</td>
</tr>
<tr>
<td>Left</td>
<td><img src="image" alt="Left" /></td>
<td>Display view to left with the main display.</td>
</tr>
<tr>
<td>Rear</td>
<td><img src="image" alt="Rear" /></td>
<td>Display view to rear with the main display.</td>
</tr>
<tr>
<td>Right</td>
<td><img src="image" alt="Right" /></td>
<td>Display view to right with the main display.</td>
</tr>
<tr>
<td>Options</td>
<td><img src="image" alt="Options" /></td>
<td>Open the edit window of the simulation screen.</td>
</tr>
<tr>
<td>Full Screen</td>
<td><img src="image" alt="Full Screen" /></td>
<td>Hide the title, menu, toolbar and status bar, and maximize the display of the scene's drawing area.</td>
</tr>
<tr>
<td>Hide menu</td>
<td><img src="image" alt="Hide menu" /></td>
<td>Hide the menu.</td>
</tr>
<tr>
<td>Viewing Size</td>
<td><img src="image" alt="Viewing Size" /></td>
<td>Change the size of 3D display area of main screen edited.</td>
</tr>
<tr>
<td>Orthographic View</td>
<td><img src="image" alt="Orthographic View" /></td>
<td>Toggle the Orthographic View on / off.</td>
</tr>
<tr>
<td>Add View</td>
<td><img src="image" alt="Add View" /></td>
<td>Displays the view of saved landscape position. Display multiple screens at a time.</td>
</tr>
<tr>
<td>Hide Views</td>
<td><img src="image" alt="Hide Views" /></td>
<td>Hide the displayed screen while Saved camera position display menu is opened.</td>
</tr>
<tr>
<td>Virtual Displays</td>
<td><img src="image" alt="Virtual Displays" /></td>
<td>Open System-wide Virtual Display editor.</td>
</tr>
<tr>
<td>UniversalUI</td>
<td><img src="image" alt="UniversalUI" /></td>
<td>Start Universal UI. Go to File – Import to load contents.</td>
</tr>
<tr>
<td>Custom Shader Sample</td>
<td><img src="image" alt="Custom Shader Sample" /></td>
<td>Open the Custom Shader Sample property.</td>
</tr>
</tbody>
</table>

### Multiple views

- In order to use the multiple view display with four monitors, it is required to have a PC equipped with a motherboard that supports NVIDIA SLI (Scalable Link Interface) and two NVIDIA video cards that support two types of video output.
- The same size monitor is recommended in case of multi-view display. When the aspect ratio is different, it may be hard to see the screen.
- Each view screen can be open on 1 monitor, but it's not displayed better than in multi monitor environment.
- [Orthographic View]

The "Orthographic View" window displays a 2D view from above according to the orthographic projection.

*This is similar to the display of the "satellite" viewing mode, but strictly speaking they aren't the same

---

### Object display on 2D view

<table>
<thead>
<tr>
<th>Objects</th>
<th>Setting items</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>Traffic flow vehicles</td>
<td>Name, Custom ID, Box (Color), Speed, Brake, Winker</td>
<td><img src="image" alt="Traffic Flow Vehicles" /></td>
</tr>
<tr>
<td>A vehicle traveling in front</td>
<td></td>
<td><img src="image" alt="Example 1" /></td>
</tr>
<tr>
<td>A driving vehicle</td>
<td></td>
<td><img src="image" alt="Example 2" /></td>
</tr>
<tr>
<td></td>
<td>&lt;Only for a driving vehicle&gt;</td>
<td><img src="image" alt="Example 3" /></td>
</tr>
<tr>
<td></td>
<td>Distance from a vehicle traveling in front</td>
<td><img src="image" alt="Example 4" /></td>
</tr>
<tr>
<td></td>
<td>Speed limit</td>
<td><img src="image" alt="Example 5" /></td>
</tr>
<tr>
<td></td>
<td>Name of the road</td>
<td><img src="image" alt="Example 6" /></td>
</tr>
<tr>
<td></td>
<td>Note :</td>
<td><img src="image" alt="Example 7" /></td>
</tr>
<tr>
<td></td>
<td>Set up the name to display it.</td>
<td><img src="image" alt="Example 8" /></td>
</tr>
<tr>
<td>Pedestrian</td>
<td>Name, Custom ID, Box (Color)</td>
<td><img src="image" alt="Example 9" /></td>
</tr>
<tr>
<td>Flying model</td>
<td>Name, Custom ID, Box (Color)</td>
<td><img src="image" alt="Example 10" /></td>
</tr>
<tr>
<td>Traffic light</td>
<td>Color display, Box (Color)</td>
<td><img src="image" alt="Example 11" /></td>
</tr>
<tr>
<td>Road model</td>
<td>Road sign</td>
<td>Road side tree</td>
</tr>
<tr>
<td>------------</td>
<td>-----------</td>
<td>----------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&lt;Road sign&gt;</td>
<td>Size</td>
<td></td>
</tr>
<tr>
<td>&lt;Road model&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>&lt;Terrain&gt;</td>
<td>Penetration efficiency</td>
<td>&lt;Intersection&gt;</td>
</tr>
<tr>
<td>Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Custom ID</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>HUD font</th>
<th>Camera position</th>
<th>Font change</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>&lt;HUD font&gt;</td>
<td>&lt;Camera position&gt;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Font change</td>
<td>&lt;Camera position&gt;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Color</td>
<td>&lt;Camera position&gt;</td>
<td></td>
</tr>
</tbody>
</table>
Detailed settings about scene views can be done in this window.

How to open this window: Right-click the draw field and click "Edit Camera Positions". Select any scene from the list and click "Edit".

**[Name]**
Input the name of view.

**[Camera position and direction]**

**Coordinate system**
Select one of the following coordinate system

- **Position**
  Setting of camera position
  - **Absolute**
    Specifies the camera position according to the coordinates of the internal coordinate system in UC-win/Road. The internal coordinate system is a convention for coordinates wherein the origin is the point at the top left corner of the project terrain, the x-axis increases in the Eastern direction, the z-axis increases in the Southern direction and the y-axis increases in the upwards direction. So especially when inputting the z value, make sure that the value lies between the Northern and Southern boundaries.
  - **Relative**
    Camera's position will be the position where the object specified as "Reference point" is and camera will follow this object.

**Direction**
Specify the camera angle. Choose either "View point" or "Angles" from "Defined by:"

- **View point**: Sets camera angle with the coordinate value in the local coordinate system.
- **Angles**: Sets camera direction with Yaw and Pitch.

**Preset**

By selecting the values of camera position and direction from preset values, save time to enter these values.

**[Frustum shift]**
Camera moves horizontally and vertically while fixing the frustum position that the camera displays.

- **Horizontal**: With the width of client area as 100%, sets how much the camera moves to right or left. Right is positive and left is negative value.
- **Vertical**: With the height of client area as 100%, sets how much the camera moves upwards or downwards. Up is positive, and down is negative value.

**[FOV]**
Set angle of view in the view. Select from the followings, and set parameters if necessary.

- **Use the same FOV as the main window**: Applies the FOV of the main screen defined in the simulation screen editor.
**Define the vertical FOV only and use aspect ratio:** Setting of vertical and horizontal angle of view. Horizontal angle of view is automatically set in the aspect ratio of the current screen.

**Define the FOV of each side separately:** Separately sets angle of view in each direction of right, left, top, and bottom against the center of the view.

**[Camera Renderer]**
Select the screen rendering method. Normally, the default is Default renderer.
- **Default renderer:** This is for the default display.
- **Camera Sensor Renderer:** This is to simulate the Fisheye Lens Display and others, if “Camera Sensor Base Plug-in” is enabled.
- **360 degree image Renderer:** This is to render 360 degree image
- **Custom Shader Sample Renderer:** This is to render based on data of object type, drawing normal, depth, object speed, and acceleration, etc.

**[Setting form for each camera renderer]**
- **Default renderer:** no setting form
- **Camera Sensor Renderer:** Camera Sensor Renderer Setting form is opened.
- **360 degree image Renderer:** 360 degree image Renderer Setting form is opened.
- **Custom Shader Sample Renderer:** Custom Shader Sample Renderer Setting form is opened.

**[Stereo]**
Display view in stereo.
- **FOV adjustment:** the view is displayed in stereo.
- **Stereo focal length:** Sets focal length in stereo view
- **Eye separation:** Sets length between both eyes in millimeters

**[Other options]**
- Flip horizontally (mirror)
- Camera roll
- Allow movements on mouse operation in view windows
## Driving Sim Ribbon

### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Log export</strong></td>
<td></td>
<td>This group has functions related to log outputs. This menu is shown if &quot;Log Export Plugin&quot; is installed and activated.</td>
</tr>
<tr>
<td><strong>Options</strong></td>
<td></td>
<td>Set simulation information to export and destinations</td>
</tr>
</tbody>
</table>
| **Start logs**|        | Start export logs of real time data set on Log file format of Log Export Plug-in.  
|               |        | Hint: it is possible to start logging by starting an event which has a Start Logs command. |
| **Stop logs** |        | Stop log export.                                                             |
|               |        | Hint: it is possible to stop logging by starting an event which has a Stop Logs command. |
| **Edit profile** |      | Edit profile for log export file name. Setting is optional.                |

### Drive Simulator

This group has functions used for the driving simulation. It is shown if "Drive Simulation plugin is installed and activated.

<table>
<thead>
<tr>
<th>Options</th>
<th></th>
<th>Adjust and enable drive simulator and activate a handle. Refer to “Drive Simulator Options” for details.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Drive</strong></td>
<td></td>
<td>Start driving simulator.</td>
</tr>
</tbody>
</table>

### ECO Drive

ECO drive group has functions used for the ECO drive diagnosis. It is shown if "ECO Drive plugin" is installed and activated.

<table>
<thead>
<tr>
<th>Options</th>
<th></th>
<th>Open ECO Drive options. E</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Start</strong></td>
<td></td>
<td>Start ECO drive.</td>
</tr>
<tr>
<td><strong>Stop to Save</strong></td>
<td></td>
<td>Stop ECO drive and save result.</td>
</tr>
<tr>
<td><strong>Analysis Viewer</strong></td>
<td></td>
<td>Display the result of ECO driving.</td>
</tr>
</tbody>
</table>

### Driving Diagnosis

It is shown if Communication Plugin is installed and activated.

<table>
<thead>
<tr>
<th>Options</th>
<th></th>
<th>Open the Obstacle Helper Tool.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Start</strong></td>
<td></td>
<td>A main menu of drive diagnosis is displayed. This is for administrator and for training.</td>
</tr>
<tr>
<td><strong>Obstacles settings</strong></td>
<td></td>
<td>Open the Obstacle Helper Tool.</td>
</tr>
</tbody>
</table>
### Communication

It is shown if Communication Plugin is installed and activated.

| Connect | ![Icon] | Open Client display and connect the other UC-win/Road. |

### Steering Wheel

It is used to control a handle on a front passenger seat. This command is displayed if Demo function is installed.

| Control Demo | ![Icon] | Ride a vehicle during driving. Switch to passenger seat mode by [Enter] key... |

### Simulink

Use Simulink group to operate “Simulink connection plug-in”. This group is displayed when the “Simulink connection plug-in” is installed and is in usable state.

| Options | ![Icon] | Displays Options form for the "Simulink connection plug-in". |
| Help | ![Icon] | Displays the help for "Simulink connection plug-in". |
Record / Play Ribbon

In this Ribbon, operate AVI recording function and POV-Ray output.

Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| Performance Log       |        | Traffic thread, drawing condition, state of data communication by cluster at that time are exported to log file. Click it again to export the log is exported.  
    Hint: Cluster client exports log per machine. |
| Distance              |        | Choose either “Two objects” or “Two points” from “Distance between” in the right click menu on the main screen, and click two objects or two points in 3D space. 3D or 2D distance between two objects will be exported to log file. Click the button again to stop exporting. |
| Movie                 |        | Functions to generate video like AVI output.                               |
| Options               |        | Settings of AVI files and ray tracing files for POV-ray.                   |
| Start                 |        | Start AVI file creation. It is possible to record movements.               
    Hint: At the time of AVI recording, size of main window is adjusted automatically to the set size. |
| Stop                  |        | Stop creating AVI files.                                                   |
| Export to POV-Ray     |        | Create folders like the following and export to ray tracing files for POV-Ray according to the “POV-Ray Options”.  
    *This option is shown if “POV-Ray Plugin” is in usable state. |
| Export Static Models to POV-Ray |    | Export models on current project. This menu is used when creating flames for movie or before start recording. |

Replay

Functions to operate Replay plugin. *This menu is shown if “Replay Option” (Option in additional cost) is installed.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Player</td>
<td></td>
<td>Open the replay option window to save movements of vehicles and characters in UC-win/Road and then replay these movements.</td>
</tr>
<tr>
<td>Toolbar</td>
<td></td>
<td>Replay toolbar is displayed in the bottom of the main window.</td>
</tr>
<tr>
<td>Search</td>
<td></td>
<td>Open the Search form that searches for the recorded file once quickly.</td>
</tr>
<tr>
<td>Options</td>
<td></td>
<td>Open the Options form for the Replay plug-in.</td>
</tr>
</tbody>
</table>

Micro Simulation

In this group, functions for the micro simulation are available.  
*It is shown if “MicroSimulationPlayer Plugin” (Option in additional cost) is installed.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Editor/Player</td>
<td></td>
<td>Open main window of the micro simulation player. It is possible to play not only the data recorded in UC-win/Road, but also results simulated by other applications like VICS.</td>
</tr>
<tr>
<td>Recorder</td>
<td>Record movements of models in Open Micro Simulation format by displaying the traffic flow. The recorded files can be reflected as simulation.</td>
<td></td>
</tr>
<tr>
<td>---------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>Toolbar</td>
<td>A toolbar for the micro simulation player is displayed in the bottom of main screen.</td>
<td></td>
</tr>
</tbody>
</table>

<Reference>

● [Start/Stop AVI Recording]

Data is only recorded whilst in motion when in point of view mode and during walking / drive / flight simulation.

*When recording an AVI, the size of the main window is automatically adjusted so that it can be recorded at the set-up size.
Sim Link Ribbon

In this ribbon, functions of data link with external traffic simulations are available.

### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Intersection</strong></td>
<td></td>
<td>This group has functions to import and visualize the data from intersection simulation OSCADY PRO in UC-win/Road. This menu is shown if &quot;OSCADY PRO Plugin&quot; is installed and authenticated.</td>
</tr>
<tr>
<td>OSCADY PRO data link</td>
<td>[OSCADY PRO data link]</td>
<td>Opens OSCADY PRO data import form window. The data can be imported from OSCADY PRO to UC-win/Road.</td>
</tr>
</tbody>
</table>

| **Road network** |             | Road networks can be exchanged between SIAS PARAMIC project and UC-win/Road. This menu is shown if "PARAMICS Plugin" is installed and authenticated. |
| PARAMICS data link | [PARAMICS data link] | Opens PARAMICS data link window. UC-win/Road can import PARAMICS data for the creation of roads, intersections, and roundabouts. PARAMICS file can also be created from the road network exported from UC-win/Road. |

| **Aimsun** |             | Aimsun is a simulator developed by TSS ([http://www.aimsun.com/](http://www.aimsun.com/)) and it can simulate traffics by setting roads, intersections, signal switching, and traffic volume. In "Information-using interactive simulation system between human – vehicle - traffic", it is possible to link vehicle movements and signal indications between a traffic simulator Aimsun and a driving simulator UC-win/Road. This menu is shown if "Aimsun Link Plugin" (Option in additional cost) is installed and authenticated. |
| Assign roads | [Assign roads] | Open Aimsun Section List Editor window. Settings about the linkage can be done. |
| Link with Aimsun | [Link with Aimsun] | A panel of linkage between UC-win/Road and Aimsun. |

<p>| <strong>Tracks</strong> |             | Use this group to link up with Tracks. This traffic simulation software has been developed by Gabites Porter Consultants and consists of about 60 modules. This menu is shown if &quot;Tracks Plugin&quot; is installed and authenticated. |
| Options | [Options] | Name network for Tracks. |
| Import Results | [Import Results] | Import the calculation result (*.xml) that Tracks output. Select and apply imported Tracks data per time unit. |</p>
<table>
<thead>
<tr>
<th><strong>Trip Classes</strong></th>
<th><strong>Trip Classes</strong></th>
<th>Edit the trip classes of Tracks.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Apply Loading</strong></td>
<td><strong>Apply Loading</strong></td>
<td>Select and apply the imported Tracks data.</td>
</tr>
<tr>
<td><strong>Move To Node</strong></td>
<td><strong>Move To Node</strong></td>
<td>Search the UC-win/Road object associated with the Tracks models from the node number and move it.</td>
</tr>
</tbody>
</table>

**EXODUS Link**
Use this group to visualize the analysis result of evacuation analysis simulation software EXODUS.
This menu is shown if EXODUS Plugin is installed. Load analysis result from File – Import and control with the following buttons.

<table>
<thead>
<tr>
<th><strong>Exodus Link</strong></th>
<th><strong>Exodus Link</strong></th>
<th>Opens the main window of Exodus Link.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Pause</strong></td>
<td><strong>Pause</strong></td>
<td>Pause the animation.</td>
</tr>
<tr>
<td><strong>Play</strong></td>
<td><strong>Play</strong></td>
<td>Play the animation.</td>
</tr>
<tr>
<td><strong>Previous</strong></td>
<td><strong>Previous</strong></td>
<td>If pausing, displays the previous step.</td>
</tr>
<tr>
<td><strong>Next</strong></td>
<td><strong>Next</strong></td>
<td>If pausing, displays the next step.</td>
</tr>
</tbody>
</table>

**VISSIM**
Use this to connect with PTV VISSIM. VISSIM plug-in is required.

<table>
<thead>
<tr>
<th><strong>Settings</strong></th>
<th><strong>Open the setting form.</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Connect</strong></td>
<td>Start connecting with VISSIM. After connecting, the button changes to Disconnect.</td>
</tr>
</tbody>
</table>
## Analysis Ribbon

Use this ribbon to perform engineering analyses and display analysis results on UC-win/Road.

![Image of ribbon](image)

### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Soil calculation</strong></td>
<td></td>
<td>Using soil calculation group, it is possible to define the case of soil calculation, calculate the cutting and the banking, output and visualize the result.</td>
</tr>
<tr>
<td>Calculator</td>
<td><img src="image" alt="Calculator" /></td>
<td>Open &quot;Soil Calculator&quot; form.</td>
</tr>
</tbody>
</table>

### Noise propagation

This group has functions related to noise simulation. This is shown if "NoisePlugin (Noise simulation option)" is authenticated.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Criterion value</td>
<td><img src="image" alt="Criterion value" /></td>
<td>Open &quot;Criterion value for simulation&quot; window.</td>
</tr>
<tr>
<td>Sound Property</td>
<td><img src="image" alt="Sound Property" /></td>
<td>Open &quot;Character of sound source&quot; window.</td>
</tr>
<tr>
<td>Drawing options</td>
<td><img src="image" alt="Drawing options" /></td>
<td>Open &quot;Display options of simulation result&quot; window.</td>
</tr>
<tr>
<td>Sound Receiver</td>
<td><img src="image" alt="Sound Receiver" /></td>
<td>Open &quot;Sound receiving plane&quot; window.</td>
</tr>
<tr>
<td>Summary Receiver</td>
<td><img src="image" alt="Summary Receiver" /></td>
<td>Open &quot;Summary of sound receiving plane&quot; window.</td>
</tr>
<tr>
<td>Moving Sound</td>
<td><img src="image" alt="Moving Sound" /></td>
<td>Open &quot;Moving sound source&quot; window.</td>
</tr>
<tr>
<td>Summary source</td>
<td><img src="image" alt="Summary source" /></td>
<td>Open &quot;Summary of sound source&quot; window.</td>
</tr>
<tr>
<td>Load simulation</td>
<td><img src="image" alt="Load simulation" /></td>
<td>Open &quot;Browse for Folder&quot; window.</td>
</tr>
<tr>
<td>Run</td>
<td><img src="image" alt="Run" /></td>
<td>Select where to output and start the simulation.</td>
</tr>
<tr>
<td>Cloud</td>
<td><img src="image" alt="Cloud" /></td>
<td>High-performance computing on cloud services: Create a data file for noise analysis supercomputing service. (*.nad: noise analysis file format)</td>
</tr>
</tbody>
</table>

### Tsunami

Visualize results of external tsunami analysis program. This menu is shown if Tsunami Plugin is installed.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Editor</td>
<td><img src="image" alt="Editor" /></td>
<td>Open &quot;Tsunami Plugin&quot; window that displays tsunami data state. Visual option of the loaded tsunami data can be set.</td>
</tr>
<tr>
<td>Player</td>
<td><img src="image" alt="Player" /></td>
<td>Open &quot;Tsunami Player&quot; window.</td>
</tr>
</tbody>
</table>
**Debris**
Create input data for FORUM8 debris flow simulation and to visualize the analysis results.
It is shown if Debris Plugin is installed and activated.
*in Japanese only.

| Simulation | Open a window for data input and visualization. |

**Fluid**
Visualize flood analysis result of xpswmm.
It is shown if VTK (fluid analysis linkage) Plugin is installed and activated.

| Load simulation | Open VTK (fluid analysis linkage) Plugin window. Load the analysis result and visualize flow line data. |

**xpswmm Link**
Use this group to load CFD analysis result of fluid analysis tool "OpenFOAM" and display the flow lines on UC-win/Road.
It is shown if xpswmm Plugin is installed and activated.

<table>
<thead>
<tr>
<th>Water level settings</th>
<th>Water level settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pipework settings</td>
<td>Pipework settings</td>
</tr>
<tr>
<td>Pause</td>
<td>Pause</td>
</tr>
<tr>
<td>Run</td>
<td>Run</td>
</tr>
<tr>
<td>Stop</td>
<td>Stop</td>
</tr>
<tr>
<td>Skip previous</td>
<td>Skip previous</td>
</tr>
<tr>
<td>Skip next</td>
<td>Skip next</td>
</tr>
</tbody>
</table>

**Start/End time setting**
Analysis result data is longitudinal data. The start and stop time of animation can be set for the data output by xpswmm before the simulation or in pause. The start time can be set in the left drop down and the end time in the right. Unit is second (s).

**Simulation speed and simulation mode**
Simulation is output as steps per time history.

<table>
<thead>
<tr>
<th>0.100 s/model</th>
<th>Loop</th>
</tr>
</thead>
</table>

**Assessment Plug-in**
This plug-in enables an assessment on UC-win/Road VR.
This menu is available if "Assessment Plug-in" is in usable state. *This plug-in cannot be used together with SfM plug-in.

| Green coverage rate | Opens "Green coverage rate calc" screen. |

-35-
<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sunshine calc</td>
<td>Opens &quot;Sunshine Calc&quot; screen.</td>
</tr>
<tr>
<td>Reflected light check</td>
<td>Opens &quot;Solar panel reflection light check&quot; screen.</td>
</tr>
<tr>
<td>L-Tree</td>
<td>Opens the L-Tree editor. Create the 3D trees using L-System algorithm.</td>
</tr>
</tbody>
</table>
### Point Cloud Ribbon
This ribbon has functions that uses point cloud.

![Point Cloud Ribbon Image]

#### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Point Cloud</strong></td>
<td></td>
<td>This group has functions using point cloud data. This menu is shown if &quot;Point Cloud Modeling Plugin&quot; is authenticated.</td>
</tr>
<tr>
<td>Measure</td>
<td>![Measure Icon]</td>
<td>Measure between clicked 2 points.</td>
</tr>
<tr>
<td>Load data</td>
<td>![Load Data Icon]</td>
<td>Open &quot;Point Cloud Modeling Plugin&quot; window.</td>
</tr>
</tbody>
</table>

#### Dekigata
Use this group to control work progress. This menu is shown if "Dekigata Plugin" (Option in additional cost) is authenticated.

*in Japanese only.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D point cloud / Dekigata</td>
<td>![Dekigata Icon]</td>
<td>Open menu for Dekigata maintenance. Create a Dekigata maintenance form by calculating difference by design data and point cloud data.</td>
</tr>
</tbody>
</table>

#### SfM
Use this group to create 3D point clouds based on photo information. This menu is shown if SfM (Structure from Motion) plugin (Option in additional cost) is authenticated.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operation</td>
<td>![Operation Icon]</td>
<td>Open SfM main menu. Import analyzed SfM analysis case and analyze it actually.</td>
</tr>
</tbody>
</table>
Device Ribbon
This ribbon has functions to connect hardware with UC-win/Road.

Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Kinect</strong></td>
<td></td>
<td>Kinect group has functions to connect Kinect with UC-win/Road. This menu is shown if F8 Kinect plug-in (Option in additional cost) is available. *32bit version of UC-win/Road only.</td>
</tr>
<tr>
<td>Run</td>
<td><img src="Image" alt="Run icon" /></td>
<td>Open F8 Kinect Plugin window. Run/Stop the Kinect connection.</td>
</tr>
<tr>
<td>Help</td>
<td><img src="Image" alt="Help icon" /></td>
<td>Open the help menu of Kinect.</td>
</tr>
</tbody>
</table>

**Oculus**
Oculus group has functions to connect Oculus Rift with UC-win/Road. This menu is shown if Oculus Rift Plug-in is available.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run</td>
<td><img src="Image" alt="Run icon" /></td>
<td>Run/Stop Oculus Rift drawing.</td>
</tr>
<tr>
<td>Reset view</td>
<td><img src="Image" alt="Reset view icon" /></td>
<td>Click “Reset view” from Ribbon menu to set default coordination to current direction of Oculus Rift. Roll angle and pitch angle are calculated from the direction of gravitational force.</td>
</tr>
<tr>
<td>Setting</td>
<td><img src="Image" alt="Setting icon" /></td>
<td>Open Oculus Rift setting window.</td>
</tr>
<tr>
<td>Help</td>
<td><img src="Image" alt="Help icon" /></td>
<td>Open Oculus Rift help menu.</td>
</tr>
</tbody>
</table>

**Oculus DK1**
Oculus DK1 group has functions to connect Oculus DK1 with UC-win/Road. This option is shown if Oculus Rift plug-in is available.

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run</td>
<td><img src="Image" alt="Run icon" /></td>
<td>Opens the option window to adjust video output of UC-win/Road to that of Oculus. Run/Stop the Oculus drawing.</td>
</tr>
<tr>
<td>Help</td>
<td><img src="Image" alt="Help icon" /></td>
<td>Opens Oculus help menu.</td>
</tr>
</tbody>
</table>
### UAV
UAV group has a function to connect UAV with UC-win/Road (newer than Ver. 11.1).
This option is shown if UAV plug-in (Option in additional cost) is available.

<table>
<thead>
<tr>
<th>Show / Hide GUI</th>
<th>Opens UAV plug-in window.</th>
<th>Switches show/hide the UAV connect interface.</th>
</tr>
</thead>
</table>

### Gaze Tracking
Gaze Tracking has a function to connect an eye tracking device with UC-win/Road (Ver14.0 or later).
Gaze Tracking Plug-in is required.

<table>
<thead>
<tr>
<th>Start Gaze Tracking</th>
<th>Start Gaze tracking. Once clicking it, the icon changes to Stop Gaze Tracking.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Host IP, Port</td>
<td>Set up IP and Port of the host receiving gaze tracking information.</td>
</tr>
<tr>
<td>Use Mouse Point</td>
<td>Use mouse point to adjust the eye tracking data.</td>
</tr>
<tr>
<td>Option</td>
<td>Open the setting form.</td>
</tr>
</tbody>
</table>

### HTC VIVE
HTC Vive group calls functions that connect HTC Vive and UC-win/Road.
UC-win/Road scenes are displayed on HMD by linking HTC Vive with UC-win/Road by Valve's SDK (OpenVR), and the camera position in UC-win/Road moves according to the movement of HMD by using the Stream VR (TR) tracking.

<table>
<thead>
<tr>
<th>Run</th>
<th>Starts / stops cooperation with HTC Vive.</th>
</tr>
</thead>
<tbody>
<tr>
<td>HMD Display</td>
<td>Enable/Disable HMD display. It is available only after importing a UC-win/Road project file (*.rd).</td>
</tr>
<tr>
<td>Reset view</td>
<td>The coordinate system of the HMD can be initialized so the user can chose the direction of the front view. However, roll and pitch are chosen according to the direction of gravity. The standard of camera coordinate is decided based on a navigation mode so that the following directions match the default coordinate of the HTC Vive.</td>
</tr>
<tr>
<td>Setting</td>
<td>Displays the HTC Vive setting form.</td>
</tr>
<tr>
<td>Help</td>
<td>Displays the Help screen.</td>
</tr>
</tbody>
</table>

### FOVE
FOVE Group is to connect FOVE with UC-win/Road.

| Run FOVE. | |
|-----------|
| ![Icon] | Reset the view of HMD. |
| ![Icon] | Open the setting form of FOVE. |
| ![Icon] | Open the Help. |
### Server Ribbon

Use Server ribbon to operate about sever functions such as VR-Cloud.

![Server Ribbon](image)

#### Overview

<table>
<thead>
<tr>
<th>Command</th>
<th>Button</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A3S</strong></td>
<td><img src="image" alt="A3S" /></td>
<td>Settings to use UC-win/Road as the A3S server.</td>
</tr>
<tr>
<td>Options</td>
<td><img src="image" alt="Options" /></td>
<td>Open the A3S option window.</td>
</tr>
<tr>
<td>Enable/Disable</td>
<td><img src="image" alt="Enable/Disable" /></td>
<td>Enable/Disable A3S server.</td>
</tr>
</tbody>
</table>

**Rhino**

Rhino group has functions for operations related to the origin used in Rhino plug-in.

With Rhino plug-in, Rhinoceros® users can access to UC-win/Road in real time.

| New origin | ![New origin](image) | Define new origin in 3D space when some Rhinoceros® connect to UC-win/Road. |
| Edit origin | ![Edit origin](image) | Edit the origin linked with Rhinoceros. |
### Pop-up Menu

The following pop-up menu will appear by right-clicking on the drawing area.
The menu items will vary depending on the currently selected object and on the situation.

#### Pop-up menu when right clicking on the drawing area

<table>
<thead>
<tr>
<th>Pop-up Menu</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adjust Terrain Height</td>
<td>Opens the Vertices Height form.</td>
</tr>
<tr>
<td>Select Objects On Selected</td>
<td>Selects all objects within the selected terrain area.</td>
</tr>
<tr>
<td>Terrain</td>
<td></td>
</tr>
<tr>
<td>Generate forests</td>
<td>Open the display of forest editing</td>
</tr>
<tr>
<td>Delete Selected Objects</td>
<td>Deletes the applicable selected objects within the selected terrain area.</td>
</tr>
<tr>
<td>Duplicate Objects</td>
<td>Duplicates those 3D models selected within the selected terrain area.</td>
</tr>
<tr>
<td>Deselect Terrain</td>
<td>Unselects the terrain. Note: Any selected objects within the terrain area remain selected.</td>
</tr>
<tr>
<td>Create Terrain Patch</td>
<td>When 50m mesh has been selected, opens the Patch Size Editor form.</td>
</tr>
<tr>
<td>Edit Terrain Patch</td>
<td>When the terrain patch data has been selected, opens the Terrain Patch Editor form.</td>
</tr>
<tr>
<td>Delete Terrain Patch</td>
<td>Deletes the selected terrain patch data.</td>
</tr>
<tr>
<td>Edit Intersection</td>
<td>Opens the Intersection Editor form. Note: Some tabs are not available as editing these can require roads to be rebuilt or can affect moving traffic. To edit these tabs, open the Intersection Editor from the 2D Plan View.</td>
</tr>
<tr>
<td>Replace Intersection with 3DS</td>
<td>Opens &quot;Intersection Replacement&quot; screen.</td>
</tr>
<tr>
<td>Distance Between Models</td>
<td>Measures the horizontal distance between two models.</td>
</tr>
<tr>
<td>Jump To</td>
<td>Jumps to the position clicked. &quot;J&quot; key on the keyboard has the same function.</td>
</tr>
<tr>
<td>Input Coordinates</td>
<td>Specify the position by the value of local coordinate or world coordinate system.</td>
</tr>
<tr>
<td>Save Camera Position</td>
<td>Saves the currently displayed scene as a camera position.</td>
</tr>
<tr>
<td>Edit Camera Positions</td>
<td>Opens the Camera Positions form.</td>
</tr>
<tr>
<td>Move Camera To</td>
<td>Moves the camera position to another saved position.</td>
</tr>
<tr>
<td>Hide Camera Views</td>
<td>Hides or re-displays any added camera views.</td>
</tr>
<tr>
<td>Full Screen</td>
<td>Maximizes the view and shows as much display area as possible while hiding the title, menu, tool and status bars.</td>
</tr>
<tr>
<td>Show / Hide menu</td>
<td>Retains the original viewing size while hiding the borders, title, menu, tool and status bars.</td>
</tr>
<tr>
<td>Start/Stop AVI Recording</td>
<td>Start and Stop AVI file recording.</td>
</tr>
<tr>
<td>Show Simulation Tool</td>
<td>Hide / show the Micro Simulation Tool.</td>
</tr>
<tr>
<td>Show sound receiving plane</td>
<td>Hide / show the sound receiving plane of noise simulation. Micro Simulation Tool.</td>
</tr>
<tr>
<td>Show replay tool</td>
<td>Hide / show the replay tool.</td>
</tr>
<tr>
<td>Renderer setting</td>
<td>Select the rendering method of the main screen.</td>
</tr>
</tbody>
</table>
● Pop-up menus in other windows
Right click to open different pop-up menu.

● Combo box filter function
Click + button of Scenario on home tab to add scenarios 「111」「051」「222」「333」.
Input only "1" in the combo box where "111" is displayed to display only those that include the characters input "111" and "051". It make it easier to select.
Default Settings
It is possible to configure the application's operational environment from File Ribbon - Application Options - Default settings. Additionally, from File Ribbon - [Application Options] - [Location settings], it's possible to register and edit local settings.

■ [Application Defaults] window
This window contains the settings for UC-win/Road's operational environment. From File Ribbon then select [Application Options] - [Default settings] and this window is displayed.

● User Interface

【Language】
Select the language for the interface. The setting is reflected after restarting.

【When starting UC-win/Road】
The operation whether to load default terrain when UC-win/Road starts or to load a new project or to read the project preserved last time or not to read anything can be selected.

【Allow multiple instances】
Allow multiple instances of UC-win/Road. Otherwise, only one instance can be started at a time.

【When loading a project】
Check not to display an error dialog if there is no error on loading the data.

Preload Models:
To load all used models to the video memory after opening a project, rather than loading them the first time they become visible in the field of view.
This option is particularly useful for presentation or demonstration purposes as there is no slow down due to loading time during camera movements.
Note: When this option is activated, the initial project loading time is longer, so that do not select this check box if it is required to reopen the project.

【When closing UC-win/Road】
The screen setting (screen position and toolbar, etc.) is preserved when UC-win/Road ends if the check is put in "Save window settings", and those states of the screen (open all screens, screen sizes, and display positions, etc.) are restored when being start next time.

【Display a3s】
"a3s" group is displayed on "Server" Ribbon. Tick in the box when a3s related operation is required.
● Folders and files

[Data directory]
The storage place of the important data for the creation of projects. Do not change this data casually. UC-win/Road recognizes the data for editing here. It is necessary to reactivate for the reflection of the change.

[Use a secondary texture directory]
When the check is applied, the texture folder specified here, after it starts, can be used though only the file that exists in a pertinent folder before UC-win/Road starts can be used. The folder composition: the folder composition in the Textures following. It is necessary to make it to the same composition as UC-win/Road.

[Compress the texture when saving]
The texture is reversible compressed when exported. The whole data amount is reduced by compression. It is applied in exported of RD, RS or RM file.

● Editing

[Location Settings]
Select location settings. When the default terrain project is newly made, the selection here is applied. Traffic rule, the ringing control, and traffic characteristics, etc. in the region are included in this setting.

[Default leg length]
Input the length of the level crossing in the default intersection leg.

[Use default textures for new intersections]
Use the default textures when a new intersection is automatically created.

[Automatically generate new roads when they are created in the plan view]
Check to generate new roads automatically when they are put in the Plan View. If checked off, they are created when "Make Road" button is clicked.

[Reduce the texture size when the total size exceeds the limit]
Check to reduce a total size of textures down to 1GB when loading a project.

[Section Editor slope unit]
It is necessary to select the setting unit, angle or 1: N to set the inclination.
Simulation run options

This panel provides options for the simulation.

**Simulation time step mode**

Defines the time step used in simulation computation. Four modes are available: Adaptive variable (default), Adaptive base frequency, fixed [internal timer] and base frequency [custom timer].

**Frequency** :

Only available when time step mode is Fixed, sets the fixed frequency of the simulation from which the time step is computed (time step = 1/frequency).

The default value is 60Hz.

**Display synchronization mode**

Defines the synchronization mode between simulation and display.

Two modes are available: Synchronous (default) and Asynchronous.

**Simulation CPU mode**

Defines the CPU priority for the simulation computation. Two modes are available: Normal (default) and Time critical.

**CPU Affinity mode**

Selects how to schedule with CPU affinity. There are two modes available, "OS automatic" or "One per main thread".

**Reposition settings**

The reposition settings provide constraints on the reposition location and the behavior of the vehicle under certain conditions.

**Don't reposition if not on drivable ground surface**

Prevents the vehicle from repositioning outside a road carriageway.

**Don't reposition if another vehicle is at the same position**
Prevents the vehicle from repositioning where another vehicle already exists.

**[Force vehicle reset if slope before or after is greater than]**
Resets the vehicle status (speed, acceleration) if the road slope is greater than the specified value.

**[Force vehicle reset if slope difference is greater than]**
Resets the vehicle status (speed, acceleration) if the slope difference between the current road and the target position is greater than the specified value.

**[Don’t reposition if slope difference is greater than]**
Prevents the vehicle from repositioning if the slope difference between the current road and the target position is greater than the specified value.

**[Don’t reposition if one wheel is out of plane of more than]**
Prevents the vehicle from repositioning if, at the target position, one wheel or more are out of the vehicle wheel average plane.

**[Force vehicle reset if height differs more than]**
When repositioning, if the target position height is different from the current position height of more than the specified value, the vehicle status will be reset.

**[Don’t reposition vehicle with reset outside simulation pause]**
The simulation must be paused if the repositioning of the vehicle implies resetting its status.

**[Allow vehicle traffic random behavior]**
If checked, on the traffic simulation, we will make behavior of each vehicle random. The maximum speed during driving, the offset from the lane center and the vehicle weight at generation are set randomly.
If not checked, on the traffic simulation, the random behavior of each vehicle will not occur. The same behavior will be done under the same conditions. To get the perfect reproducibility, please fix the simulation time step mode.
Go to [File] Ribbon - [Application Options] - [Location Settings].

It's possible to create standard location settings for countries other than those that come pre-specified by opening the Location Settings wizard with the "New" button.
●Driver lane keep

The offset value of vehicles from the center of each lane can be adjusted by clicking "Edit" on Location Settings. The position of the vehicle in the lane varies at random in width during the driving simulation based on the calculation formula. Therefore the traffic congestion during the simulation appears more realistic.

*This is saved only for the existing project.


■ [Project Options] window

This is displayed by selecting [Project Options] from [File] Ribbon.

●Information

[TITLE]
Input the title of data for the current project.

[Description]
Input the description of data for the current project, if required.

[Copyright string] Input the logo shown at the right top corner as the following examples.

[Unique ID]
Click the "Lock" to edit.

[Image]
Thumbnail: The thumbnail of the data can be registered.
  • Update: Capture a screen shot of the current screen.
  • Load: Specify the image.
  • Clear: Clear the image.

Logo file: Specify the logo etc. shown at the right top corner of the main
It is possible to add a command line parameter to the shortcut to do one of the following at start up:

- Launch UC-win/Road in Full Screen mode.
- Start UC-win/Road without a project; reducing the time it takes to load.
- Automatically load a specific project.
- Start UC-win/Road with a view opened.
- Automatically launch a scenario.
- Overwrite the maximum number of cut and fill section point

**How to set the command line parameter**
1. Copy the UC-win/Road desktop shortcut icon.
2. Right-click the new shortcut icon and select Properties.
3. On the Shortcut tab in the Target field, add the applicable command line parameter at the end of the target (see below for a description):
   - `-full screen`  
   - `DONT_LOAD_DEFAULT_PROJECT`  
   - `LOAD=\"file path\"`  
   - `-leftview, -rightview, -rearview, -2Dview`  
   - `scenario=\"scenario name\"`  
   - `-OverwriteProjectCutFillPointNumbers=number of value`
4. Click OK.

**Simulation and design defaults**

**Location settings**
Displays the location settings file applicable for the current project. It is not possible to select a different location setting for the project. However some basic details can be edited such as default driving rules, traffic lights, and driver characteristics.
*This will not be supported for Location Setting file.

**Intersection default leg length**
If required, change the default leg length of intersections for this project by increasing or decreasing the value.

**Traffic flow algorithm version**

**Allow using offroad lane search algorithm at end of current lane**
It comes to be able to move between roads where the ramp is not connected by applying the offroad lane search algorithm at end of current lane to the road.
If selected **only for the current vehicle or for every vehicle in the traffic**, the algorithm is applied at the end position of the current lane. The road where it can drive as follows at the end position in this algorithm is retrieved by the origin point of the vehicle.
Note: If selected No, the function is ignored and it becomes impossible to move between roads where the ramp is not connected for the traditional algorithm to be applied.

**Allow jumping on other roads when overlapping geometrically:** If checked, it becomes possible to move to another road in the part where the road has intersected or the part that overlapped. Use this function when driving on the structure such as ramp and bridges on the way of the road. The minimum height is set. It moves if there is difference of elevation more than the difference of the value set here.

Note: This option can be checked only when only for the current vehicle selected

**Minimum height difference for jumping:** When jumping to another road, it is possible to move only to the road with a road that is higher than the road where driving now.

---

**[Road section end depth range minimum value]**

If the road surface and the vertical plane that is generated at the road ends and turning point is generated in the same position, using this parameter, it reduces the occurrence of the line to the section direction that occur on the road surface by. Adjusting OpenGL rendering parameter (Depth Range) directly is to reduce the line drawing to the section direction.

*Input range: 0.005 - 1.000*

**Hint:**

Two of polygon of the road surface and vertical surface as a line that the road surface and vertical surface is in contact with the road surface is displayed. Because it cannot determine the polygon to be displayed in the front if the two polygons at the same position is displayed, depending on the point of view, one of the polygons is displayed in a random manner. In order to reduce this phenomenon, slightly changing the depth information of the polygon, and displays the road surface on a priority basis. Usually the depth range is set between 0 and 1, but the depth range of vertical surface is set between the value set here and 1, and it changes a little bit in the back. The depth range is set by the minimum and maximum values, but the value set here is the minimum. The maximum value is fixed to 1.

---

**[Road microstep size]**

Shortest pitch (micro-step) to calculate the cross-sectional shape at the time of the road 3D model generation. The initial value is 10 cm. Using the distance set here, the cross-sectional shape of road is calculated. It checks the changes in the cross-section in the road axis direction, if it is not linear, the information to generate a new polygon is left.

**Hint:**

For the smaller value, the shape will be beautiful, but it takes a lot of time to calculate. For the greater value, the shape will be rough, the calculation time is shortened.

In addition to the micro step, it generates a polygon in the following step length regardless of the cross-section changes such as when it is on the transition and on the curve.

- Longest pitch: microstep x 100
- Longest pitch of curve: microstep x 20 to microstep x 100, depending on R
- Longest pitch in Transition: microstep x 40

---

**[Roads turning point overlap error threshold]**

On the Plan View, depending on the attribute of turning point adjacent to the road alignment, if the start and the end point of transition curve overlaps, they are usually treated as invalid turning point. But, by setting the acceptable range here, some overlaps are accepted. The input range is 0 - 50m.

The threshold is used as follow:

If the distance between the end point of a curve and the start point of the next curve is strictly greater than the threshold value, the turning points are valid. If the distance is smaller the turning points are invalid.
Depending on the data, the loss in precision due to the limit of the floating point precision may result in slightly different invalid turning point detection. Please adjust this parameter to solve this problem.

Ex) Curve 1 end point is common with curve 2 start point

**[Maximum number of cut and fill section points]**
Edits the maximum number of cut and fill section points for the cutting and banking. The default value is 1000(range: 2 - 1000). It stops generating the slope with berms, even if it needs more than this value to generate the slope.

Hint:
In the case the slope with the berm endlessly, it takes much time to generate the road. But reducing this parameter, the generation time will be shorter.

**Coordinate system**
Latitude and Longitude of the origin and ante origin for the current project can be changed here. The coordinates of the whole terrain can be changed, if required.

**Offset setting**

**Nothing**

North: 26257.180594

East: 56443.293588

Set up the offset values in North-South and East-West. These values are used to convert the global coordinate between local coordinate.

\[(\text{Global coordinate}) = (\text{Local coordinate}) + (\text{Offset})\]

**Reference coordinate setting**

Select a reference coordinate. Select one of the 5 below.

1. **Undefined**
The reference coordinate setting form is not displayed. It is not possible to convert to the longitude and latitude.

X is for the east direction and Y is for the north direction on the global coordinate.

2. **Japan Plane Rectangular**
Select a geodetic datum and the number. By selecting Undefined as a datum or a number, the coordinate may be incomplete, so that it is not possible to convert into the longitude and latitude. By defining the both, it is possible to convert.

X is for the north direction and Y is for the east direction on the global coordinate.
Datum: Undefined, Tokyo, JGD2000 or JGD2011
Number: Select 1 ~ 19 or check Undefined.

3. New Zealand Transverse Mercator
The reference coordinate setting form is not displayed. It is possible to convert to the longitude and latitude.
X is for the east direction and Y is for the north direction on the global coordinate.

4. UTM Grid coordinate system
Select the zone number and the Northern/Southern hemisphere.
By selecting Undefined as a number or a hemisphere, the coordinate may be incomplete, so that it is not possible to convert into the longitude and latitude. By defining the both, it is possible to convert.
X is for the east direction and Y is for the north direction on the global coordinate.

5. Linear Transform
Define the origin and the movement per 1 degree of latitude/longitude in east and north direction.
By not defining geodetic reference system, entering 0 as the movements or entering same movement direction of longitude/latitude, the coordinate may be incomplete, so that it is not possible to convert into the longitude and latitude.
The direction of X and Y in global coordinate system changes depending on the settings.

X-Y order: define the directions of X and Y. For North – East, X is for north and Y is for east. For East - North, X is for east and Y is for north.
Geographic coordinate system:  Undefined, WGS84, Tokyo, JGD 2000, JGD 2011 or NZTM 2000
Origin:  define the longitude and latitude of the origin.
Longitude 1sec vector:  define the distance per degree of longitude.
Latitude 1sec vector:  define the distance per degree of latitude.

**Terrain**
Input the terrain size.

**Program folder structure**
UC-win/Road Installed Folder¥

- Data: Data that does not need to be changed is stored here.
- DefaultPlugins: Default plug-in modules are stored here.
- GISView: GISViewer
- Help(CHS): Help document data (Chinese)
- Help(CHT): Help document data (Taiwanese)
- Help(ENZ): Help document data (English)
- Help(JPN): Help document data (Japanese)
- Help(KOR): Help document data (Korean)
- IFCSvr: Setup file of IFC server used for IFC plug-in is stored.
- MDS Steering Wheel: Driver setup file of old MDS Steering Wheel is stored.
- Plugins: Plug-ins are stored here.
- SetupDrivers: Setup files of each driver used for DS are stored here.
- Shaders: Shaders used in UC-win/Road are stored.
- SupportTool: A setup file of the inquiry support tool
- TeighaX: A setup file of TeighaX used in DWG plug-in
- ThirdParty Licenses: Information such as license agreement when using third party source code is stored.
- Uninstaller: Information of uninstaller created at the installation is stored.

**Data folder**
C:\UCwinRoad Data x.x¥

- Characters: Character model files
- Data: Japan.map, Tokyo/Osaka.map etc…
- Effects: 5m mesh data
- Locations: The Location Settings information
- Log: Logs
- Models : 3D model samples
  - ... : 3D model samples with texture
- Movie : AVI, frames, POV-Ray data
- Plugins : Plugins and files for system plugin
- Save : Sample data
  - FBX scene : FBX data
  - GISSampleData : GIS sample data
  - InRoads : InRoads sample data
  - Intersection : Intersection textures
    - MarkingsLib : Intersection texture editing markings library
  - LandXML : LandXML data
  - MicroSimSample : Micro Simulation sample data
  - Model : Samples of animated models
  - NoiseSimulation : Sample data of noise simulation
    - Highway
    - Kyoto
    - Nihondaira
  - Section : Road sections
  - SfMPlugin : SfM plug-in sample data
  - Shapefile : Shapefile sample data
  - Shoreline : Several lake preset files
  - TRACKSSamples : TRACKS samples
  - TrafficSaveSamples : Traffic snapshot sample data (.trs)
  - Tree : 3D trees
  - TRLVehicleModelsLibrary : Vehicle model data
  - xpswmm_Sample : xpswmm samples
- Scripts : VRCloud script samples
- Sounds : Sound files
  - AirFriction : Air friction sound files
  - Engine : Engine sound files
  - Hone : Horn sound files
  - Signals : Turn signal sound files
  - Tires : Tire sound files
- Textures : Textures
  - BackDrop : Background
  - Clouds : Clouds
  - Cockpit : Cockpit
  - Event : Event
  - Fire : Flame, Smoke
  - Flag : Flags
  - Model
3DS : 3DS models

Building : Buildings (walls)

Lighting : Artificial lightings (at night)

ParametricEscalator : Escalator textures in the parametric model

ParametricStairs : Stairs textures in the parametric model

Parkinglot : Ground textures for the parking lot model

RainOnRoad : Water puddle on the road surface

RainOnWindshield : Rainwater on the windshield

River : Rivers (used in previous version)

Road

Bridge : Undersides of bridge

Carriageway : Carriageway surfaces

CuttingBanking : Cuttings and bankings

Intersection : Grade intersections

Marking : Road markings

Obstruction : Obstacles

Surface : Road surface other than carriageways

Tunnel : Tunnel walls

Sign : Signs

Sky : Sky

SkyDome : Sky dome

Snow : Snow

Terrain : Terrains

Ground : Ground surfaces

Satellite : Satellite photographs, high precision aerial photographs

Streetmap : Spatial Data Framework maps

Tree : Trees

Bark : Trunks of 3D trees

Blossom : Flowers of 3D trees

Leaves : Leaves of 3D trees

Water : Rivers

Waves : Water surfaces

Video : Video file data used in scripts and video wall
Operation method

Operation with keyboard

If the camera mode is "Look", "Zoom", or "Move", it is possible to move the observing point and shift the line of sight by keyboard.

With keyboard.

<table>
<thead>
<tr>
<th>Key Combination</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>[↑] key</td>
<td>Advancement</td>
</tr>
<tr>
<td>[↓] key</td>
<td>Retreat</td>
</tr>
<tr>
<td>[←] key</td>
<td>Move left</td>
</tr>
<tr>
<td>[→] key</td>
<td>Move right</td>
</tr>
<tr>
<td>[Alt] + [↑] key</td>
<td>Move up</td>
</tr>
<tr>
<td>[Alt] + [↓] key</td>
<td>Move down</td>
</tr>
<tr>
<td>[Alt] + [Q] key</td>
<td>Speed up (+1 km/h)</td>
</tr>
<tr>
<td>[Alt] + [A] key</td>
<td>Speed down (-1 km/h)</td>
</tr>
<tr>
<td>[Alt] + [Z] key</td>
<td>Reset speed (3 km/h)</td>
</tr>
</tbody>
</table>

- Moving speed: 1 to 45 km/h
- If [Shift] key is pushed while moving, the passing speed becomes 3.3 times.

With keyboard.

<table>
<thead>
<tr>
<th>Key Combination</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>[Ctrl] + [↑] key</td>
<td>Face up</td>
</tr>
<tr>
<td>[Ctrl] + [↓] key</td>
<td>Face down</td>
</tr>
<tr>
<td>[Ctrl] + [←] key</td>
<td>Turn to left</td>
</tr>
<tr>
<td>[Ctrl] + [→] key</td>
<td>Turn to right</td>
</tr>
<tr>
<td>[Alt] + [W] key</td>
<td>Increase the rotation degree (+1 deg./s)</td>
</tr>
<tr>
<td>[Alt] + [S] key</td>
<td>Decrease the rotation degree (-1 deg./s)</td>
</tr>
<tr>
<td>[Alt] + [X] key</td>
<td>Reset the rotation degree (2 deg./s)</td>
</tr>
</tbody>
</table>

- Movement angle: Two degrees per second
- Setting range-turning angle degree: 2 to 30 deg./s.

With 10 keys.

<table>
<thead>
<tr>
<th>Key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Left front</td>
</tr>
<tr>
<td>8</td>
<td>Forward</td>
</tr>
<tr>
<td>9</td>
<td>Right front</td>
</tr>
<tr>
<td>4</td>
<td>Left</td>
</tr>
<tr>
<td>5</td>
<td>Right under</td>
</tr>
<tr>
<td>6</td>
<td>Right</td>
</tr>
<tr>
<td>1</td>
<td>Left rear</td>
</tr>
<tr>
<td>2</td>
<td>Rear side</td>
</tr>
<tr>
<td>3</td>
<td>Right rear</td>
</tr>
</tbody>
</table>

Drive On Road

<table>
<thead>
<tr>
<th>Key Combination</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>[↑] key</td>
<td>accelerate (+5 km/h)</td>
</tr>
<tr>
<td>[↓] key</td>
<td>decelerate (-5 km/h)</td>
</tr>
<tr>
<td>[←] key</td>
<td>move to left lane</td>
</tr>
<tr>
<td>[→] key</td>
<td>move to right lane</td>
</tr>
</tbody>
</table>

- Range: speed 0 ~ 1000 km/h, the viewpoint height 0.2 ~ 10 m.
- It is possible to ride in the front passenger seat of other vehicles, including vehicles generated by the traffic flow. It is also possible to drive with a steering controller on the passenger seat.

Satellite

- Ascend: [PageUp] key or scroll the wheel forward
- Descend: [PageDown] key or scroll the wheel backward

Flying (Free)

<table>
<thead>
<tr>
<th>Key Combination</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>[↑] key</td>
<td>Accelerate</td>
</tr>
<tr>
<td>[↓] key</td>
<td>Decelerate</td>
</tr>
<tr>
<td>[Shift] key</td>
<td>Double Speed (during flight)</td>
</tr>
</tbody>
</table>

Fly Flight path

<table>
<thead>
<tr>
<th>Key Combination</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>[↑] key</td>
<td>accelerate (+5 km/h)</td>
</tr>
<tr>
<td>[↓] key</td>
<td>decelerate (-5 km/h)</td>
</tr>
</tbody>
</table>

- Flight speed: 1 to 1000 km/h

Hint: How to change operation key arrangement depending on modes.

Edit operation keys from Home Ribbon – Navigation option. Refer to "Navigation Option" for more details.

*To operate driving simulation with keyboard, uncheck “Do not use the game controller” from File Ribbon - Application Options -
Game Controller Options

- **Change Vehicle**
  - Click Change Vehicle button in Home Ribbon.
  - Select a vehicle from the list of vehicle models and click OK button.
Navigation Options


It is not possible to change the global keyboard keys.

- Global Settings tab

**General.**

**Boost factor:** Multiplicator used on the movement speed of the camera when the Run/Turbo action is activated.
When inputting a value less than 1.00, the movement speed will be slow.

**Zoom factor:** Percentage applied to the current FOV (Field of View) when the Zoom is activated.

**Rotation speed:** Speed of the camera rotation controlled.

**Reverse mouse wheel:** Reverse actions of mouse wheel.

**Mouse Wheel speed:** Speed of mouse wheel speed/

**Free mode**

**Movement speed:** Movement Speed of camera with a keyboard.

**Flying speed:** Flying speed of camera.

**Jump duration:** Duration of animation during Jump.

**Mouse wheel axis:** Axis used for movements with a mouse wheel. (Left-Right/ Up-Down/ In-Out (Default)/ North-South/ East-West/ Vertical)

**Allow going through terrain:** Check to allows camera going through the terrain. (When a camera is under the terrain, it is possible to move freely. But if this option is unchecked, the camera can go through the terrain from below but not from above)

**Drive, Travel, Fly and Follow modes**

**Lock camera vector:** Lock a camera vector to rotate with a followed model (in Drive or Follow mode) or a point (in Travel or Fly mode).
· Keyboard Global tab

Run / turbo: Turbo the speed of camera movement while pressing the key. The speed is multiplied by the Boost factor. (If the Boost factor is higher than 1, the movements speed will be faster.)

Toggle zoom: Change the FOV (Field of View) while pressing the key. The current FOV is multiplied by the Zoom factor. (Available in all the Navigation modes)

Move forward: Move the camera forward. (Available for all Navigation modes)

Move backward: Move the camera backward. (Available for all Navigation modes)

Speed up/ Speed down: Increase/ Decrease the speed of the camera with a keyboard (available for Fly, Free, Travel and Walk modes).

Decrease height/ Increase height: Decrease/ Increase the heights of camera, travelling or flying point. (Available for Fly, Free, Travel and Walk modes).

Lock camera vector: Follow a model or a point with the same viewing angle (available for Drive, Fly, Follow and Travel modes with the Spin or Satellite camera move modes only).

Horizontal camera: Keep the camera at the same height, and disable all Up-Down rotations (available for Free and Walk modes).

Keyboard key to deselect object: Selecta SPACE or ESC.

Look behind left - Look ahead right: Change the looking direction while pressing the key. (available for Look camera move mode only)


*Numerical keys on ten keys are used as default.
- Free tab
Set up keyboards to operate in Free mode.

- Travel tab
Set up keyboards to operate in Travel mode.

- Fly tab
Set up keyboards to operate in Fly mode.

- Drive tab
Set up keyboards to operate in Drive mode.
Uncheck "Do not use the game controller" in Game Controller Options from File – Application options – Game controller options.
Click Keyboard Setting of Drive Simulation to open the setting from.

- Follow
Set up keyboards to operate in Follow mode.

- Walk
Set up keyboards to operate in Walk mode.

Walk mode
Standing height: Height of camera at human level.
Walking speed: Speed of camera movements
Crouched height: Crouched height of the camera
※Enter a key defined on Walking tab to crouch. It can be used to pass under obstacles.

- Bicycle
Set up keyboards to operate when driving a bicycle.

(Reference)
If "the force of pedaling" is set "30N • m", the speed of bicycle is 15.km/h. (On condition that the setting is default and the road is smooth.)
26 inch tire: 26 *2.54*3.14159265359/100 = 2.07470778843
RoadDataViewer Plug-in
RoadDataViewer plug-in lists the information of the object, texture and intersection etc. in tree in UC-win/Road project. And the data consistency can be confirmed or verified.

■ Screen of RoadDataViewer
Go to the ribbon, Edit - Road data - Viewer.

Elements on the project are listed. It is possible to edit them, and to verify them which meets the specified condition.

"Validate" button: Verify elements according to the condition of "Setting". After that, the elements which meet the condition are extracted.
"Setting" button: Set up the condition in detail
[?] button: Open "Help".
Right-click pop-up menu: Move viewpoints, rename or delete the selected element. It is also possible hide elements from the list via Kind Filter.

■ Setting form

Kind filter: Select element types extracted or listed.
Texture size: Extract all textures that match the criteria set here. Very large texture size may cause system performance issues.
Connect ramp: Calculate the distance between road connection points for each and every road, and narrows down the search to road connection points in which their distance is less than 10m.
Sunken model
Calculate how much the model is buried underground or underneath a road in terms of percentage, and narrows down the search to models with more than X (value entered in (c)) % of their body buried.
*By double clicking on a specific object in the list, the camera will automatically move close to it. Intersections and traffic connection points will always be viewed from the top.

*Example of the verification result
Before and After View for Model Visibility and Group Display

Go to Home Ribbon – Environment - Visibility, and display models for the situation. Then click 🖊️ to open Visible Models form.

On this form, check model to display, check off to hide for each visibility.

Example

Left: Before. The cherry trees, the yellow benches and green street lights.
Right: After. Display the greeneries, the green benches and the white street lights.

POV-Ray Options

Exporting for ray tracing is supported. Ray-tracing is a rendering technology which can generate a real 2D image by chasing a track in light from the viewpoint to the light source during simulation.

Rendering the exported file with the freeware "POV-Ray (Persistence of Vision Ray-Tracer)" by POV- Team to increase the realism of 2D image
Each setting for ray-tracing
Model Editor - POV-Ray tab
It is possible to set the Texture of the model for POV-Ray, as well as the light.

Texture Setting

Check "Override texture" option after selecting model parts under "POV-Ray" tab of the "Model editor" to generate the default texture.
If the texture for POV-Ray is set for the fake light, blinker or brake light, it cannot be output.
Check "ignore the fake light" option to output it, however, it is effective with the Blend mode only.

Light Source Setting
Check "Is Light Source (only at night)".
The default is "Spot light of Fade distance 8m".
The light source is situated at the gravitational center of the model parts; hence, depending on the model parts, light source is sometimes absent. In this case, check "Cast No Shadow" option or check "Override Light Source" to edit the language of scene description.

Example
With enclosed light source
With "Cast No shadow" option

Mist Setting
Mist is caused by diffusion of light with fog.
Check "Light Has Mist Halo (only at night) to generate a halo.
Choose model parts which are set for the light source under "POV-Ray" tab of "Model Editor" form. The default is "Type2", which is a condition of very small drops of water.
Lake Setting

To generate the default texture for POV-Ray, click POV-Ray in “Edit shoreline” form and check “Override Shoreline Texture” option.
**POV-Ray Option Settings**

Go to the ribbon, Record/Play – Movie – Option, and click POV-Ray Options button in the Movie Manager Options form.

### Option tab

**Sun Casts Soft Shadows**: Express soft shadows.

**Use Radiosity**: Activate radiosity.

**Maximum Trace Level**: Solve the phenomenon that the black area is generated when “Black in texture is transparent” is enabled.

**Visibility**: Limit rendering object in the radius from the viewpoint position. Time taken for rendering is minimized.

**Gamma Value**: 4 kinds of mist

### Mist tab

**Ignore Local Mist From Model Light Source**: The mist is ignored.

**Use Global Mist**: Use the global mist.

**Override Mist**: Edit the mist by the description language.
Exporting POV-Ray File

Go to the ribbon, Record/Play – Movie – Export to POC-Ray* to export the current scene to **User data folder** ¥Movie¥POVRay¥Scene_***

Launch POV-Ray and open ***.pov in the Scene folder file or double click ***.pov file.

Customize properties and click RUN button to start rendering.

The file is saved to the folder of original SCENE with a file name as "UCwinRoadScene.bmp".

Depending on settings, it will take long time to render. It is recommended to test a small image first of all.

Sample data. (FORUM8 CG Movie Service: Rendering using Super Computer)

Note:

Check export POV-Ray in the Movie options to render each frame of AVI to the POV-Ray.

Go to Ribbon, Record/Play - Movie - Export Static Models to POV-Ray, and then go to Ribbon, Record/Play - Movie - Start to start AVI recording for POV-Ray. After the recording, open and render the ***.pov file in Frame******, file by file.
Context

The environmental setting is reflected at once only by preserving various settings in one context. An environmental change and the comparison of the simulations can be easily done by preparing and switching two or more contexts.

The context edit screen opens by ribbon Home - Environment - Context editor

The following element can be set.

・All conditions in visual options
・View options
・Currently selected design
・Traffic settings
・Pedestrian Network

Click on “Current settings” - “Save as new” and a name of the content of the current main screen to save current context.
Basic – Road and terrain
1. 3D Elevation Data Selection, Reading and Editing

1. Terrain data acquisition

There are some possible ways to acquire terrain data.

(1) Use default topographical data.
(2) Use the Geographical Survey Institute's 50m mesh
(3) Use the 50m mesh of New Zealand (provided by GeoGraph)
(4) Use topographical data of other countries
(5) User defined terrain (using a blank map)
(6) Import from file
(7) Load the GSI tiles

2. Reading terrain data

(1) Use the Geographical Survey Institute's 50m mesh
Go to "File" menu - "New Project" - "Japan"
A complete map of Japan with 50m meshes is displayed.
If the 50m mesh fails to load...

Copy the file from the CD-ROM and restart the program. It is also possible to read from the 50m Elevation Mesh Data Japan CD-ROMs I - III.
Any part of the mesh can be selected. Each side of the division is 10km

- One division 10x10km  
- Two divisions in length 10x20km
- Two divisions in side 20x10km  
- Two divisions in length and breadth 20x20km.
- Latitude longitude
- The world land surveying system  
- Old geographical coordinate system

The first numerical value of four digits shows the first region division and the second region division is shown by the following two digits.

Select an area and click OK to display the terrain.

About 50m mesh data
The mesh data Japan I - III of numerical value map 50m altitude is arranged, saved in C:\YUCwinRoad Data xx\YData folder as Japan.map file and used. This file is retrieved priority when retrieving it. When this file doesn't exist, retrieve CD-ROM drive. Importing the original CD is now supported, so set it to CD-ROM drive if required when the map data is changed etc. In this case, change the name of Japan.map saved in \YData folder or delete or move it.

- Use 5m mesh of Geospatial Information Authority of Japan
There are two methods of loading the digital map in 5m mesh (Elevation) provided by Geospatial Information Authority of Japan into UC-win/Road. For either loading method, 5m mesh data is converted into the terrain patch data. Once the data is taken as the terrain patch data, street map, road, building etc. can be placed on top of this terrain.

Note: The 5m mesh data that can be imported directly into UC-win/Road is only “Digital Map 5m grid Mesh (Elevation)” published by Geospatial Information Authority of Japan. Users who have CD-ROM or other devices with 5m mesh data can use this function. To import the basic map information that can be downloaded on the Web site of GSI, the data must be converted to the Shape file before doing.

- How to import simultaneously when creating a project
Load a terrain and create a project at the same time. It is possible to load the 5m mesh data of Japanese terrain.
1. **Save 5m mesh data into the folder**

Copy the data of 5m mesh (*.lem, *.csv) into arbitrary folder. Since the following folder is seen first by default on a program side, it is convenient to save it in the folder. «User data folder>>¥Data¥5m

2. **Select the 5m mesh data to load**

Select Main menu - [File] — [New project] — [Japan], and the Load Terrain Data window will open. On the window, click on button. Then select the 5m mesh data (*.lem) to load in the File Open dialog. Tip: it is possible to select multiple files by holding down the Ctrl key or Shift key while clicking the files. The selected mesh data is loaded. The area of 5m mesh data has red diagonal lines.

3. **Select the loaded 5m mesh data**

When the terrain of 50m mesh is selected, it is possible to view a zoomed-in image of the selected area on a different window where the mesh (1.5km x 2.0km) of 5m mesh elevation data is displayed. On this window, click the area of 5m mesh to load.

4. **Set the terrain patch size and load**

Click OK on the Load Terrain Data window to open the window for inputting the terrain patch size. Click OK after inputting the average terrain patch size. The 5m mesh will be loaded as terrain patch data each having this size.

The geographical features of 50m mesh

The geographical features importing 5m mesh
● Importing method after creating a project

Import the 5m mesh altitude data from the state that terrain has been already imported.

1. Save 5m mesh data into a folder

Copy the data of 5m mesh to the arbitrary folder. Save it to <<User data folder>>¥Data¥5m.

2. Select 5m mesh data to import

Select Lem file from Main ribbon, Edit - Terrain – Patch▼ - LEM File.

3. Setting and importing the terrain patch size

When clicking [Open] button, the screen to input the patch size is displayed. When inputting the 1 side of terrain patch data and clicking "OK" button, the 5m mesh is imported as a terrain patch data for each size.

(2) Use the 50m mesh of New Zealand

From the drop-down menus, go to "File" - "New Project" - "New Zealand"

Load the terrain of New Zealand provided from GeoGraph.

As there is only one coordinate system for the New Zealand map, there are no functions to change coordinate systems.

(3) Use the terrain data of other nations.

Terrain data in China and Australia in geographical features DB*CGIAR-CSI SRTM 90m Database* can be selected. For instance, select the region like "All parts of the world" - "China" - "North East" in the above column when reading geographical features in China and display the map.

Select the terrain below:

- 13km×13km
- 13km×26km
- 26km×13km
- 26km×26km

with the button to select the area.

*To create another countries project;

It is required to be saved in <<User data folder (C:\UCwinRoad Data xx by default)>>¥Data folder. If was not copied when installing it or it is not in the folder for some reason, copy map file and wap file from maps folder of products media (CD-ROM or DVD) to <<User Data Folder>>¥Data folder and activate the program.
From the drop-down menus, go to "File" - "New Project" - "User Defined"
For user defined terrain, choose from four different areas:
10km x 10km
20km x 10km
10km x 20km
20km x 20km.

(4) Optional terrain (using a blank map)... User defined

(5) Use the elevation data file
Under "File" menu – "New Project" – "Load From File", choose to load assorted files of terrain data for a new project. The terrain data supported also include seabed data, making the recreation of a containing ocean floor also possible. There are several reasons why one would want to use this method when creating a new project.

1. More freedom in terrain size
By loading terrain data that contains multiple elevation data, users are free to specify which particular area they choose to use in the project. In contrast, the project created by other method is limited to sizes such as 10km X 10km or 20km X 20km.
2. Specifying the area to be generated

By defining the longitude & latitude, or simply a rectangular area, a certain area can be generated regardless of aspect ratio or coordinates.

3. Numerous data formats are supported for terrain elevation SRTM90 (90m mesh), ASTER (30m mesh), image files, etc. It is also possible to integrate different multiple data into one and synthesize a terrain.

Note: Some of the data contained are only free under the circumstances that they are used for research purposes. Please consult their appropriate author before engaging in commercial activities.

4. Terrain Data Filtering

The polygon of the generated terrain may be freely altered. The elevation of the polygon vertices are complemented through cubic interpolation to maintain high-detail terrain generation.

- [Load]

To import elevation data files, click "Load..." button to select the files.

On the map, the entire loaded files are displayed. When selecting one of them on the list, the corresponding position on the map comes to the center.

- [Clear]

Clicking "×" on the list to close all files by clicking the "Clear" button.

- [Origin and scale]

Most elevation data file format contains information about the location on earth as well as their size. If the elevation file that has been imported does not contain this information, it is possible to define the origin and scale of each elevation data file. To do this, select a file from the list and make sure "Allow override" is selected in "Origin and scale"

  - Origin: Use the latitude and longitude inputs to define the location of the top-left corner of the elevation data file
  - Altitude: Use this to define an altitude offset
  - Scale: Use the latitude and longitude inputs to define the size of east-west and south and north of the elevation data file.
*When the loaded data is put at the wrong position, edit the origin to move it at real place.

- **[Altitude range]**
  - **Ignore special value**
    A special value meaning that the elevation file is not valid for a given position can be set (it is set automatically if the format of the elevation data file supports this feature). Typically, this special value is used for on the sea.

  - **Restrict altitude range**
    It is possible to set the data as invalid in a particular point if it is not within predefined boundaries. This is useful to mix several elevation data files together: one file can be used only for the value above sea level and another one only for the value under the sea level.

- **Project properties**
  Once all the needed elevation data files have been imported, it is possible to choose which area of the terrain will be used in the project. To do so, the “Project properties” tab is used.

- **[Project location settings]**
  This list is used to display and choose which location settings will be used when creating the project.

- **[Coordinate system]**
  The longitude/latitude treated WGS84 and WGS84 UTM zone coordinate system is generated. When ‘Calculate from area center’ is checked, it is calculated automatically. When unchecked, assign the zone number and the hemisphere (N- northern, S- southern)

- **[Terrain generation]**
  - **Terrain resolution** : This defines the precision of the terrain. Higher values provide better resolution but implies a slower terrain rendering.
  - **Interpolation** : This defines the algorithm used to interpolate elevation data to the terrain of the project that is created. By this set, the different size of data is connected smoothly each other. Select one from “None” (nearest) / “Bilinear” / Bicubic”.

  ![Terrain Interpolation Options](image)

  - **Default value** : Some elevation data have holes and this value can be used to fill those holes.
  - **Missing data on borders** : This defines the policy used in the case some data is missing on the border. "Hide polygon" / "Use default value
  - **Holes in data** : This defines the policy used in the case some data is missing in the area. "Interpolate altitude" / "Use default value"
To select the area of the terrain to be used in the project, simply click on "Select terrain with mouse" button and draw a rectangle on the map with "Rectangle Select tool".

It is also possible to precisely define the Origin and Anteorigin of the project by directly inputting the latitude and longitude.

▲ Example of Data of 125km (east-west) x 113km (north-south)

■ Raw Binary Elevation Data File Form

Most of their formats include headers that have information about their contents, but sometimes elevation data is distributed using raw binary files. A raw binary file is a file that only includes elevation data, without any additional information that can be used to know the size, scale or location of the data. If the file cannot be read, create a descriptor for it.

If the file contains raw binary elevation data, it is possible to define its descriptor manually about the elevation data.
Define the data size and layout.
- **Width**: The number of points that are in the file (east-west)
- **Height**: The number of points that are in the file (north-south)
- **Header offset**: The number of offset byte in case the file contains a header that has to be skipped
- **Data type**: The way data is stored in the file
- **Byte order**: The way to order byte

Define the origin and scale of the data. The scale in seconds per pixels, so if the data has a width of 100 pixels and the scale at 3 seconds per pixel, it means the data width is 300 seconds (= 5 degrees)

(6) Use external 3 dimensional terrain data

Merge existing terrain data with the selected 50m mesh from "Edit" Ribbon – "Patch" from Terrain.
Click "Open" and load the terrain file previously created.

Check "Merge with Terrain" to optimize the original terrain and the terrain data. The data to be used is displayed in green and the data to be ignored is displayed in grey.

**Ignore Closest**: Ignore data with an altitude similar to the terrain patch data.

**Ignore Square**: Ignore data inside a square made by the terrain patch data’s elevation points.

**[Triangulate] button**: Apply the terrain patch data to the existing data and generate the terrain's polygons.
Generated terrain data is displayed on the left.

If data is converted from terrain data such as contour lines, it becomes extremely detailed. The generation of terrain converted not just in the area around roads but other much larger areas takes a while, so it’s not really recommended. It’s recommended that the 50m mesh be used as it is in areas that are a long way from the target area.

**About the terrain patch data format**

To import terrain data created by an external tool, it must be in the prescribed XML file format. Please refer to “Making of External Terrain Data” in the Help files.

If using the separately available “UC-win/Road Data Conversion Tool”, plan data (DXF files) and layer data (*.dat) can be converted to the prescribed format (XML file).
(7) Loading the GSI tiles

1. Go to File – New Project – Importing GSI tiles, open the [GSI tile setting] screen. It is required to connect to the Internet.

2. Expand the range to create. It is also possible to specify the latitude and longitude.
   - Drag: Move the map
   - Mouse Wheel: Zoom in/out the map
   - Up and down, left and right arrow: Move the map
   - +: Enlarging the map - -: Shrinking the map
   The zoom level is displayed in the menu bar in the standard map.
   - Maximum zoom level: 17 - Minimum zoom level: 2

3. Standard map, Reduced shade map, Photo (After 2007), and Photo (1974-1978) are available.
   - The "Photo Map (After 2007)" takes advantage of the "electronic national land based map (ortho-image) of the Geographical Survey Institute tiles" that GSI has published to the public.
   - for the "Photo map (1974 - 1978)", the published article, "National Land Image Information (first phase: 1974 - 1978)", is being used.
   - "Photo Map (After 2007)" and "Photo Map (1974-1978)" only support a zoom level range of 9 - 17, so in the case of a zoom level of 9 or less cannot be specified.
   - "E-Land, Infrastructure and basic view (ortho-image)" contains some areas, such as mountainous regions for which images have not been captured, As such, there are areas for which no images will be displayed.

4. Click the button to begin the processing of information, and display the terrain data and map information based on the current settings.
3. Editing terrain data  
(1) Changing the elevation for each 50m vertex

1. Click on the terrain to modify 
   (A yellow triangle is displayed).
2. Click on a vertex (it will be displayed in yellow).
3. Change to the desired elevation.

To edit the elevation, move the vertex.
Click Edit - Scene - Plan View - Build Roads, and then click OK to edit the terrain.

* Cutting and banking are not generated at times, even if clicking "Edit Layout in Plan View" then "Build Roads".
In this case, open the Vertical Alignment editing window for the road and click OK.

* Click ［More >>］ button on the Terrain Vertex Editor to display the detail of the position.
  - Local XY
  - World Coordinate System YX
  - Latitude / Longitude
(2) Editing multiple terrain polygons

1. Select a terrain by left clicking while holding the Shift key to select consecutive triangles or by whilst holding the Ctrl key to select them one by one.

2. Relative movement and leveling of land can be performed by right clicking and selecting "Adjust Terrain Height".

**Delta Height (Relative Height):** The vertical interval of each altitude point from a present altitude is input in every the meter.

**Absolute Height:** The altitude of all the altitude points is input in every the meter.
(3) Arbitrary mesh modifications

1. Click on the terrain to modify (A yellow triangle is displayed).
2. Right click and select "Create Terrain Patch".
3. Select the target area by clicking with the mouse in the Patch Size Editor window. 1 mesh in this window is 50m on each side.

4. Add a point by clicking Add Points.
5. Right click on a point and select "Edit Point".
6. Enter the coordinates and elevation. Alterations to the point’s position and elevation can be checked in the 3D display on the right. The triangulated polygons are automatically generated in this window. The data can be saved as an XML file with the save button and read anytime.

7. Click OK and the terrain is regenerated. It can be re-edited or the elevation of the vertices can be edited.
It is possible to display roads and rivers in the terrain editor
It is displayed with a red frame in the Plan View window.

It is also possible to edit the vertical control point collectively.
To select all at once, hold the Shift key and Drag. To select multiple elevation points, hold the Ctrl key and Click.

8. If there's a road and the edited terrain has been cut or banked, please open the "Plan View" window from the main ribbon, "Edit" - "Scene"
After selecting multiple points, right click on any location, and select "Edit Selected Points".

Edit the elevation on the displayed Terrain Patch Editor.

When patch data exceeds 2000 points, elevation points are not displayed in the 3-D view on the right hand side. Editing is possible by clicking "Ignore Limits".
(4) Combination (merge function)
Combination of data with the same location and size is possible through the merge function.
Merge by selecting "File" - "Merge..." and selecting the data.

▲Merge roads and 3D models

It is possible to merge separate working data such as roads, models and rivers.
Note: Cross-sections, aerial photos, camera positions, etc., are also combined. However, please note the possibility of data overlap
2. 2D Image Data Selection / Loading

The following mappings are supported:

- 2500m spatial infrastructure data value maps from the Geographical Survey Institute
- Detailed data value maps from the Geographical Survey Institute (10m mesh of land use)
- Ministry of Land, Infrastructure and Transport DM (digital mapping) files
- Aerial Photos (BMP image .bmp or JPEG image .jpg)

Aerial photo grids are treated as layers. It is possible to create multiple layers, select them individually, overlap, rearrange, delete, or split them.

1. 2500m spatial infrastructure data

Go to "Edit" Ribbon - "Terrain", and then click "Street Map".

Select "Open Street Map File".

Supported street maps (the black mesh frame) are displayed.

If a message "An incorrect coordinate system has been detected" appears, it does not correspond to the selected file. Change the settings.

Click on mesh frames to display the street maps.

Click "OK" to finish it.

The spatial infrastructure data for Tokyo and Osaka is saved to Tokyo.map/Osaka.map, which is Tokyo [MapFile] / Osaka [MapFile]. For other locations, click "Use CD" from map selection drop-down menu.

* If the map files are not installed, Use CD.
* When data belonging to more than one prefecture is included on CD, the files cannot be read. Copy all the folders for necessary prefectures into the Data folder and run UC-win/Road. Select "Directory Name" in the "Maps" column.

2. Load DM data

After Ver.6.1, it is possible to load the data from other than the specified folders. In case of Ver.6.0.2 or earlier, create the following folder directory.

UCwinRoad /Data /DM /new folder (area name, etc.) - dm

Also, ****.dm and INDEX.file have to be in the dm folder.

2-1. Folder Preparation

Copy the DM Data from CD to the DM Folder in /Data. (UCwinRoad/Data/DM)

e.g.:00abc123

2-2. A DM file and INDEX file are saved in each folder.

UCwinRoad/Data/DM/00abc123….07nd815.dm

…Index.file

/00abc456….07nd816.dm

…Index.file

2-3. Select DM from "Load Street Maps" and click on the search button.

When the coordinate system is different, a message "the coordinate system is different" appears.

When loading is completed, the blocks are displayed as red outlines.

2-4. Clicked on the red outline to load and display the DM information.

2-5. Save the data in XML.

DM data loading example

Click on displayed grids to load the data into it. Note however that the Display 3D Data button is only enabled if the data contains 3D terrain information.
After importing, only terrain data can be displayed with 3D button. When displaying the information other than the terrain, click button again.

"Display Option" can change the color of display element.
3. Load satellite photos

It is possible to paste aerial photograph (air photograph) one another from the top regardless of with or without pasting the street map and the mesh size. There are 2 methods.

(1) Add a street map grid

Click "Add a street map grid" button.

Size
Width, Height: Set the number of blocks to correspond to the number of sections of images.
Automatic size: the size will be adjusted automatically.

Grid position: input the coordinate of bottom-left and top-right of the grid.
- Local: Coordinate for input of UC-win/Road section
- Global: Position in World Geodetic System
- TKY: Position in old geodetic system
- DMS: Coordinate in DMS format.
- DD: Coordinate in DD format.
Load street map

Click Load street map. Specify 2 points in an image to paste it.

Click on the allocated grid to open a list of available images. Select an image and press OK.

When using the image in a folder which is different from the displayed path, specify the path with "Retrieval of Street Map Data".

Note
To load own aerial photos, save the photograph data in the Satellite folder. Multiple images can be saved in the same folder. If the error message "insufficient memory" is displayed, the data cannot be saved. Please save the data prior to loading aerial photos. Restart UC-win/Road after loading the images and assign the images one by one.
4. Remove Street Map

Click "Delete" to change the cursor to .
Delete mode starts. Click cells to delete.
Click and release the delete mode.

Right click on a cell to delete and select “delete”.
- Delete the layer: Delete the selected layer.
- Delete the cell: Delete only selected cell.
- Delete the picture: Delete only the image and leave the cell.
- Delete unused cell
  *It can be selected when there is a cell that an image is not put the unused cell in a layer is deleted.

5. Edit layer

Right-click the layer to open the popup menu.

■ Add
- Add a geolocalised file in the selected layer. Select this menu and select a file in the file dialog.
- Add a street map grid in the selected layer. Select this menu and make a grid in the Satellite Picture Grid form. Once a grid is created, the grid will be displayed in the selected layer.

■ Arrange
To change the Z value of a layer, select one of the menus.
- Bring to front: Z value will be the highest.
- Bring to frontward: Z value will be incremented by one.
- Send to backward: Z value will be decremented by one.
- Send to back: Z value will be the lowest.
■ Edit

- Edit the layer offset

Open the Street Map Layer Editor window. Enter relative offset values or target coordinate values in Local or X-Y coordinate. And then Click OK to move the layers to the specified position.

- Divide into cells

Right click on a grid box and input the desired number of rows and columns. This is useful when trying to limit to a certain area to be precisely displayed (such as an aerial photo). On the other hand, one can also choose to delete unnecessary place. The following pictures contain splits of different columns or rows for easy reference.

6. Street Map Options

Click "Street Map Options".

Set up detailed items and change the display color.
7. Divide street map
When the size of one image is too big, divide it and assign only the required sections.

1. Go to the ribbon, "Edit" - "Terrain" - "Dice streetmap"
2. Select the image to be divided
3. Specify the vertical and horizontal distribution value.
4. After division, Select from the street maps.
*Note: The file is named automatically.

8. Use of world file
When the world file (bpw or jpgw) including coordinate information is selected by reading the street map, the attribute of the corresponding image can be set. UTM information of present terrain feature is displayed on the screen.

- Specify "Open Street Map File" or "Open Street Map Folder"

- The world file prepared with the same name as the image is opened. The content can be confirmed and edited with text editor because the world file is text format. The extension becomes jpgw when it is with JPG image and becomes bpw when it is with BMP image.

*Example: Image (JPEG) F030922905.jpg, World File F030922905.jpgw

Finally the image is pasted on the position to be specified by world file.

Example of world file
0.488: X Scale: Length in direction of X of 1 pixel (m)
0.0: Angle of rotation in direction of X (Ordinarily : 0)
0.0: Angle of rotation in direction of Y (Ordinarily : 0)
-0.488: Minus Y scale: Length in direction of Y of 1 pixel (m)
546106.406: East longitude coordinate on the center position of the upper left pixel of the image
3838995.037: North latitude coordinate on the center position of the upper left pixel of the image
3. Horizontal Road Alignment
1. Road plain view

Click "Home" Ribbon - "Edit" - "Open Plan View", or go to "Edit" Ribbon - "Scene" - "Plan View".

Toolbar

- Load spline road or polyline road.
- Zoom In
- Zoom Out
- Full Size
- Area zoom
- Move views
- Draft mode (The road formation time is reduced as the cutting, banking and intersections are not regenerated.)
- Re-build Roads
- Road building options (Select roads to build)
- Make Road (Make New Railroad, Add New Roadway, Add New Track)
- Make Road Spline
- Make River
- Make flight Path
- Male Polyline
- Make Backdrop
- Make Cross Section
- Make a lake
- Shift Objects
- Place 3D model

: This is available for invalid intersections.

Layers

- Display or hide below layers in the plan view.

Preset: Display/hide all layers or display only terrain, roads or river etc.
The color corresponds to the altitudes of the topographical map. The center of the circle indicates the current camera position. The angle of the fan represents the angle of the view. Camera position can be moved by double-clicking anywhere in the plan view. Or right click and go to “move camera” - “change camera view”

**Status bar**

The coordinates of the current cursor position is displayed in this format.

1. The position starting from the bottom left of the topographical map in meters (X-direction, Z-direction)
2. The current longitude and latitude
3. The position in the coordinate system according to the world geodetic system in meters
4. The position on currently selected road, river, etc.
5. The name of currently selected road

**Pop-up menu in Plan View**
2. IP entry method and spline road alignment entry method

Defining horizontal road alignments
There are 2 methods.
(1) IP point coordinate entry: Entry of the road alignment by IP method
(2) The central point coordinate entry: Entry by the spline road alignment which passes a specified point

(1) IP point coordinate entry

1. Click on "Make Road" button , or go to "Start" - "Start Road".

2. Click on the terrain to start road, create bends and end road.

3. Click "Make Road" once again or right click and select "Finish Road.

The "Starting Point" is created at the point at the point clicked first and the "End Point" at the point clicked last. The dots in between are “Turning Points”. As it is difficult to place these “points” at the correct coordinates, place them at the approximate locations with the correct number of turning points.

4. Right mouse-click on the "point" to and go to Edit - turning point to set its position, type and parameters.

- Position

■ Local (Unit: m)

The bottom left corner of the current mesh data is the origin.
X-axis: West to East (horizontal), from the very bottom left corner
Y-axis: South to North (vertical), from the very bottom left corner

■ Globals (Unit: m)

(In case of Japanese terrain and Japan.map 50m mesh data)
JGD (World Geodetic Datum 2000, 2011)
X-axis: same origin and meridian as the world geodetic datum.
Y-axis: axis that is perpendicular to the a-axis

(In case of New Zealand terrain and NewZealand.map)
New Zealand Unique Coordinates (NZTM2000)
Origin: 173°E, 0.0°S
X-axis: axis that overlaps with the meridian
Y-axis: axis that is perpendicular to the x-axis
(For Wap files, SRTM, ASTER data): UTM Coordinates
(For others): Linear convert
TKY (Unit: m)
Old Japanese Geodatic Datum (Tokyo Datum)
X-axis: old Geodatic Datum specified origin, same as meridian
Y-axis: perpendicular to x-axis
Do not apply to countries other than Japan.

DMS
Longitudinal and latitudinal position via degree, minute, second format
Longitude: angle measured from the Greenwich Meridian Line. Range -180 – 180 degrees
Latitude: angle towards North or South Pole from the equator. Range -90 (south) – 90 (north) degrees.

DD
Longitudinal and latitudinal position via degree format
Longitude: angle measured from the Greenwich Meridian Line. Range -180 – 180 degrees
Latitude: angle between North and South Pole from the equator. Range -90 (south) – 90 (north) degrees

Type
Railroads are created in a similar way to roads by specifying either cubic parabolic curve or half wavelength sine curve as a transition curve (do not apply to clothoid curves).
For horizontal railway alignment, when setting to the 2 curve types above and when the turning point type is “transition curve - circle - transition curve”, the transition curve will be automatically become the curve shape specified.
- Transition curve - Circle - Transition curve
- Transition curve
- Circle - Transition curve - Circle (currently not available)
- Circle - Transition curve
- Circle
Depending on the turning point, select the curve type.
*Note: Only “transition curve-circle-transition curve” and circle are available if cubic parabola or half-wavelength sine selected.

Parameter
Enter parameters for the selected road type. It depends on the type of alignment.
Aa=Ab: Both parameters are the same. The program will automatically calculate the values
Input Aa: Set the clothoid parameter on the starting side. The value at Ab (the end point) is automatically calculated
Input Ab: Set the clothoid parameter on the ending side.
The value at Aa (the clothoid parameter on the starting side) is automatically calculated.

The transition curve on the starting side, the circular section and the transition curve on the ending side is displayed in red, green, and blue.
Red X at the turning point indicates it is invalid. Edit it to remove this X.
If the X remains, reconsider the parameters of the alignment, type of change point and curve. “Auto-Fix” is also available in the right-click to correct the invalid turning point automatically.
When creating more than one road, repeat the above operations.

(2) Entry by spline

1. Click "Make road spline" or right click and go to "Start" - "Start spline road".
2. Click on plan view. Select the number of starting and turning points as well as the end points.
3. Click "Make road spline" button once more or right click and select "Finish Road".

The "Starting Point" is created at the point first clicked and the "End Point" at the point clicked last. The dots in between are "Turning Points". As it is difficult to place these "points" at the correct coordinates, place them at the approximate locations with the correct number of turning points.

*To save the defined spline road in XML format, click "Save" from the right-click menu. The saved file can be loaded by "Load spline road".

(3) Adding / Editing of Turning Points

Right click on the road to add a turning point. Right click on a turning point and go to Edit – Edit Turning point....
3. Generation of Roads

When editing or generating roads, Draft mode is available to prevent erroneous roads and to reduce time.

Draft mode
Draft mode roads will only be partially rendered, which can save loading times and is excellent for previews. Draft mode affects both Plan view and Main view.

- Draw cutting / banking:
  Yes: Any road cutting surfaces are calculated and displayed.
  No: Earth cuts and embankments are omitted for a faster preview.

- Draw rivers:
  Yes: Compute and display rivers.
  No: ignores Rivers.

- Ignore intersections:
  Yes: new intersections cannot be created in Draft mode. Existing intersections are depending on the current view. *In Plan view, “ignored” intersection(s) are represented by a dotted outline and cannot be edited.

- Roads to be drawn:
  Select the roads to be partially rendered. Unrendered roads will not be displayed in the Main navigational view and no traffic can be generated on it.
  In the plan view these are also displayed as a dotted outline but cannot be edited. Roads not selected cannot have ramps.

- Terrain alpha:
  This is terrain transparency. 0% is completely transparent and 100% the opposite. By default, 75%.
Road generation options

Road generation time can be shortened by selecting only the roads that are to be generated.

Select the “Road building option” button to open the menu as shown:

- Generate all roads automatically
- Do not generate any road automatically
- Select the roads to be generated automatically
- Select the roads not to be generated automatically

Generate all roads automatically: All roads are generated when clicking on the Build Roads button.
Do not generate any road automatically: All Roads in the Plan view remain as simple lines and will not be drawn.
Select the roads to be generated automatically: Click and drag a box over the roads to be generated.
Select the roads not to be generated automatically: Click and drag a box to degenerate roads.

▲ Click and drag a box to prevent generation  →  ▲ Selected Roads are no longer generated.
4. Vertical curve road
1. Adding / editing longitudinal turning point
On the Plan view, right-click on the road alignment and go to "Edit - Edit Road". The Vertical curve editor is opened.

Red indicates the section of the selected road with overlap.
Blue indicates relations in height when the road passes over another road.

Curve options
Click Options… to set up parameters of the curve.

- Set this curve as navigable (travel, drive, fly) :
  If unchecked, flight (walk) path becomes unavailable.
- Default speed / altitude :
  The initial speed and altitude of the flight path.
Defining the longitudinal road alignment

1. Click or right click and go to “Add turning point”
2. Click at a point to add turning point. Repeat as necessary.
   ※ Turning points cannot be placed at the position where the cursor changes to .
3. Right click on it and select “Edit turning point” to edit it.

2. Inserting sectional change positions

1. Select or right click and select “Add section change”.
2. Left click at the point of section change.
3. Right click on the line and select “Edit section change”.
4. Select a section to use.

When the sectional change position is displayed in red, it is invalid due to sectional discontinuity.
3. Smoothing Sectional Change

Create smooth section change in cross-sectional inclination or in the road that widens.

Right click on a transition section to open the Editor.

Enter the section "Transition" between the sections that need smoothening.

Check the sectional change by zooming in to the section of the road clicking on the update button.

If it does not work, redefine the section.

Hint:
1. Are the number of the nodes the same?
2. Is there a large gap between the X and Y coordinates?
3. Are the coordinates similar to the road?

Click "Edit by the list" to confirm the section change.

A section is added in this view.
4. Creating bridge / tunnel section

1. Click "Add Bridge"
2. Click on the road alignment.
3. Right click on the bridge (light blue), and select "Edit Bridge".
4. Enter the position and the length of the bridge or the angle if the bridge has a title.

Earth cuts and banks are automatically generated if no bridges or tunnels exists as the road goes through the terrain.

For tunnel section, click "Add Tunnel" button, and perform the same operation as above.
The bridge (light blue), tunnel (brown), banking and cutting sections are generated automatically.

- **Position**: Where a tunnel / bridge starts.
  (entry range : 0.0m to the entire road. As long as no other bridge or tunnel sections exist within 10 m of the specified number).
- **Length**: The length of the tunnel.
- **Start/End earth cover**: The earth cover height at the entrance and exit. By default this is set to 2 meters. Tunnels start and end here.
  ※UC-win/Road will consider the depth of overburden by comparing the relative position of road alignment and cross-section. Tunnels will not be generated if the program determines it impossible. Check the curves drawn if tunnel is not generated.
- **Tunnel light**: The lighting color inside tunnels

**Tunnel lighting**
Comparison by setting colors
Add unmodified terrain section

Add non-edited terrain section. Click the position to add in longitudinal linear area.

- Non-edited terrain section is a section in which the cutting section is not generated automatically. It is used when creating the natural river which has non-artificial banks.
- The initial value of non-linear terrain section is 100m
5. Setting of road attribute

1. Registration of road surface

Go to ribbon Edit - Road – Surfaces.

- Edit road surface texture

Select a texture and a road surface. Unused texture is related to a default road surface.
2. Edit road surface

Edit the properties of the road surface, such as friction, sound and Force Feedback Effect for the tires.

Go to Ribbon Edit - Road - Surfaces and double click on a road surface on the road surface list.

Road conditions, such as dry, wet, snow and icy, can be individually assigned for every road.

*It is also possible via the weather tab in Options - Visual Options.

**Friction tab**

The friction will determine how the tire behaves on the road, which will in turn affect how the car behaves on the road. The curve, called Slip Ratio Curve, determines the friction coefficient of a tire on the road depending on the slip.

To define a precise curve, add any number of points by clicking on the add point button. For each point the slip ratio and the friction coefficient are editable by clicking on the corresponding value. Do not change the origin.
Sound tab

When driving a vehicle, the tires sound represents the sound of the vehicle tires.

Two sounds are played: The rolling sound of wheels turning on the road surface, and the skidding sound when the tire slips. For both sounds there are curves for volume and pitch that depend on the speed and the slip. Each tire has two sound sources, one for the rolling sound and one for the skidding sound. For each sound source the final volume and pitch are a combination of the speed and slip curves for this wheel. Sound of tire skidding is available only for vehicles with advanced physics model.

Curve line:
The graph represents the sound curve of each sound type. There are 4 sets of 2 curves: roll curves based on speed and slip, and skid curve based on speed and slip. The values of each curve can be edited in the Volume and Pitch tabs, and on the graph, the volume curve is shown in red and pitch curve is shown in green. Use the "Test" button to test the rolling sound of tires. It is possible to move the track bars' cursor to play the sound for a specific speed and slip.

Volume: The sound files of the rolling tires and skidding (slipping).
Pitch: The sound pitch is modified depending on the vehicle speed and slip.
Overall volume: the overall volume when testing sounds in the Road Surface Editor.
Sound File: The sound files of the rolling tires and skidding (slipping).
■ Force Feedback Effect tab

It is assigned to different roads based on different conditions. With a steering wheel or game controller, tiny difference in road conditions such as asphalt, gravel, ruts, etc. can be represented and felt through the haptics motors.

*A force feedback capable controller must be plugged in with the proper drivers installed to enable this.

**Effect file name:** Select a file of the force feedback effect. The files listed here are saved under <<User Data Folder>>\Effects by default.

**Minimum speed of the vehicle to play the effect:** Set the minimum speed that the vehicle must travel at in order for the force feedback effect to be activated. There will be no effect when the speed is less than it.

**Maximum speed of the vehicle to play the effect:** Set the upper limit on the speed required to activate the force feedback effect. There will be no effect when the speed is more than it.

**Minimum physical bump size of the road surface to active the effect:**
When driving on a road with a height more than the height set here, the effect of force feedback is generated. If set 0, the height will be ignored.
6. Road bump
Setting of road bump
Go to the ribbon Edit - the ribbon Road - Bump.
* It is necessary to add at least one road to the project.

```
Distance : Define a position to place the bump by inputting how far it is from the starting point in meters.
Offset 1 : Set the offset on right side.
Offset 2 : Set the offset on left side.
Length : Set the distance which has been the bump effect with m unit.

File name : Select the force feedback effect file.
The files listed here is saved in the following folder by default. <<User Data Folder>>¥Effects

Minimum height : Set the minimum height of the bump which the vehicle must run on to activate the effect.
When the value is greater than 0, force feedback effect will be activated when the vehicle runs over a bump with a height having a value greater than this value. When the value is 0, the height will be ignored.
Minimum Speed : Define the minimum speed that must be achieved to activate the effect.
Maximum Speed : Define the maximum speed which is the upper speed limit to activate the effect.
```
7. On/Off ramp

The ramp can be arranged on the road where there is a change in number of lanes.

Double click on a road to open Editing road form. Or right click on a road and go to Edit – Edit road.

E.g.) 3 lanes → 2 lanes, 2 lanes → 3 lanes
Yellow △ indicates the point on the road where the ramp can be created.

The settings for ramps

1. Add a section change.
2. Enter the position at which section change occurs
3. Define the ramp’s horizontal alignment.
4. Define the ramp’s longitudinal alignment.
5. Define the ramp’s section.

Connection patterns of the ramp are as follows:
- Off ramp - On ramp
- Off ramp - Road
- Road - On ramp
Connecting off ramp and on ramp

Move the cursor to the off ramp triangle. When the cursor changes to 🔽, right mouse-click and select "Start Off Ramp" from "Start". Add turning points with the cursor. When the cursor changes to 🔵 at the on ramp, left click to connect the ramp.

Connecting off ramp and another road

Move the mouse to the off ramp triangle. When the cursor changes to 🔽, right mouse-click and select "Start off Ramp" from "Start". Add turning points as appropriate. Right click at the off ramp to be connected and select "Finish Road". An intersection with another road can also be created.
Connecting another road and on ramp

Right click on another road and select "Start" - "Start On Ramp". Add a turning point.

When the cursor changes to at the on ramp, click to connect the ramp.

Examples of ramp connection

Off ramp click  off ramp connection

On ramp connection  Main screen
8. Intersection
1. Create intersection

1. Make a road intersect in Plan view.
2. Align the height for roads in the Vertical Curve Editor.
3. Click "Build Roads" in Plan view.

When creating the intersection fails, red meshes are displayed.

Click 🔄 to display invalid intersections at the center of the screen. To solve this problem, select "Intersection troubleshooter" by right mouse-click on the mesh and select "View Intersection Error".

Reasons of failure in intersection formation
- altitude of intersecting roads is too different
- intersecting road is too close to a parallel.
- unsecured banking
- The center of intersection cannot be found.
- intersection area is overlapped with bridge or tunnel

Editing & Recreating Intersection
- Check to edit the intersection height difference in the Plan View.
- Perform right mouse-click on the intersection and change the intersection area from Edit - "Edit Intersection Size".

Pop-up menu
Right click on an intersection to open the pop-up menu.
Remake Intersection: Regeneration of the intersection
Ignore Intersection: Disregarding the intersection
Edit - Edit Intersection Size: Edit the distance from the center of the intersection.
Edit -Edit Intersection: Editing contours, slope face and texture.

**Intersection Size**

**3-Way Junc**

Draw one road first, and then create a second road that only slightly makes the starting or ending point of the second road cross the alignment of the first road.

Note: Even if it exceeds, if the distance of the exceeded part is shorter than the size of the intersection size, the leg for it is not created.
L-shaped intersections

Slightly intersect the starting and ending points of two different roads.

* The part of intersection will be created only if the distance goes over the size of the intersection.

Roads need to be short enough to prevent unintentional generation of a normal crossroad while making a 3-way intersection or L-shaped intersection. Fix the error by simply click and dragging the start/end points to shorten it.
2. Editing Intersection

Edit the leg lengths, contours, road texture, sidewalk and banking, drive paths on the intersection editor.

(1) Leg length tab

Changing the size

In cross leg selection, the leg length can be changed. The intersection size can be minimized by clicking on "Shrink Legs".

The division of the intersection

The intersection can be sub-divided when the approaching intersection creates another intersection.

The compound intersection can be divided by clicking on "Split Intersection". By this division, the intersection creation will occur necessarily.
Checking "Roundabout" will create a roundabout island in the center. The rotary intersection can be further edited. Edit each radius. The size of radius is in relation to "Island" < "Road" ≤ "Path" ≤ "Ground".

- Island Radius (Range: 1.00 to radius of the road - 1 m)
- Road Radius (Range: radius of the Roundabout Island + 1 m to the radius of the driving route)
- Path Radius (Range: radius of the road to the radius of the land)
- Ground Radius (Range: the radius of the driving route - 100.00 m)
- Lanes ... Input the number of lanes required to create the rotary. (Range: 1 - 12 lanes)

"Flare": Configures smoother shape in large intersection.
When checking on

When checking off

(2) Shape tab

Road, sidewalk, banking and Roundabout Island can be edited.
Right mouse-click on the line and the control point can be added or deleted.

It is possible to edit the control point and arrange the shape.

Handle: The curved part can be changed by dragging and moving.

Anchor point: It can be moved by dragging with the mouse.

Refer to the following sections to find out more information on the detailed editing method of the control point. "Operation" - "Editing" - "Editing intersection" - "Editing intersection" screen - "shape" tab
(3) Road texture tab
Texture setting of the intersection

When the road is not the absolute vertical or horizontal intersection, intersection texture and shape is displayed at an angle. The part indicated by the line is the shape of the actual intersection.

The intersection texture is displayed on the right side of the screen. It shows how many degrees the actual road is turning at an angle, in a clockwise/anticlockwise direction. Texture can be browsed and if OK button is clicked, the texture rotates internally and is pasted in the appropriate position.

Click Texture editor button to open Intersection Texture Editor... By adding the crosswalk etc. to the displayed intersecting regions, the texture of the optional intersection can be edited. Save the image of the new intersection into the appropriate folder and the new image will automatically overwrite existing image. When the texture which was previously saved is not vertical and horizontal, texture rotation can be carried out by dragging the arrowheads on the texture or by direct input of the value. After the texture has been pasted, rotation cannot be carried out. Should this occur, delete the texture, reopen the file once again and adjust the angle to fit the texture that was originally saved.

Example: Texture of the roundabout

Texture cannot be pasted on the roundabout island of the roundabout. Therefore, after exporting to 3DS model, specify the texture on the model.
(4) Sidewalk/ Banking tab

Pushing down each button, correspondence texture is displayed.

Select a texture for each sidewalk and banking, and enter the values for "Scale" "Rotate".

(5) Drive path tab

The drive path and the driving path at the intersection can be edited.
1. Left mouse-click on the arrow of the approaching intersection.
2. All possible drive paths are displayed. Set the weight (driving ratio). Driving is absent when weight is defined to be 0.

The control-point can be added by right clicking on a drive path.

Right-click on the route to assign movement control points (way points) to every single driving path.
This way point enables speed control of traffic flow and event transition in scenarios.

*Refer to the Waypoint section for details.
(6) 3DS model export of the intersection
It is possible to export the intersection created in 3DS model.

(1) Click while the intersection is in editing mode to convert the intersection into 3DS export. The model is saved in \model\intersections#.

(2) Click to open "Intersection Replacement" form. Click on "Load 3DS", and select the model (*.3DS) saved in (1).

(3) Click "OK" button in the Model Editor to register the model to "Intersection Replacement".

(4) Click on the intersection model located in "Intersection Replacement" screen and click "OK" button. The intersection is replaced by the registered intersection model.

Right click on the intersection with 3DS output to open "Replace Intersection with 3DS" screen.
By replacing normal intersection with 3DS model, editing with an external tool is possible. Hence, high-precision intersection can be created.
3. Editing Intersection texture
This is a CAD tool which draws marking on intersections. It is possible to draw vector markings, construct marking library for reuse of vector marking, adjusting color balance of texture, outputting vector marking to CAD file (*.dxf), etc.

Go to the Road Texture tab and click on Texture editor to access the window.

Function explanation of main screen

<table>
<thead>
<tr>
<th>- Ruler</th>
<th>The coordinate value (eastern-north ...west...-south) at the cursor position in a local coordinate system is displayed at the edge of the screen. The horizontal ruler shows the east and west position from the starting point and a vertical ruler shows the south north position from the starting point. The bridge (light blue), tunnel (brown), banking and cutting sections are generated automatically.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toolbar Tool</td>
<td>Automatically Generate Markings.</td>
</tr>
<tr>
<td></td>
<td>Carriageway Texture</td>
</tr>
<tr>
<td></td>
<td>Markings Library</td>
</tr>
<tr>
<td></td>
<td>Clear all markings</td>
</tr>
<tr>
<td>Display</td>
<td>Zoom to selected area</td>
</tr>
<tr>
<td></td>
<td>Pan</td>
</tr>
<tr>
<td></td>
<td>Full size: Put the screen into the state of an initial display.</td>
</tr>
<tr>
<td></td>
<td>Zoom in: Expand the screen based on the center of the display area.</td>
</tr>
<tr>
<td></td>
<td>Zoom out: Reduce the screen based on the center of the display area.</td>
</tr>
<tr>
<td>Drawing</td>
<td>Edit mode: Switch to Edit mode.</td>
</tr>
<tr>
<td></td>
<td>Square: Draw square.</td>
</tr>
<tr>
<td></td>
<td>Circle: Draw circle.</td>
</tr>
<tr>
<td></td>
<td>Bezier polyline: Draw [Bezier polyline].</td>
</tr>
<tr>
<td></td>
<td>Bezier polygon: Draw [Bezier polygon].</td>
</tr>
<tr>
<td></td>
<td>Zebra zone: Draw Zebra zone.</td>
</tr>
<tr>
<td></td>
<td>Stop-line.</td>
</tr>
<tr>
<td></td>
<td>Crosswalk</td>
</tr>
<tr>
<td></td>
<td>Text Markings</td>
</tr>
<tr>
<td>Basic edit</td>
<td>Undo: Return it based on the operation of the previous state.</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>![Redo:]:</td>
<td>Do the operation returned before over again.</td>
</tr>
<tr>
<td>![Add to marking library:]:</td>
<td>the selected marking to the library.</td>
</tr>
<tr>
<td>![Make group:]:</td>
<td>Make two or more selected marking a group.</td>
</tr>
<tr>
<td>![Release the group.]:</td>
<td></td>
</tr>
<tr>
<td>![Cut]:</td>
<td></td>
</tr>
<tr>
<td>![Copy]:</td>
<td></td>
</tr>
<tr>
<td>![Paste]:</td>
<td></td>
</tr>
<tr>
<td>![Delete]:</td>
<td></td>
</tr>
</tbody>
</table>
Property panel

Beziers curve property

Property of basic shape

Set up the properties of basic shape.

Property of traffic marking

Go to "Menu" - "Option" - "Traffic Markings Location Settings".
Layer list
Layer list is displayed under the edit screen right property panel.

Marking

```
Layer list

Markings | Background | Base lines |
---------|------------|-----------|
✓ show markings

Background
```

The roadway, slope, crosswalk, road, geographical features, and street map.
Note: Only when the geographical features check box is checked, a street optional map is displayed.

Base line

```
Layer list

Markings | Background | Base lines |
---------|------------|-----------|
✓ show base lines

Background colour: [ ]
```

The display of the following elements, the intersection legs, the visible outlines, and the running routes is controlled in the reference line.

Edit of marking

Drawing mode

```
[ ]
```

Making shape on the intersection, marking, and the texture can be added in the drawing mode.

Click [ ] or press the escape key to escape the mode.

Drawing method of basic shape

Click the tool button of shape to draw from among a [basic shape], [Beziers curve] and [traffic marking].

Define the property of shape with the property panel. This is possible before it draws while drawing. Move the mouse until clicking the intersection, and becoming a necessary size. Click once to fix shape, and apply shape to the intersection.

Use editing point

Eight red points appear in surroundings of shape when the use shape of the edit point is selected.
A blue point is displayed at the center of shape.
In the red point that is called a resize point, it is possible to change the size, width and height of shape via the points.
A right and left point must change width, and resize these points while keeping the fineness ratio of shape. An upper and lower point changes height. However, do not maintain the fineness ratio of shape in this case.

**Auto-generation function** generates the automatic generation function intersection automatically. "Option" -" Auto-generation settings"

- Define the intersection marking parameter of default by using automatic optional generation.
  - **Stopline**: Set width and the color of the intersection stopline.
  - **Lane edge**: Set the color of the edge of the intersection lane.
  - **Contour line**: Set the color of the visible outline of the intersection.
  - **Gap line**: Set the dashed line. The short dashed line contains two consecutive dashes, and defines the color and the distance of each dash here.
  - **Ignore complex**: Generate a smooth marking automatically at shape with a complex road.
  - **Save as default**: Default to a present setting and preserve it.
  - **Restore default**: Read default values.

**How to export the marking texture**
Go to "File" - "Export Markings to" as RMK or DXF file
9. Road Sign, Markings
Before arranging road signs or markings, create at least one road before.

1. Arrangement of the sign

Go to the ribbon, Edit - Road – Side Objects .

1. Click a sign to use.
2. Click "New"
3. Set each items.
4. Click apply and check the modification in 3D.

Position

Input a road name.
- **Distance**: distance from the road starting position
- **Offset**: offset distance from the roadside
- **Rotate**: rotate angle.
- **Side**: arrangement position at the road. Choose from Right or Left.
  The direction and the offset starting point of the sign will change accordingly.
- **Over Head**: enable overhead signs that spans above a part for the entire of the road.

Sign

- **Height/Width**: size of the sign.
- **Offset**: offset from the pole.
- **Paint Back Of Sign**: background color. When the front and the back colors of the sign are the same, uncheck it.
- **Set black areas transparent**: Black becomes transparent.
- **Display in Orthographic View**: Display in 2D view.
- **Mirror the texture of the backside**: The backside of the sign flips horizontally. Use for unsymmetrical signs.
2. Editing of the sign
Click a sign to open the edit form. It is possible to edit the parameters above.

Distance

A "Distance" tab on the roadside sign editor is added. It is possible to confirm the visibility of the sign.
3. Marking

Go to the ribbon, Edit – Road - Side Objects.

1. Click the marking to use.
2. Click "New".
3. Set each item.

**Position**

Input a road name.

**Distance:** distance from the start of the road

**Offset:** Offset distance from the edge of the road.

**In Lane:** Arrange the marking in a driving lane.

**Marking Not Square:** check the marking is not square.

**Side of the road:** arrangement position on the road. Choose from Right or Left side

**Group**

**Number:** the number of the signs to be arranged.

**Start Position:** the starting position of the first sign.

**Spacing:** The spacing between objects, which are arranged in the intervals.
Basic - Landscape
River

1. Click "Make River" button.
2. Create river in the same way as creating a road.
3. Define the road alignment.
4. Define the section with river shape.
Select a river texture. The shape of the river shore can be defined with a section.
5. Back to the main window. The river has been created.

- Define the movement of the texture in “Edit section “...” Speed”.
- Change the water speed.
- Turn on [Environment Movement] to begin the river movement.
Backdrop
It is possible to add or edit a landscape of mountain, buildings or fences, etc.

1. Select "Make Backdrop" in Plan view.
2. Click on the desired position for the backdrop. It is possible to add multiple points.
3. Double click on the backdrop and select the image.
4. The backdrop is generated in the main screen.

Fill in gaps within backdrops of objects, such as fences and guardrails, black to make them transparent. Backdrops are pasted to fit the terrain.

The movements of backdrop can be activated by [Environment] is switched on.
Lake
The water surface can be visualized from "Edit" - "Plan view" - "Make a Lake". It is also possible to visualize the ripple by the wind by switching on "Environment Movement" after activating wind from "Visual Options" - "Weather" - "Activate Wind".

Enter the water surface with closed spline curve. Water surface is defined by right-clicking on the aqua blue line - Edit - Edit properties of shoreline.

- Smooth edges
- Straight edges

Basic settings
- Name: Name of the lake
- Display mode: Shaded or Texture
  *Shaded maintains high performance/
- Edge type
  Straight: Line, Smooth: Bezier Curve
- Height above sea level: -20000.00 - 20000.00 m
- Opacity: Transparency of the water surface. 0% (Transparency) - 100% (Opaque)

Shading
- Water color: Color used to render the water.
- Quality: Define how many waves are blended together to render the water surface. Higher values will result in nicer water surfaces but slower rendering. If the rendering is too slow, reduce this value.
- Wave length: (Min) – (Max). The range within which the waves that compose the water surface are. Use higher values for large water surface (i.e. the sea) and smaller values for ponds or even fountains.
- Highlight algorithm, Highlight amount: 4 highlight algorithms are provided. The strength of highlighting can be set.
- Reflection: (Min) – (Max). It will be minimum when the camera is looking straight down to the water surface and maximum when the camera vector is tangent to the water surface.
- Direction range: Direction of the waves
- Amplitude factor: Height of the waves. Small values for ponds and fountains and high value for the sea.
- Speed: Speed at which the wave are moving (this is a ratio to the wind speed).
- Load...: Load the shading setting saved before. (file extension: .rps)
- Save...: Export the current shading setting. (File extension: .rps)

When Texture is selected as Display mode, set up the texture settings.
Turn on Environment to activate the water surface movement.

Water surface texture: Choose from "UCwinRoad DATA/Textures/Waves".
Wave texture Size: Size of water surface texture
Still texture: Water surface is affected by "VisualOptions - Weather - WindBlowing"
**Flight Path**

**Plan view definition**

The flight path is defined in Plan view. The path can be used not only for the flight path, but also the walking path for the character.

Draw flight path with 📊 button or right-click menu. Click on a starting point, turning point 1, turning point 2 and end point in order. Then right click and select "Finish Flight path". Click "Finish Flight path Loop" to loop it.

**Add turning point**

Right-click on the plan view, and add turning points from "Add" - "Vertex to Flight path".

**Edit turning point**

Drag a turning point or open its turning point editor by right-clicking and select "Edit" - "Vertex on flight path …". Enter the coordinates on Local, X-Y, TKY, DMS or DD tub and click "OK".

Right click on plan view, and go to "Edit" - "Flight path" to open the Vertical curve editor. A curve indicates the current vertical curve, and ○ mark indicates a turning point.

**Vertical Curve Definition**

Right click on the flight path, and go to Edit – Edit flight path to open Curve Editor.

**Add turning point**

Right-click anywhere and click "Add Vertex".

**Edit turning point**

Edit a curve by dragging ○ marks or right click to select Edit Vertex..

**Lock turning point**

Right-click on a turning point and click "Lock Vertex". It is displayed as ■.

**Delete turning point**

Right-click on a turning point and click "Delete Vertex".

It is possible to edit the altitude of the point selected manually.

Define a flying path on the main screen
1. Go to Edit–Flightpath – Record Flight Path.
2. Press the space key at the point to start a flight path.
3. Move the camera position using the move button of the viewpoint and press the space key on a turning point.
4. Repeat step 2 and 3 as needed.
5. Click on the record button of the flight path to finish.

Click the yellow sphere to change the color to red.
Click the red sphere on the flight path to open the popup editor. The spheres can be moved by dragging the sphere while holding down Ctrl and Alt.

Defining a walking path
Define a walking path on which MD3 character can walk.

On the Vertical curve editor for the flight path, click On Ground to place the entire path on ground.
The thick blue line indicates where an intersecting horizontal curve is.
3D Model (Registration, Editing, Arrangement)
Create and arrange a 3D model (new building) by the following procedure.

3D model creation → Registration to UC-win/Road → Arrangement → Editing

1. The registration of 3D model
Go to the ribbon, Edit - Scene – Library to open Model Panel. It is possible to create, download and load many kinds of models.

Model Editor
Set up the model on Model Editor. On Model panel, click New - “Building Model”.

![Model Editor](image)

![Model Panel](image)
Load more models

Click the menu, Load - 3DS model (*.rm, *.3ds) to load 3D format models (.3ds) or the 3D model files saved in UC-win/Road (*.rm, *.rmc).

When the model is extremely big or small, it cannot be displayed in the viewer. Change the size or scale.

*If a loaded 3DS model has no textures displayed:
  - 3DS model textures are saved as image files separately from the model itself. Check if the texture files are in the same folder as the 3DS model loaded.
  - Check if the export process is indeed properly finished when exporting models from other software.

About UC-win/Road database

It is possible to download materials such as 3D models, textures, Road models (mobile models), road sections and MD3 characters. The database is always added and updated. It requires to be accessed via internet.
2. Create complicated 3D building model

Go to Menu New – Building Model. Click Building Edit on Model Editor.

1. Create a model with the drawing tools.
   - move an individual top
   - move the shape
   - rotate the shape

2. To delete the individual shape, specify it with , right click to select [Delete shape]. And click to delete all.

3. The height and altitude can be changed. Select the shape with .

   For example, in the case of the balcony with 5m height from bottom of the building, input 5 in the Shape height. In this case, the height of the shape of building is assumed 0m.

   Hint: If the bottom of the shape is higher than 0m, input its altitude in the "Shape altitude".

4. Click “Set building texture...". Assign the texture set up its scale. Click Apply to finish.

   In order to apply different textures to each walls and buildings, check “Surface details” before clicking “Convert” button.
6. Click on "Convert" to apply the changes.
7. Set up the other settings as needed.

Generate 3D model using Shapefile building data

1. Go to File "Import" - "Import Shapefile".
2. Click Add Files to specify (a) file(s).
3. Select each object type as File import parameters.
4. Set up the conversion method and settings.
5. Click Import to start import.

Output 3D model as Shapefile

It is possible to export 3D models created as Shapefile.

On Model Editor, click Building edit.
Click, ![folder_icon] and specify the directory and the file name. The building model are saved as Shapefile data.
3. **Model Editor**
Double-click on a model or select Edit from right click menu on Models Panel to open Model Editor.

It is possible to set up it on this form

4. **Texture tab**
Apply the texture
On the Textures tab, set up the textures for each part of the model.

The basic texture (for daytime) and the lighting texture (for night) are supported.

**Black is transparent**: Black textures become transparent.

**Blend / Overwrite**
- Blend: Show with mixing a basic texture and a lighting texture
- Overwrite: Show with overwriting basic texture by the lighting texture

**Only at night**
It makes Blend/Overwrite valid only at night.

**Lighting effect (Bloom)**
Enable the representation of the light spreading.
Examples
Left: Blend, Right: Overwrite

Left: at daytime, Right: at night

The model transparency (the layer)
1. Select a layer to be penetrated.
2. Check “Transparent” and edit the alpha value by %.
Edit and delete texture (node)

Select a texture (node) and press F2 key to change the node name.

| Model | well2 | well1 | past3 | past2 | past1 | metal | glass | base |

Sound source tab

When noise simulation option is effective, 3D model can be set as sound source.

Property

Select the sound source characteristic.

Set sound source characteristic from the ribbon, Analysis - Noise propagation - Sound Property.

5. Light sources tab

It is possible to add lighting effects to 3D models. Go to Visual Options – Display, and check the “Advanced Lighting” checkbox to display the lighting effects. The light effects can be applied to models such as vehicles, streetlights, or buildings. While there is a limit on the amount of light sources that can be displayed simultaneously, there is no limit on how many light sources can be set. The limits can be adjusted on Performance tab in the Visual Setting screen.
A model can have multiple lighting sources, but they must belong to at least one lighting groups.

Click on "Add Light Source Group" to generate a new light group. Select the new group and click "Add Light Source".

The settings for lighting are basically the same as for spotlights, and are represented by radial lines with the current position as the apex (this area is the area illuminated by the selected light source in the 3D view).

**Position:** Set the position of the light source on the model.

**Direction:** Set the yaw or pitch angle of the light source. The High Beam Pitch is used when high beam is activated.

*Check the "High Beam Mode" checkbox under the model preview window to toggle between low beam and high beam mode.

**Light Properties:** Set the color, angle, intensity, and graduation angle of the light.

**Attenuation Factors:** Sets the attenuation factors with respect to distance. The formula is as follow:

\[
\text{Luminance} = \frac{\text{Intensity}}{ \text{Constant} + \text{Linear} \times \text{Distance} + \text{Quadratic} \times \text{Distance}^2}
\]

* Distance : The distance from the light source.

Repeat as needed for any more additional light sources.

**Don’t use these settings for the main vehicle, use advanced headlights:**

Use advanced headlights for the car currently driving. It does not work for static models.

**Active**

Enable the currently selected light source group.

**Only when vehicle lights are turned on**

Lights are only activated during nights or when deliberately switched on.

**Switching between low beam and high beam:**

Low beam / high beam switching function can be assigned to either a game controller or a manual simulator switch. Cars that belong to the traffic flow automatically uses low beam lights during nights or inside tunnels.

Although high beam needs to be manually activated, low beam can be automatically switched on and off during night or daytimes. To enable it automatically, go to File – Application Options - Game Controller Options and check the “Automatically turn headlights On/Off” checkbox. This option is also available in the Visual Options window, under the Display tab.
5. Setting visual options

Go to Home Ribbon - Visual Options to open the Visual Options window. Under Display tab, check "Advanced Lighting".

### Advanced Lighting
Enable advanced lighting effects. Generally, it is recommended to switch it off unless truly needed to increase performance.

### Street Lights
Enable street lights when Advanced Lighting is checked. Check to draw it.

### Models Lights
Enable model lights. Check to draw it assigned on the 3D Model.

### Street Lights Positions
Display positions of street lights with line.

### Headlights Positions
Indicate where headlights are positioned on vehicles.

Check "Advanced Lighting" on the Performance tab in the Visual Options screen.

### Streetlights and light sources max display number:
This sets the maximum number of street lights, or light source set to models that can emit lights at a time.

The currently driving car’s headlights take the highest priority. For street lights or other light sources, as long as it does not exceed the maximum number set here, it will be displayed from the object that is the closest to the user.
Edit light sources on models

It is possible to edit dynamic light sources (e.g. cars) and static light sources (e.g. street lights).

Use advanced headlights:
Use Advanced Lighting when driving.

Add Light Source Group:
Add a new light source group.

Add Light Source:
Add a new light source to an existing group.

Delete:
Delete the currently selected light source or light group.

Copy:
Copy the currently selected light source or group.

Active:
Enable / disable the selected light source group.

Only at night:
The light is only activated only at night

Position: Edit the offset to the light sources. Changing the setting,
Orientation: Set the light source angle.
Yaw: Horizontal direction facing
Pitch: Vertical direction facing
High Beam Pitch: Pitch angle when high beam is activated.

Light Properties
Color: The color of light coming out of the light sources.
Angle: Lighting angle.
Intensity: Lighting brightness intensity.
Gradation angle: Lighting graduation angle.
Attenuation factors: Attenuation factors of the light relative to distance.

Constant, Linear, Quadratic numbers can be assigned. Attenuation is calculated based on the following formula:

Brightness = Intensity / (Constant + Linear * Distance + Quadratic * Distance^2)

*Distance here is defined as from the start of the light source.

Always display with night model
It is possible to turn on the lightning such as lights of vehicle, night texture and bloom in spite of day and night.
Note: This function is applied to a vehicle traveling as a vehicle in the traffic flow only. That's why the night texture or bloom are not activated.
This can be activated from Visual Options setting, Context setting or Scenario Event setting.

1. Visual Options
Check Always display with night model on Display tab.
2. **Context Editor**

Check Always display with night model on Context Editor.

3. **Scenario Editor**

It can be activated using Scenario function.

Check Always display traffic with naitve vehicle model on Others tab of Event Editor. After the event starts, the display switches depending on this setting.
6. Arrange and move 3D Model

Arrangement of the model

1. Select a 3D model on Models Panel.
2. Arrange it at the clicked position.
3. Click the model to open an editor.
4. Changing the setting, it will be reflected in the 3D view.

Move the model with the buttons on General tab. Check the option not to change the height of the model.
How to move the model by dragging

- To move it vertically: Click it, and drag with pressing Alt key
- To move it horizontally: Click it, and drag with pressing Ctrl key.
- To rotate it: Click it, and drag with pressing Shift+Ctrl key.

dH,θ: The relative height and angle

Press [Ctrl] + [Shift] + [Alt] + click on the road to arrange a 3D model as a road side model.

Edit Model Level of Detail

Right click on a model on the Models panel. The Model LOD Editor will open.

I LOD (Level Of Detail) speeds up the drawing of complicated models by displaying a simplified model for a distant model. For example, a building model may have detailed internal information to display nearby, but it does not need to be drawn because they are not visible from a distance.
7. Arrange model and wire as the roadside objects

Go to the ribbon Edit – Road – Side Objects.

1. Select a model to be used.
2. Click “New”
3. Edit the parameters.

Position tab

Select the road name to be set up.

**Distance**: Distance from the road starting position.

**Offset**: Distance from the roadside.

**Rotate**: It is in 0 degrees that the X-direction of the model is confronted with the direction of the running. It is positive to be counterclockwise.
Number: Enter the number of signs.
Start Position: Set the position of the first sign.
Spacing: Set the spacing between road side models.

(Reference)
Arrangement interval $X_n$ between $n$-th and $n+1$-th objects are calculated using the start position $X_0$, fixed value, increment and random number,

$$X_n = start\ position + (n - 1) * fixed + (n - 1) * (n - 2) / 2 * increment + random\ numbers$$

Fixed value of arrangement interval: interval between the first object and the second object
Increment of arrangement interval: increment after the first interval on group arrangement randomness of arrangement interval: a randomness of group arrangement.

(horizontal) Offset: horizontal offset on group arrangement. The edge of road is 0.00m. The inner side is + direction.

Horizontal offset $X_n$ of $n$-th object is calculated as below.

$$X_n = fixed\ value + (n - 1) * increment + random\ numbers$$

(horizontal) Offset fixed: a fixed value of the group.
(horizontal) Offset increment: increment when increasing offset on group arrangement
(horizontal) Offset randomness: randomness when add randomness on group arrangement.
Group Power Lines

It is possible to install a wire among the models which belong to the same group.
Click two poles with Shift key to generate wire between the poles.
8. Zone editing

The zone is an area enclosed by three or more points selected by the mouse as vertex on the terrain. It is a polygon that connects adjacent points with a straight line, and the sides are defined as the boundary of the zone. Within the zone, it is possible to confirm the height, create buildings or forests, and download the features.

Zone creating mode

Move to the zone creating mode.

1. Open a new project

2. On the Edit ribbon, select Create Zone in the Zone Editing group.

3. Zone creating mode starts. The explanations are shown in upper left corner. Click any points on the terrain to select points.
2. Create zone

1. Select more than three points to specify the shape of the zone.
   - About vertices, the previous and the next one are connected by red walls.
   - The first and the last vertex are connected with a blue ball at the position of the mouse cursor by red wall.
3. Once the shape is defined, press the [Enter] key to fix it.
4. Press Enter to start Zone editing mode again. The defined zone area is surrounded by the red wall.
Note: If the boundary line is not displayed, check "Zone" in the "Display" tab - "Visual Options".

5. Press the [ESC] key to finish the zone creating.

3. Edit zone
The zone can be edited.

Zone editing mode
The explanations of zone editing is displayed in upper left corner. Click the created zone to start the zone edit mode.

It is possible to edit the shape of created zone by the following three methods.
1. Add vertex
2. Move vertex
3. Delete vertex
- **Select vertex**

Click a vertex. The color is changed to yellow.

![Select vertex image](image)

**Move viewpoint**

Press [v] key to move the camera to the viewpoint from the sky. At this time the camera mode becomes satellite.

- **(1) Add vertex**

Move the mouse cursor on the red wall to display a blue ball at the cursor position. Click it to add a new vertex.

- **(2) Move vertex**

  (A) With mouse cursor
  
  Select a vertex to move, and drag it. It moves according to the mouse cursor and are fixed at the point where dragging stops.

  (B) With mouse movement
  
  Select a vertex to move, and holding down the left button with pressing [Ctrl] key. It moves according to the mouse movement. Release the left button or Ctrl key to fix the position.

- **(3) Delete vertex**

Select a vertex to delete, and press [Backspace] key. The vertexes on both side of the delete one are connected.

**Finish zone editing**

Click "OK" button in the "Zone Editor" to finish the zone editing. The zone editing mode finishes and backs to the normal main form.

**Cancel zone editing**

Press[ESC] key on the zone editing to cancel the operation and back to the normal main form. All changes are canceled.

**Building creating function (Zone editing)**

It is possible to create an object by specifying its bottom shape by an arbitrary polygon. The bottom shape is created using the zone editing function.
1. Start creating objects
There are two methods to create an object.

A. From the ribbon menu
1. Open a new project.
2. On the Edit ribbon, select Create Building in the Zone Editing group.

3. The function of creating building starts, and the mode is changed to the shape creating mode. Create the bottom shape, and press [Enter] key to change the mode to the option editing mode and open the Building Generator form.

B. From the existing Zone
Create the object using the existing zone.

<table>
<thead>
<tr>
<th>カラム</th>
<th>値</th>
</tr>
</thead>
<tbody>
<tr>
<td>ゾーン名</td>
<td>zone_A1(5553.31, 4590)</td>
</tr>
<tr>
<td>最低標高</td>
<td>550.00 m</td>
</tr>
<tr>
<td>最高標高</td>
<td>550.08 m</td>
</tr>
<tr>
<td>面積</td>
<td>132.15 m²</td>
</tr>
</tbody>
</table>

1. Click a zone to change the mode to zone editing mode.
2. On Zone Editor form, click Create Building button.
2. Creating and editing the object
After shaping the bottom shape, an extruded model is created with a default 10 meters height. The model can be edited and saved afterwards.

How to create a new object

1. Click on the "Create Building" icon.

2. Define a bottom shape. Click on the vertices for the bottom shape of building. Click vertexes not to intersect each other. If the red walls intersect, the model without the top is created.

3. After the bottom shape selected, push the Enter key to close the form, generate the model and open the Building Generator form.

Hint
- On the way of creating a bottom shape, press the [BackSpace] [ESC] key to delete the last vertex.
- On the way of creating a bottom shape, press the [ESC] to cancel all creation and back to the main form.
- On the way of creating a bottom shape, press the [v] key to moves the camera to the viewpoint from the sky. At this time the camera mode becomes satellite.
Edit in the Option Editor screen

The generated model can be edited from the edition form. When starting the option editor mode, the following Building Generator is opened. Easy editing can be done on this form. Click OK after edited to apply the changes directly.

Edit objects after saving

Saved objects can be treated as other 3D models in UC-win/Road. Click on an object in the 3D view to display the Model Tool. It is possible to change the position (horizontality and height) and rotate angle.

The Model Editor form will be displayed when the Edit button in the Model Tool is clicked. More detailed model editing such as model type, slope angle, and scale is available.
Reference
The height of the reference surface is that of the lowest point at the time of the point selection. The reference surface is an open surface.

Direction of extrusion
The surface is vertically extruded by a specified height. The top surface (the opposite side of the reference surface) will be covered after the extraction.

Forest creating function (Zone Editing)
The forest creation function using zone edition function is a function to create forests in a specified area on the UC-win/Road. The forest can be created by putting trees within the area specified using zone edition function.

How to create forests
There are two methods to create forests.

A. From the ribbon menu
1. Start the UC-win/Road and create forests.
2. On the Edit ribbon, select Create Forest in the Zone Editing group.
3. After selecting it, the forest creating function is started and it changes the mode to the area creating mode. Specify an area to create a forest.

B. From the existing zone
1. Click the "Create Forest" button in the "Zone Editor" for the zone already created.

Hint: When a forest is created in the Zone Editor form, the area creating mode is skipped and the "Forest Generator" form is displayed since the area is already specified by the zone.

Specifying the forest area
On the area creating mode, specify the area to create the forest by the same procedure as zone creating.

1. Specify the area to create the forest. Click on the vertices of polygonal area to create the forest in order.
   - Clicking on any places, the points will be the vertexes of the zone and will be displayed in blue balls.
   - A red wall shaped object from the balls is also displayed. This red wall object coincides with the sides of polygon. About vertexes, the previous and the next one are connected by a red wall and the last one is connected with the first one.

2. After specifying the forest creating area, press the [Enter] key. Then, the area creating mode is finished, a forest made in initial value and the "Forest Generator" is displayed.

Hint
   - On the way of creating forest area, press the [ESC] key to cancel all forest creation and backs to the main form.
   - On the way of creating forest area, press the [BackSpace] key to delete the last selected vertex.
   - On the way of creating forest area, press the [v] key to move the camera to the viewpoint the sky. At this time the camera mode becomes satellite.

- Setting tree
Set trees to be put on the displayed Forest Generator form.
After adding the tree, inputting the attributes and clicking "OK" button, trees are placed in the specified area at the rate specified.

Editing after added
Select the forest tree to open the Model Tool form. On this form, the position and angle can be edited individually.

Hint
Clicking "Delete" button on this form, a message whether to delete all trees in the zone or not is displayed.
Download features function (Zone editing)
The download feature function is one of the zone editing function that creating zone on the specified area in the UC-win/Road, in which we can download to arrange building and forest models defined by OpenStreetMap on the internet.

Start Download features
There are 2 ways to import zone information.

A. Download features from ribbon menu
1. Start UC-win/Road and create a project.
2. Click Download features on the ribbon menu to start zone creating mode.
3. Specify the area where to get information. Click on the vertices of polygonal area in order.

B. Download features from zone editor
1. Open the Zone Editor form the existing zone and click "Download features" button.
   * In this case, Area the area creating mode is skipped and the "Model selection" form is displayed since the area is already specified by the zone.

   【How to specify the area】
   1. Click "Download Features" on the ribbon menu to start zone creating mode.
   2. Specify the area where to obtain information. Click on the vertices of polygonal area in order.
   3. Clicking any places, the points will be the vertices of the zone and will be displayed in blue balls.
   4. A red wall shaped object from the balls is also displayed. This red wall object coincides with the sides of polygon. About vertices, the previous and the next one are connected by a red wall and the last one is connected with the first one.
   5. After specifying the download features zone area, press the [Enter] key. Then, the area creating mode is finished.
      • On the way of creating area, press the [ESC] key to cancel all download features zone creation and backs to the main form.
      • On the way of creating area, press the [BackSpace] key to delete the last selected vertex.
      • On the way of creating area, press the [v] key to move the camera to the viewpoint the sky. At this time the camera mode becomes satellite.
Select features

"Select Features" form is displayed after finishing area creating mode or click "Download" button on the "Zone Editor" form. Select the data source and feature type to be downloaded. When both checkboxes are unchecked, it is not possible to click download button.

Edit objects to generate

After downloading the features, the "Select Features" form is displayed, on which there is a features list generated with the initial value. Using this form, it is possible to edit an object to be imported finally.

Add building object to library

A building object created using OSM (OpenStreetMap) import function is not added to a model library just by creating it. To add it to library, select it severally and click OK in the "Model Tool" form.

Select Features Form

After downloading buildings or forest information, a form is opened to select objects used on UC-win/Road. On this form, choose to save generated objects (on a project) or not and edit the names and the parameters. Parameters can be edited in the same way as forest creating and building creating.

Click "Download" button in "Select Features" form to display objects with default settings and the object list.

Each initial value is as follows.

<table>
<thead>
<tr>
<th>[Name]</th>
<th>OSM(Data type)_List number(*1)</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>[Forest]</th>
<th>【Building】</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number: 100</td>
<td>Height: 10(*2)</td>
</tr>
<tr>
<td>Arrangement ratio: 10</td>
<td>Model type: Building</td>
</tr>
<tr>
<td>Min.height: 6</td>
<td>Color: Gray</td>
</tr>
<tr>
<td>Max.height: 10</td>
<td></td>
</tr>
<tr>
<td>Min. rotate angle: 0</td>
<td></td>
</tr>
<tr>
<td>Max. rotate angle: 360</td>
<td></td>
</tr>
</tbody>
</table>

(*1) The name is displayed if it is set on OpenStreetMap
(*2) The height is displayed if it is set on OpenStreetMap. And for the building, if the story of the building is defined, the height = the story * 3.5.
Object list (unfolded)
Open Category to unfold the list.
The selected object on Model list is displayed with highlighted colour.

▼ When selecting a building object  ▼ When selecting a forest object

Save
Save the objects or not by checking or unchecking the checkbox in object list.
When unchecked, the model is hide temporarily on the main screen. And when checked again, it is displayed again.
check/uncheck a category name to select all objects of the category together.

Rename
Click the name column on Object list to rename.

Editing objects
Click the setting button ( icon) on object list to edit the selected object.
But the model is unchecked, it is uneditable.
When clicking a setting icon of forest type object, the same editing form as forest creating function is displayed. When building type object, the same editing form as building creating function is displayed.

Copy editing information
Click the Copy button ( icon) to copy and retain the editing information of the object list. One information per model type can be retained.

Paste editing information
Click the Paste button ( icon) to overwrite the editing information.

Object information
Select a object on object list to display the object information on the right window.
9. Addition of the driver / the pilot

It is possible to set the registered model as a driving vehicle running car on the road, and as a flight object on the flight path.

Right click and go to "Move on" - "Road / Railroad" or "Flight path".

Set the driving/flight route, the initial speed, the start position.

Follow another model
When a driver has already been set, it follows the selected driver.

10. Delete the models

Individual deletion
Click on each model and then the delete button in the window.

Mass Delete
1. Select more than one terrain.(Shift or Ctrl + Click)
2. Right-click, and select "Select Objects On Selected Terrain"
3. Select the types to be deleted from "Delete Selected Objects"
Multi-select deletion
1. Click a model continuously while pushing Ctrl key. It is canceled when clicking once again.
2. Right-click, and select the types to delete from "Delete Selected Objects" of the menu. Cancel the selection by pressing the Space bar.

11. Copy the model
1. Click model continuously while pushing Ctrl key. It is canceled when it is clicked once again.
2. Right click anywhere to open the menu. Select “Duplicate Models” to copy the models. To cancel, press the space bar.
3. When clicking at a desired position, the chosen models are generated into similar orders.
Also, while selecting more than one, the horizontal movement can be done by Ctrl + dragging, vertical movement by Alt + dragging, and rotation can be done by Shift + Ctrl + dragging.

* Currently there is no Undo function.
* The model duplication window may occasionally become hidden behind the main window.
Models Panel
1. Models Panel

Go to Edit – Scene – Library to open Models Panel.

From "Menu" - "New", "Load" or "Download", models and textures can be added.

Clicking “Synchronize” to synchronize with model library.

And it can be shown or hidden by type and can be sorted by name, extension or date.

On the tags manager, the tags in the current project can be searched and viewed.
2. Integrate duplicated models or sections

By the operations such as merging files or copying a model, many duplicated models are generated, which increases the file size. This function enables to integrate duplicated models or sections and to delete unnecessary ones. This function is composed of 3 processes; Similarity calculation process, Object replacement process and Tidy Up process.

1. Similarity calculation process
Detect objects which have similar settings or internal parameters as Similar objects. The parameters to be compared depend on the object type.

2. Object replacement process
Similar objects can be replaced with a target model which has similar settings. This process replaces all places referring to the similar objects with the target object. For example, when a similar model used for a resource of model instance with a target model is replaced, all objects with the similar model are replaced. This process is applied to all settings.

3. Tidy up Process
Tidy up similar models at unused state. This process deletes the similar models from the project. It is the Integration function so that the project is downsized using these processes.

Show Similar Models
Use the flame on the bottom of the model panel to show similar models to a selected model and to replace them with it. Resources which are similar to Target resource are shown as similar resources. And they can be replaced with a target resource, so that the replicated resources can be integrated.
Parametric model
It is possible to generate parametric models. Open the model panel from Home Ribbon - Edit tab - Library.

1. Models panel
Loadable objects
Signs, Stairs, Escalators, Polygonal column (prismatic column and cylinder), Ladder, Wall and Barrier

1. Click menu button and select “New” - "Model".
2. Select “Stairs” here as an example.
3. The selected parametric model is added to the project.
4. Select the stairs model in the resource list to display the basic parameters. Set them as needed.
5. Right-click and select Edit to open the editor. Set parameters as needed.

2. Editor
Example: Editing newly created / loaded stairways.

- **Stairs Origin:** Origin position (0, 0, 0). Bottom or Top.
- **Rise:** Height of rise.
- **Run:** Length of the steps.
- **Steps width:** Width of steps.
- **Flight of stairs:** the number of steps from start to landing.
- **Flight of stairs origin:** Position where it begins to count number of steps. Top or Bottom.
- **Textures:** Select Texture for each part

Note: The folder of stairs textures is located in: \<UC-win\>-\Road User Data Folder>\Textures\Parametric Stairs
3. Arrangement of parametric models
Select a model and click on 3D space to arrange it.

- Edit instance position and shape parameter
Click the arranged parametric model to open the Model Tool form.
*The parameters edited on the Model properties tab are applied to only this model.

Click Edit on Model Tool to open the Editor again.
On Instance tab, check "Apply changes to all instances which share the same model" to apply them to all instances of this model.

4. Instances tab
On the Instance tab, the entire height and the depth of the landing can be set and the stairs can be customized by adding, editing and deleting the wall and handrail.

Wall
Visible: Display or hide the wall.
Height: Height of the wall.
Origin: Origin from right side or left side of the stairs.
Thickness: Thickness of the wall.
Simple wall: Check to generate a parallelepiped wall (from bottom to the top). Uncheck to generate a wall that follows the stairs path.
Wall Begin: Start position of the wall. It can start before the starting point of stairs.
Wall End: End position of the wall. It can end after the end position of stairs.
Color: Color of the wall. It is merged with the texture set in the model tab.
Shininess: Shininess of the wall.

Handrail
Visible: Display or hide the handrail.
Height: Height of the handrail.
Origin: Origin from right or left side of the stairs.
Position: Offset from the origin.
Radius: Radius of the bar of the handrail.
Type:
Normal - The handrail support is added to steps.
Wall - The handrail support is added to a wall.
Handrail Begin: Start position of the handrail. It can start before the starting point of the stairs.
Handrail End: End position of the handrail. It can end after the end point of the stairs.
Color: Color of the handrail.
Shininess: Shininess of the handrail.
FBX Scene
UC-win/Road supports high accuracy FBX file.
It can be imported and edited from Edit - Scene - Library - Model Panel - Menu – Load and Download.

1. Load FBX file

1. Load or download FBX scenes (.fbx).

2. Click "OK" button on the edit form of FBX scene.

3. The new model is registered and added to the model panel.

The following file types are supported: FBX (*.fbx), 3DS (*.3DS), DXF (*.dxf), Collada (*.dae) and Alias OBJ (*.obj).

Import only single side: Draw all polygons only in one side.
Calc normal in SDK: Calculate the normal with SDK.
Merge vector (slow): Vectors are integrated when vertex coordinates and the normal vector overlap (as well as reading before Ver.8). As for this function, reading might be very slow when the file has a lot of numbers of vertices. It is material same as 3D models.
2. Edit FBX file
Edit FBX model on FBX scene editor..

Structure tab
On the Structure tab, expand the structure and click to open Fbx node editor.

Node Options
Check the "Allow transparency" to allow the transparency of the FBX node.
Uncheck the "Both sides polygon" to draw one side which of the normal vectors of each polygon is directed.

Diffuse tab
Node material: Edit the texture attribute.
*It is possible to select two or more items at once.
Texture: All textures that the material contains are listed.
Add new: Add a new texture for the material.
Change image: change the image of current texture.
Wrap-U, Wrap-V: Select wrap types of U and V direction.
Wrap type: REPEAT or CLAMP
Scale-U, Scale-V: Texture scales on U and V direction.
Offset-U, Offset-V: Texture offsets on U and V direction.
Rotation: Rotate texture by degree.

On Diffuse tab, set up daylight textures.
On Emissive tab, set up night textures.
On Transparent tab, set up the transparency processing.
*When a texture with alpha channel is assigned on Diffuse tab, the transparency will be processed.
However, a transparent texture on the transparent tab takes priority.
3. Assign FBX model

The registered scene can be assigned via “Add details” or “Edit Models”.

1. Click “Add details” button.
2. Select “FBX Scene” from the list.
3. Select a model from Model Panel, and put it on the arbitrary position.
Texture Tree

1. Arrange

It is possible to download materials for the texture tree and arrange it.

Go to the ribbon, Edit > Scene > Library > Model Panel > Menu – Load, Download.

Choose a texture from Model Panel.
When the height of the tree is not specified, it is randomly set.

To use the created tree texture, go to C:/UCwinRoad Data X.X /Textures /Trees.

After arranging it, it is possible to change the size from Add > Model Tool.

2. Roadside Tree Editor

It is possible to edit the tree texture in Roadside Tree Editor. Go to the ribbon, Edit > Road > Side Objects – Tree.

1. Choose a tree
2. Click “New”.
3. Edit each parameter.

On the tab in Roadside Tree Editor, set up the following parameters. Road name, Distance (distance from the road starting position), Offset (distance from the roadside), Rotate, Height, dH, and side of the road.
On the Group tab, set up the following parameters. Number, Start Position (the distance from the starting point of the road) and Spacing.

3. Create forest
1. Choose more than one terrain, and click "Create Forest" on the right click menu.

2. Enter the number of trees to be arranged.

3. Forest is generated.

To delete it, hold the control key while selecting the terrain area, then right-click to open a menu and click Delete Selected Trees.
3D Tree

1. Register and Edit

To create a 3D tree newly, open the Models Panel by selecting the ribbon, Edit - Scene - Library, and select the menu. Go to New - 3D tree to open the 3D tree editor.

3D tree is roughly divided to "standard tree" and "palm tree".

"Standard tree" is composed of the tree, trunk, branch, leaves and blossoms.

"Palm Tree" is composed of the tree, trunk and leaves.

"Instance Default"

Check the following options as needed.

Show Leaves
Show Blossoms
Wind Blows Leaves
Wind Blow Branches
Create with the standard tree
"Treetab"

<table>
<thead>
<tr>
<th>Tree</th>
<th>Trunk</th>
<th>Leaves</th>
</tr>
</thead>
</table>
| Type: | ○ Standard tree  
       | ○ Palm tree     |
| Description: | |   |
| Wiggly: | |   |
| Designed age: | |   |

- **Description**: the name of the tree
- **Wiggly**: set the curve of the tree.
- **Designed age**: set the age of the tree.

**Tree shape profile:**
Divide a tree in 10 and set the length of the branch.
The left side is the root, the right side is the tip. Drag the point into upward to stretch the branch, and downward to shorten the branch.

"Trunk tab"

<table>
<thead>
<tr>
<th>Tree</th>
<th>Trunk</th>
<th>Branches</th>
<th>Leaves</th>
<th>Blossoms</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Height:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Texture:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Repeat texture:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Trunk shape profile:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Radius**: set the radius of the trunk
- **Height**: set the height of the trunk
- **Texture**: set the texture of the trunk. The image files are saved in "Tree folder/  
  Bark folder".
- **Repeat texture**: set the number of repetitions of the texture.

**Trunk Shape Profile:**
Divide a trunk in 10 and set the thickness of the trunk.
The left side is the root of the trunk, the right side is the tip. Drag the point into upward to make it thick, and downward to make it thin. The dotted line indicates the radius.

"Branches"

<table>
<thead>
<tr>
<th>Tree</th>
<th>Trunk</th>
<th>Branches</th>
<th>Leaves</th>
<th>Blossoms</th>
</tr>
</thead>
<tbody>
<tr>
<td>Branch angle:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Branch position:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Branch density:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Branch shape profile:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Branch angle**: set the angle of branch growing. The left is the minimum angle and the right is the maximum angle. The angle upward is 0 degree, and downward is 180 degrees.
- **Branch position**: set the position of the branch. The tip of the trunk is 100 %, and the root is 0 %. The left is the lowest, the right is the highest.
- **Branch density**: set the density of branches.

**Branch shape profile:**
Divide a branch in 10 and set the thickness of the branch.
The left side is the root, the right side is the tip. Drag the point into upward to make it thick, and downward to make it thin. The dotted line indicates the radius of the root of the branch.
"Leaves"
Check “Show Leaves” of "Instant Default".

Leaf size: set the size of leaf texture.
Leaf positions: set position of the leaves. The tip of the branch is 100%, and the root is 0%. The left side is the lowest, and the right side is the highest.
Leaf density: set the density of the leaves.
Flatness: set the offset and the direction from the branches.
Color variation: set the color of the leaves.
Texture: Set the texture of the leaves. Images are saved in Tree\Leaves folder.
Leaf shape profile:
Divide a branch in 10 and set the leaf position. The left side is the root of the branch, the right side is the tip. Drag the point into upward to make it far from the branch, and downward to make it close to the branch. The dotted line indicates the position of the branch.

"Blossoms"
Check “Show Blossoms” of "Instant Default"

Blossom size: set the size of the blossom texture.
Blossom density: set the density of blossoms.
Texture: set the blossom texture. The images are saved in "Tree folder/Blossom folder"

2. Load and download
On Models Panel, go to Load - 2D/3D trees or Download - 3D tree (*.tree) to load or download 3D tree.

3. Arrange
Select a model on Models Panel, and click on any points where to arrange it.

The arranged 3D tree can be moved with "Model Tool". It is possible to enable "Show Leaves", "Show Blossoms", "Wind blows leaves", "Wind blows branches" and "tree age".
Flag
Go to Edit - Scene - Library - Model Panel to load or download a flag.

1. Choose a texture of the flag, and arrange it.
2. Click it on main screen to change the size and the position on Model Tool.

- Set the height of the flag and the pole.
- Choose from "No Pole", "Line Pole" and "Solid Pole"
- If "Cloth Flag" is checked, the flag will move more realistically according to wind speeds. Enable movement via the Environment button in the tool bar, or go Options - Environment Movement. Also, change wind speed on Visual Settings.

*By assigning black colors to a part of the flag model, it is possible to have non-rectangular shaped flags
Fire and smoke
Fire and smoke can be visualized – various phenomena such as flame, smoke, steam and sparks from a fire can be visualized.

1. Defining fire and smoke
Define fire and smoke from the ribbon, Edit - Scene - Models Panel, or Edit - Scene - Add Details.

First load or download the fire and the smoke texture in the Models Panel or the Detail Tools windows opened by the ribbon Add Details. Then select the fire or smoke texture using the radio button, and the cursor will be changed to select the texture from the list. Click on a position to put a fire or smoke in the main screen.

Texture of fire and smoke can be downloaded from our database.
* The below textures are listed:
  * Textures saved in << user data folder >> ¥ Textures ¥ Fire

1. Edit fire and smoke
It is possible to edit fire and smoke by selecting the defined fire or smoke.

"Fire / Smoke" Tab
Select "Fire/Smoke" tab in "Model Tool".
* This tab is displayed only for the fire and smoke.

**Life time:**
Enter the minimum and maximum period of time in seconds each particle of fire and smoke has before it expires.
(The range of input: 0.01 - 99999.00s)

**Initial Velocity:**
Enter the minimum and maximum initial speed of the particles of fire and smoke in meter per second going in X, Y, and Z directions.
(The range of input: -20.0 - 20.0 m/s)

**Advanced**
Fire and Smoke Editor opens, in which it is possible to make detail settings for fire / smoke generation and shape
2. Detailed editing of fire and smoke

- Particles
The fire and smoke are expressed by applying texture to numerous particles, and carrying out special processing. Various parameters like the default position of these particles, the initial speed, the life time etc. can be adjusted.

The selection of the texture:
The following list of textures is displayed.
  - Textures saved in << user data folder >> ¥ Textures ¥ Fire

Max number of particles:
Change the number of particles that compose the fire and smoke (maximum number of particles that exist at one time).
(The range of input: 5-5000 pieces)

Particle rate:
Set the particle rate of the fire and smoke.

Color:
Adjust RGB value of the color of the particles. The range of input is 0.00-1.00 with 0.00 being the dimmest and 1.00 being the brightest.

Particle size attenuation:
Correct parameters set by the viewpoint position of the size of the particle. If the distance from the viewpoint position is d, the size of the particles is calculated by the coefficient that is less than the values entered in a, b and c.
  \[ \frac{1}{(a + bd + c*d^2)^{0.5}} \]

Initial particle size:
Change the size of the particle. After the correction in the viewpoint position, the size of the particle actually displayed becomes a size in which the factor is multiplied.
(The range of input: 0.01 - 99999.00 pixel)

Life time:
Input the range of the life time of the particle in second.
The range of input: 0.01 - 99999.00 s

Dispersion shape:
Dragging the point (red circle below), it changes the range of initial velocity and initial position of fire / smoke. By this operation, it changes the height, width, and root width of fire / smoke, then the shapes changes. The figure shown by dot below becomes the rough size of fire / smoke.
Section:
X-Y/Z-Y radio button
Switch the fire and smoke shape editor between Z-Y surface and the X-Y surface.

■ Source

Initial Position Range: Input a generation position of particle, the range of distance from the center and dH, and the height offset from the elevation. (The range of input: 0.00 - 99999.00m)

[Maximum height] Check box:
It is possible to hide the fire or smoke over the height set here. This function is for fire in the room without going through the ceiling and the wall. In the case setting the maximum height for it showing, check this checkbox to input the height in meter unit. (The range: 0.00 - 99999.00m)

[Default] button: Back to shape of default fire and shape of default smoke.

■ Movement parameters

Minimum dispersion velocity: Input the minimum velocity of the particle dispersing in X, Y, and Z direction in meters per second. (The range: -200.00 - 200.00 m/s)

Maximum dispersion rate: Input the maximum velocity of the particle dispersing in X, Y, and Z direction in meters per second. (The range: -200.00 - 200.00 m/s)

Speed factor: Make an adjustment of displaying speed of fire / smoke on 3D view. Dragging the bar to the right will be speed up, and to the left will slow down. Backing to the default speed, click Reset button. The parameters below are for smoke only, therefore they will be ignored in the case of fire.

It is possible to set the following parameters for the smoke.

Source temperature: Set the temperature of heat source. (The range: -273.00 - 999999.00 deg (Celsius))

Cooling factor: Input the cooling factor. (The range: 0.00-99999.00 /m)

Vertical acceleration factor: Input the vertical acceleration factor. (The range: -999999.00 - 999999.00 m*K/m^2)

Temperature change factor: Input the temperature change factor. (The range: 0.00-99999.0 /m)

Air friction ratio: Input the air friction ratio. (The range: 0.000-100.000)

Air temperature at source: Input the air temperature at the source. (The range: -273.00-999999.00 degree (Celsius))

Air temperature decrease rate: Input the air temperature decrease rate. (The range: -9999.00-9999.00 degree/m)

Travel growing factor: Input the travel growing factor in free space. (The range: -999999.000-999999.000 px/m)

Travel growing factor in tunnel: Input the travel growing factor in the tunnel. (The range: -0.000 - 999999.000 px/m)

【Reference】
Petals and leaves falling from the trees and water splashing can be visualized by changing the texture of particles.
4. Setting smoke tunnel
Smoke and fire inside tunnels can be visualized. Smoke tunnel has a rectangular section. If the source of the smoke is located inside a tunnel, the direction to which the smoke travels can be set arbitrarily. By doing so, the smoke is kept inside the tunnel. Furthermore, placing wind sources will allow the simulation of the behavior of smoke particles taking into consideration the wind direction, and then smoke particles traveling in the wind direction.

To set smoke tunnel, go to the ribbon “Edit” - “Smoke Tunnel” – “Edit”.
(1) Click the “Create” button in the “Smoke tunnel editor” window to open “Smoke tunnel creator”.
(2) In the Smoke tunnel creator, select a road in “Select a road” list. Tunnel(s) that exists at a certain interval along this road will be listed under “Select a tunnel” on the right.
(3) Select a tunnel from this list and click “Create from tunnel”.
(4) Newly created settings will appear in the “Tunnel list” in the Smoke tunnel editor.
(5) Click “Add” in the “Wind sources” box. Click a wind source item to set the position as a distance from the tunnel entrance. Wind speed can be set in km/h. Therefore; tunnel entrance is represented as 0m. Set a value bigger than 0km/h to add speed to wind.
(6) Click the "Environment movement" button to place the smoke within the tunnel and move it in the direction been set. Smokes placed inside the smoke tunnel all move in the same direction.

(7) Check “Smoke tunnel” on “Display” tab in “Visual Options” window to show smoke tunnel with blue frames, and displayed the wind sources and wind direction in magenta. “Visual Options” can be opened from the ribbon, Home – Simulation – Virtual Options. Click on smoke to appear a yellow lozenge in the area.

*If the smoke does not move in accordance to its settings, it may be partly buried. Increase its altitude a bit to move the way that it has been set to move.
3D Text

1. Go to Edit - Scene - Library - Model Panel.
   Click [3D text]
2. Enter a text and define settings on 3D Text Model Editor.
3. Click on a position to place it.
4. If Face to camera is checked, the model can be rotated along the view point.
**Video Wall**

It is possible to arrange a registered video wall, which represents an electrical bulletin board. From Menu – Load or Download – Video Wall on Models Panel, select the video wall and click at a point to place it on.

Available video file extension: **Avi, mpeg, mpg, wmv, mp4, mov, flv, f4v, mkv, mts, m2ts**

Click it on the main screen to open Model Tool. The position can be set as the ordinary model. The height can be set from [Edit model]–[Video walls] form.

Turn on "Environment" to play the video.
Changing View Display And Viewpoint Position

It is possible to change a viewpoint, save and edit it.

1. Save Current Position

Right-click anywhere in the main window to show the pop-up menu and select "Save Current Position".

Or from the main menu, ribbon Home – Camera Position - Save camera position.

The current camera position is saved as "Camera Position x". Several camera positions can also be saved. The numeric keys 1 to 0 are automatically allocated to the first ten saved camera positions.

2. Move Camera To

Right-click anywhere in the main window to show the pop-up menu and click Move Camera To to show a list of the saved camera positions.

Or from the main menu, select one of the camera positions in the combo box from Home – Camera Position.

3. Add Camera Position

Go to "Camera" - "Add Camera Position" to save the current camera position. The "View Editor" is opened, where it is possible to set up the details.

- **Name**: Name of the view.
- **Mirror**: View will be horizontally reversed like that in a mirror.
- **Coordinate system**: The criteria used when positioning the camera
  - **Absolute**: According to the coordinates of the world coordinate system
  - **Relative**: Camera follows the object specified as "Reference point".
  - **Reference point**: Select an object that the camera follows. The Main Camera or the Vehicle can be selected. Selecting Vehicle may not cause normal view during not driving.
- **Position**: Input the camera position. If it is set to "Absolute", input the coordinates of the local or world coordinate system.
  
  If it is set to "Relative", input the offsets in the coordinate system of the object to be followed. These values are saved separately.

- **Direction**:

- **Camera Angle**: Specify the camera angle.
  - **Rotation**: Specify the camera angle by yaw and pitch. If the positioning is set to "Absolute", specify the project's coordinate system. If it is set to "Relative", specify on the coordinates of the object set as reference point.
  - **Viewing Point**: Specify the camera angle using the viewing point. The direction to which camera position is connected with the view point becomes the direction of camera. If the positioning is set to "Absolute", input the coordinates in the world coordinate system. If it is set to "Relative", input the offset from the coordinates where camera is positioned at.
  - **Roll**: Specify the roll angle of camera.
Example of mirror

4. Edit Camera Positions
Right click on main screen and go to Edit camera. Or go to Ribbon Home – Camera positions.

5. Output landscape
The screen can be output to printer and file.
- To print: Go to ribbon [File] - [Print]
- To output: Go to ribbon [File] - [Save current image]

Mancellcolor file (.mcs) can be exported at File- Export - Save to MancellColor file…

* The size of saved image is the same as the displayed area on a main screen.
**Saved camera view**
Open a new screen ("Saved camera view" screen) and display the image from the camera position which is registered by "Save camera position". By selecting several views, it is possible to view the image from various camera positions.
**Lighting function**

Check "Advanced Lights" and "lighting object (bloom)".

Light effects use the Bloom function, which can visualize lights, such as traffic lights, warning lights and headlights.

Bloom Function

Click an object model to open the edit form.

Check texture to activate the bloom. The color and the texture of the bloom can be specified.

It is also possible to make the bloom active only at night.

The bloom function on traffic lights, indicators and brake lamps is activated during a traffic simulation.
Advanced lighting – Headlight/Street light –

Check "Advanced lighting" on a drawing option to enable the expressions of headlights and street lights.

- Headlight

The headlights of the vehicle turn on. Generate the traffic flow, get on the vehicle by a click and turn on the headlights.

*The headlight lights turns on by driving.
* Confirm the headlights by moving the aspect to outside the car using a mouse wheel or "Movement" tool.

Headlights Options

Click or go to ribbon Edit - Headlights - Headlight Options.

Display headlight positions:
Display a small white ball on the light sources.

Headlight 1
Color: Color for the headlights.
Left/Right headlight: Enable the left and/or right headlights.
Height: Height of the headlight relatively to the bottom of the car model.
Width: Lateral offset of the headlight relatively to the center of the car model.
Shift: Longitudinal position of the headlight relatively to the front of the car model.
Yaw angle: Left-right direction in the horizontal plan.
Pitch angle: Up-down direction relatively to the horizontal plan.
Roll angle: Rotation in the cross plan.

Light attenuation: Defines light attenuation according to the distance between the lighted point and the light source.
The formula: \( \text{Brightness} = \frac{\text{Source Brightness}}{(\text{Constant} + \text{Linear} \times \text{Distance} + \text{Quadratic} \times \text{Distance}^2)} \)
- Brightness: The final brightness of the lighted point
- Source Brightness: The brightness of the light source
- Distance: The distance between the lighted point and the light source.
- Constant, Linear and Quadratic: the parameters set in the user interface.
The basic shape of headlight is an ellipse centered on a given position. The brightness becomes weaker as the position of the pixel is away from the center and the light source. In addition the top and bottom half of the light can have a different brightness in order to represent it more realistic. The parameter is same on left and right.

**Max brightness**: Brightness of the center.

**Top light divider**: Brightness of the headlight is divided by this value.

**Max/Min vertical angle**: Ax/Min angle where the light is rendered.

**Max/Min horizontal angle**: Ax/Min angle where the light is rendered.

**Ellipse X**: Horizontal coordinate of the center.

**Ellipse Y**: Vertical coordinate of the center.

**Ellipse bright area size**: Size of the brightest area.

**Ellipse attenuation**: Light attenuation according to the distance between the lighted point and the center.

**Width ratio**: Ratio of the vertical and horizontal radiuses.

### Reflections

Set up reflection by each model.

---

**Streetlights**

Click .

With streetlight mode, click on the terrain or the road to arrange streetlights. The streetlight on the list. When arranging it on the terrain, the streetlights with the selected previously.
If the streetlight which was arranged is not appeared, "Street light" might be turned off. Check "Streetlight" From menu-"Option"-"Model display of the scenery".

The street light is set by the followings;
・The color of light
・The strength of light
・The spread angle of light
・The position of the source of light
・The irradiation direction
*There is no absolute limit on the amount of light sources, but only about 50 of them closest to the main camera are rendered.

Turn on positional display of the street light to display the extension of light. And The selected light is displayed in yellow.
Basic - Railway
Definition of horizontal line for Railway
Define the horizontal line, the vertical line and section line for the railway on the UC-win/Road,

1. Simple method
Create horizontal, vertical and section lines by creating road lines. Select cubic parabola and Sign half wave length curve used on railway line for transition curve used on the turning point of horizontal line. Select single line on vertical line.

(1) Definition of horizontal line
In the Plan View, add start point, turning point1, turning point2, and end point. Refer to Definition of Horizontal Curve for detail.
Right click the turning point (IP point) and select Edit - Edit turning point.

On the Type tab of Turning point editor, Select Cubic parabola or Sine half wave length as a Transition Type.

Select the curve shape
- Transition curve - Circle - Transition curve
- Circle
  Input the following parameters.
1) In case of Transition curve - Circle - Transition curve,
Leading transition curve length
Circle radius
Trailing transition curve length
2) In case of Circle
Circle radius
(2) Definition of Vertical curve
In the Vertical Curve Editor, add the starting point, turning point 1, turning point 2, and the terminal point. Then edit the added each turning point to form the curve. First double-click the targeted point or right-click it and select Edit Turning Point...

On the Turning point editor, select the curve type. The difference between the types is only how the curvature of the curve section changes. In general, the Quadric curve is used for the road and the Circle for the railway.

(3) Definition of Section curve
Define the section curve by the same method for the road.

2. Railway Definition in Details
Railway alignments consist of 3 elements: surveyed centerline (railroad), structural centerline (roadway) and track centerline (track). Each has its own horizontal alignment, vertical alignment, and section alignment, which are defined in the exact respective order. Track centerlines can have cants and turnout switches which lead to other tracks. Fundamentally, define track centerline and sections to create a track shape and a road bed. And define roadway (structural) centerlines to create railroad bed, earthworks, tunnels and bridges.

(1) Surveyed centerline (horizontal alignment) definition
The surveyed centerline is the designed line for railroad construction planning, which is also used to indicate the railroad distance in reality. It consists of horizontal and vertical alignments, and it is not displayed in the 3D space

1. Open Plan View from a tool bar button or go to Edit - Plan View.
2. Click the railroad icon and select “make new railroad”. Add a start point (BP), turning point(s) (IP), and endpoint (EP).
3. Right click to finish.

4. Right click on the turning point and go to Edit - Edit turning point on....

5. Specify the coordinates on Position tab (X-Y tab uses world geodetic datum).

6. Define the type of transition curve on Type tab.

7. Define the curvature radius and the length on Parameter tab.

Repeat 4 - 7 steps for BP and EP.

8. Right click on a point along the line and go to Add > Add vertex to... to add a turning point.
9. Click the added turning point to edit it. If it is displayed with a red X, it is too close to an existing transition curve. Edit its position by changing the coordinates or drag it to somewhere else.

**Editing the start point station distance on the surveyed centerline**

1. Right-click on a surveyed centerline alignment and go to Edit > Edit properties of….

2. Specify the Start Point Station Distance. The negative value is also possible.
(2) Surveyed Centerline Definition – Vertical Alignment

1. Double click on a railway centerline or right click on it and go to Edit > Edit Railroad…

2. Change the name at the top left box.

*By default, the line is close to the terrain.

3. Right click on a point to add a vertical turning point and select Add Turning Point, or simply click Add Turning Point in the toolbar and left-click on a point on the curve.

4. Right click on a turning point and go to Edit Turning Point. Define Left slope, Left distance, Station distance, Coordinate Y, VCL and Curve type. Repeat it for a start and end point.
(3)(Roadway) Structural centerline (horizontal) definition

Click the drawn surveyed centerline. The line will turn yellow.

Structural centerline can be drawn in a similar way as the surveyed centerline. There are 2 methods:
- IP Curve: click turning points to draw a line.
- Offset Curve: draw a line at the offset from the surveyed line.

**<IP Curve>**
2. Draw a line and fine-tune by entering coordinates from the right-click menu.

**<Offset Curve>**
1. Draw a line in the same way as IP point and select Offset Curve.
2. Click on positions of start point and end point, then right click to finish.

*Note: In this example, the lines are deliberately drawn with a gap.*
3. Right-click on the starting point of structural centerline

4. Edit parameters on Offset tab.
   - Station Distance: Station distance of the structural line.
   - Offset: Offset distance from the surveyed line. (Positive: to the right)
   - Interval: Interval of plotted points.
   Repeat it for the end point.
   If the offset values of start point and those of end point are different, the line will be interpolated.

5. Turning points can be added by right click menu.

6. Right click the added turning point to edit it.
(4) Structural Centerline (Vertical) Definition
Define in the same way as a surveyed centerline (railroad).

(5) Track Centerline (Horizontal) Definition
Define in the same way as a structural centerline (horizontal).

(6) Cant settings
Cants can be specified from automatic calculation or manual calculation.

**Automatic Cant Calculation**
There are 2 ways.
A. Right click on the track line in Plan View and go to Edit > Edit properties of railroad track <name>.
B. Right click on a desired position on the line and go to Add > Add railroad track cant calculator to track.

A: From Properties window.

- Start Point Station Distance
- Track Guage: Rail guage in mm
- Speed Setting: speed used for cant calculation
- Railroad Type: Conventional lines is canted at the inner rail, whereas Shinkansen is at the railroad track center.
- Track Direction: Both Direction / Prograde / Retrograde.

*In method A, the cant algorithm is $C = \frac{G \cdot V^2}{0.127 \cdot R}$.
C: Cant (mm)
G: Track guage (m)
V: Speed setting (km/h)
R: Curve radius (m)

B: From Add railroad track cant calculator to track <name>.
- Station Distance
- Cant Centre: Conventional Lines use Inner rail, Shinkansen uses Track center.
- Maximum Cant: Maximum allowable cant. If the results are bigger than this, this value will be used instead.

**Manual Cant Setting**

Right click on the track centerline and go to Add > Add manual cant calculator to track <name>.

(7) **Track Centerline (Vertical) Definition**

Define in the same way as Surveyed Centerline.
Simulation
Waypoint (The motion-control point)
The movement of dynamic models can be controlled when passing waypoints on road or flight path.

The following motion settings are possible.
1. Speed change
2. Lane change
3. Viewpoint change (Up and Down, Left and Right)
4. Focus on a model
5. Control dynamic models

1. Add waypoint
Right-click on road or flight path in the plan view and click Add - Add Waypoint to Road. It can be edited by right-clicking "Edit" - Waypoint".

- On road
- On flight path

2. Movement Settings
Right click on a Waypoint and go to "Edit" - "Waypoint", or double click on a Waypoint to open "Waypoint Editor". Click "OK" to activate all waypoints.

Position: distance from the point
Direction: Choose from Both ways, Forwards, Backwards.
Lane: Choose from any, 1 to 12.
Instance (Model): Choose from model or driving object.
Activate: Activate or Deactivate.
Command: Select the action.
Args 1, 2: Input according to the command.
3. List of the command and the args

<table>
<thead>
<tr>
<th>Command Name</th>
<th>Motion</th>
<th>Args 1</th>
<th>Args 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHANGE SPEED</td>
<td>Change speed</td>
<td>Change speed</td>
<td></td>
</tr>
</tbody>
</table>
| CHANGE LANE       | Change lane          | Inside/outside direction, rate * (Range -100% ~ 100%)  
A positive value: to inner lanes  
A negative value: to outer lanes.  
100% always makes vehicles change lanes to the specified direction.  
*Refer to the below example. | Change distance |
| RESET             | Cancel in the operation | Speed after RESET  
*0 maintains the current speed. |                                             |
| SLOWLY TURN HEAD  | Change the viewpoint to right or left by 45 degrees per second | Angle  *Positive value: to the right.  
0 means straight ahead. |                                             |
| SLOWLY TILT HEAD  | Change the viewpoint to up or down by 45 degrees per second | Angle  *Positive value: to the right.  
0 means straight ahead. |                                             |
| TURN HEAD         | Change the viewpoint to right or left | Angle  *Positive value: to the right.  
0 means straight ahead. |                                             |
| TILT HEAD         | Change the viewpoint to up or down | Angle  *Positive value: to the right.  
0 means straight ahead. |                                             |
| VIEW CAMERA       | Change the monitor display contents in the 3D cockpit | Display saved camera views.  
*0 turns off the car monitor.  
*1~998 displays a view corresponded to the number. And blank will be ignored  
*999 switches the monitor to complete 2D perspective. |                                             |
| LOOK AT ME        | Focus on a specific model  
*Flight path only  
*Requires model that has action commands set, or model that has the "control target" checkbox checked. | Arg 1 |                                             |
| Command of the mobile model | Start the movement of dynamic models | Change speed |                                             |
| CHECKPOINT        | Used in scenarios    | Inside/outside direction, rate  
*Range -100% ~ 100%  
A positive value: to inner lanes  
A negative value: to outer lanes.  
100% always makes vehicles change lanes to the specified direction.  
* Refer to the below example... |                                             |
| SET PROFILE       | Set the profile to be used for the log export | Speed after RESET  
*0 maintains the current speed. |                                             |
| LOG START         | Start logging data   | Angle |                                             |
 Movements are automatically reset at the start of roads / spline roads / flight paths
 If view perspective has been changed at the initial stages of a movement command, it will remain unless otherwise changed again. It does not affect the movement direction.
 The following does not affect movement controls:
 - Deleting a model after it has been setup in waypoint window, the command would still remain.
 - Deleting the action commands after setup will cause the command row to read "> - DELETED -<"
 Commands other than "CHANGE SPEED" and "CHANGE LANE" has no effect on the traffic flow.
 CHANGE LANE:
 (e.g.)
 To make 30% of the cars traveling on lane 1 change to lane 2: Select 1 in Lane column and in Args1 enter 30%.
 To make 30% of the cars traveling on lane 2 change to lane 1: Select 2 in Lane column, and in Args1 enter -30%.
 To make all cars in lane 3 never change lanes: Select 3 in Lane column and in Args1 enter 0%.
 Note: Lane changes are limited to adjacent lanes only at a time. It is not possible to skip lane, such as changing lanes from lane 3 directly to lane 1.
 Animated model commands: The model must be animated to have an effect.
 *Starting from Ver. 8 or later, it is possible to add movement control points in the intersection editor.
 Here, traffic flow speed control and scenario event progression can be defined.
 - Open intersection editor, right-click on a drive path and select Add Control Point.
 - Control positions are measured from the starting point of the drive path.
Vehicle Settings

1. Car Performance

Set "Edit models" for the driving car.

Select a vehicle model from the list.

Select a vehicle from Scene - Library - Models panel.

Select a car from the displayed "ModelsPanel" screen and click on "Edit" button.

Setting model type and vehicle details

Set the model type to "Vehicles".

Vehicle editor

Front axle: Applicable to the Z axis: this value determines the vehicle's center of rotation (in meters).

The input range is 0.000m - model width.

Note: This field applies to vehicles, railroads, and cabs only.

Wheelbase: Applicable to the Z axis: this value determines the interval between the front wheel and the rear wheel (in meters).

The input range is 0.00 to (length of road - Front Axle distance).

Track width: Applicable to the X axis: this value determines the interval between the right and left wheels (in meters). The input range is 0.00 to (length of road - Front Axle distance).

Wheel Max Steel Angle and Turning Radius:

The wheel's maximum steer angle and the turning radius are directly linked with the wheelbase and the track width so defining one will determine both of them. The input range for the maximum steer angle is 0.00 - 80 degrees while the range for the turning radius is calculated from the wheelbase and the track width.

Center of Gravity height: Position of the vehicle's center of gravity on the Y and Z axis – effects the suspension of the vehicle.

Range: 0 to 5 meters.

Roll Axis height: Position of the roll axis – effects the suspension of the vehicle. Range: 0 to 3 meters.

Pivot: Applicable to the Z axis: this value determines the position at which the cab and the trailer are connected (in meters). The input range is half the vehicle length to the full vehicle length.

Note: This field is applicable to the truck cab and truck trailer only.
Set the model type to "Cabs".

Truck Cab Editor Tab
Settings for front axle, wheelbase, pivot, and vehicle performance can be adjusted.

Make the model type "trailers".

Truck Trailer Editor Tab
Settings for the wheelbase and pivot can be adjusted.

Vehicle Performance Profiles:
Choose from the following vehicles.
- Small car
- Medium car
- Large car
- SUV
- Sports car
- Scooter
- Motorcycle
- Truck
- Bus
- Cab/Trailer
Setting of Lighting

Vehicle Lighting

In the [Vehicle Editor] – [Car lights] tab, settings for the right and left turning lights as well as hazard lights and brake lights can be applied.

Setting of Wheels

If the model type is set to "Car", "Setting of the car" and "Wheel" tab become effective.

Select the node to operate as a wheel and then select the front wheels, back wheels and chassis.
<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable number</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small Car</td>
<td>based on Toyota Echo 1.3</td>
<td></td>
<td></td>
<td><img src="https://example.com/3D_Cockpit.png" alt="3D Cockpit" /></td>
</tr>
<tr>
<td>NumberOfGears</td>
<td>4</td>
<td></td>
<td>Number of gears</td>
<td></td>
</tr>
<tr>
<td>gearRatio[1]</td>
<td>2.875</td>
<td></td>
<td>Gear 1 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[2]</td>
<td>1.568</td>
<td></td>
<td>Gear 2 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[3]</td>
<td>1</td>
<td></td>
<td>Gear 3 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[4]</td>
<td>0.697</td>
<td></td>
<td>Gear 4 Ratio</td>
<td></td>
</tr>
<tr>
<td>finalDriveRatio</td>
<td>4.052</td>
<td></td>
<td>Final Drive Ratio</td>
<td></td>
</tr>
<tr>
<td>engineTorque[0]</td>
<td>80</td>
<td></td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[1000]</td>
<td>100</td>
<td></td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[4400]</td>
<td>122</td>
<td></td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[6000]</td>
<td>63/(0.105*6)</td>
<td></td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>wheelRadius</td>
<td>0.29</td>
<td></td>
<td>Wheel Radius</td>
<td></td>
</tr>
<tr>
<td>transmissionEfficiency</td>
<td>0.8</td>
<td></td>
<td>Transmission Efficiency</td>
<td></td>
</tr>
<tr>
<td>frictionCoefficient</td>
<td>0.35</td>
<td></td>
<td>Friction Coefficient</td>
<td></td>
</tr>
<tr>
<td>maxBrakeForce</td>
<td>9800</td>
<td></td>
<td>Maximum Brake Force</td>
<td></td>
</tr>
<tr>
<td>minMass</td>
<td>1000</td>
<td></td>
<td>Minimum Weight</td>
<td></td>
</tr>
<tr>
<td>maxMass</td>
<td>1400</td>
<td></td>
<td>Maximum Weight</td>
<td></td>
</tr>
<tr>
<td>turningCircle</td>
<td>9.8</td>
<td></td>
<td>Minimum Turning Circle</td>
<td></td>
</tr>
<tr>
<td>track</td>
<td>1.44</td>
<td></td>
<td>Track</td>
<td></td>
</tr>
<tr>
<td>motorcycle</td>
<td>FALSE</td>
<td></td>
<td>Not motorcycle</td>
<td></td>
</tr>
<tr>
<td>designSpeedFactor</td>
<td>1.1</td>
<td></td>
<td>Designed Speed Factor</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable number</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>MEDIUM CAR</td>
<td>based on Toyota Corolla 1.8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NumberOfGears</td>
<td>4</td>
<td></td>
<td>Number of gears</td>
<td></td>
</tr>
<tr>
<td>Profile</td>
<td>Variable number</td>
<td>Default Value</td>
<td>Comments</td>
<td>3D Cockpit model</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------</td>
<td>---------------</td>
<td>------------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>LARGE CAR</td>
<td></td>
<td></td>
<td></td>
<td>3D Cockpit model</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>NumberOfGears</strong></td>
<td>4</td>
<td>Number of gears</td>
<td><strong>based on Toyota Camry 3.0</strong></td>
</tr>
<tr>
<td></td>
<td><strong>gearRatio[1]</strong></td>
<td>2.81</td>
<td>Gear 1 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>gearRatio[2]</strong></td>
<td>1.549</td>
<td>Gear 2 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>gearRatio[3]</strong></td>
<td>1</td>
<td>Gear 3 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>gearRatio[4]</strong></td>
<td>0.735</td>
<td>Gear 4 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>finalDriveRatio</strong></td>
<td>3.933</td>
<td>Final Drive Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>engineTorque[0]</strong></td>
<td>80</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>engineTorque[1000]</strong></td>
<td>100</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>engineTorque[4400]</strong></td>
<td>284</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>engineTorque[6000]</strong></td>
<td>145/(0.105*5.6)</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>wheelRadius</strong></td>
<td>0.326</td>
<td>Wheel Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>transmissionEfficiency</strong></td>
<td>0.8</td>
<td>Transmission Efficiency</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>frictionCoefficient</strong></td>
<td>0.3</td>
<td>Friction Coefficient</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>maxBrakeForce</strong></td>
<td>18990</td>
<td>Maximum Brake Force</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>minMass</strong></td>
<td>1600</td>
<td>Minimum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>maxMass</strong></td>
<td>2110</td>
<td>Maximum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>turningCircle</strong></td>
<td>11.6</td>
<td>Minimum Turning Circle</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>track</strong></td>
<td>1.53</td>
<td>Track</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>motorcycle</strong></td>
<td>FALSE</td>
<td>Not motorcycle</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>designSpeedFactor</strong></td>
<td>1.1</td>
<td>Designed Speed Factor</td>
<td></td>
</tr>
<tr>
<td>SUV</td>
<td></td>
<td></td>
<td></td>
<td><strong>based on Toyota LandCruiser VX</strong></td>
</tr>
<tr>
<td></td>
<td><strong>NumberOfGears</strong></td>
<td>5</td>
<td>Number of gears</td>
<td></td>
</tr>
<tr>
<td>gearRatio[1]</td>
<td>3.52</td>
<td>Gear 1 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------------</td>
<td>-------</td>
<td>--------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[2]</td>
<td>2.042</td>
<td>Gear 2 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[3]</td>
<td>1.4</td>
<td>Gear 3 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[4]</td>
<td>1</td>
<td>Gear 4 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[5]</td>
<td>0.881</td>
<td>Gear 5 Ratio</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- finalDriveRatio 3.909 Final Drive Ratio
- engineTorque[0] 300 Engine Torque[rpm]
- engineTorque[500] 400 Engine Torque[rpm]
- engineTorque[1400] 430 Engine Torque[rpm]
- engineTorque[3400] 150/(0.105*3.4) Engine Torque[rpm]

- wheelRadius 0.393 Wheel Radius
- transmissionEfficiency 0.8 Transmission Efficiency
- frictionCoefficient 0.35 Friction Coefficient
- maxBrakeForce 19560 Maximum Brake Force
- minMass 2700 Minimum Weight
- maxMass 3260 Maximum Weight
- turningCircle 11.8 Minimum Turning Circle
- track 1.62 Track
- motorcycle FALSE Not motorcycle
- designSpeedFactor 1.1 Designed Speed Factor

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable number</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPORTS CAR</td>
<td>NumberOfGears</td>
<td>6</td>
<td>Number of gears</td>
<td>3D Cockpit model</td>
</tr>
<tr>
<td></td>
<td>gearRatio[1]</td>
<td>3.166</td>
<td>Gear 1 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[2]</td>
<td>1.904</td>
<td>Gear 2 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[3]</td>
<td>1.392</td>
<td>Gear 3 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[4]</td>
<td>1.031</td>
<td>Gear 4 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[5]</td>
<td>0.815</td>
<td>Gear 5 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[6]</td>
<td>0.725</td>
<td>Gear 6 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>finalDriveRatio</td>
<td>4.312</td>
<td>Final Drive Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[0]</td>
<td>120</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[500]</td>
<td>150</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[1400]</td>
<td>171</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[3400]</td>
<td>100/(0.105*6)</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>wheelRadius</td>
<td>0.3</td>
<td>Wheel Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>transmissionEfficiency</td>
<td>0.8</td>
<td>Transmission Efficiency</td>
<td></td>
</tr>
<tr>
<td></td>
<td>frictionCoefficient</td>
<td>0.28</td>
<td>Friction Coefficient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>maxBrakeForce</td>
<td>14880</td>
<td>Maximum Brake Force</td>
<td></td>
</tr>
<tr>
<td></td>
<td>minMass</td>
<td>1100</td>
<td>Minimum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>maxMass</td>
<td>1240</td>
<td>Maximum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>turningCircle</td>
<td>10</td>
<td>Minimum Turning Circle</td>
<td></td>
</tr>
<tr>
<td></td>
<td>track</td>
<td>1.47</td>
<td>Track</td>
<td></td>
</tr>
<tr>
<td></td>
<td>motorcycle</td>
<td>FALSE</td>
<td>Not motorcycle</td>
<td></td>
</tr>
<tr>
<td></td>
<td>designSpeedFactor</td>
<td>1.2</td>
<td>Designed Speed Factor</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable number</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCOOTER</td>
<td>NumberOfGears</td>
<td>5</td>
<td>Number of gears</td>
<td>3D Cockpit model</td>
</tr>
</tbody>
</table>

- gearing is a WAG
<table>
<thead>
<tr>
<th>Variable</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>gearRatio[1]</td>
<td>3.5</td>
<td>Gear 1 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[2]</td>
<td>2.5</td>
<td>Gear 2 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[3]</td>
<td>1.7</td>
<td>Gear 3 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[4]</td>
<td>1.3</td>
<td>Gear 4 Ratio</td>
<td></td>
</tr>
<tr>
<td>gearRatio[5]</td>
<td>1</td>
<td>Gear 5 Ratio</td>
<td></td>
</tr>
<tr>
<td>finalDriveRatio</td>
<td>13</td>
<td>Final Drive Ratio</td>
<td></td>
</tr>
<tr>
<td>engineTorque[0]</td>
<td>2</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[1000]</td>
<td>3</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[4000]</td>
<td>8</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[8000]</td>
<td>10.3</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>engineTorque[9500]</td>
<td>$11/(0.105*9.5)$</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td>wheelRadius</td>
<td>0.25</td>
<td>Wheel Radius</td>
<td></td>
</tr>
<tr>
<td>transmissionEfficiency</td>
<td>0.95</td>
<td>Transmission Efficiency</td>
<td></td>
</tr>
<tr>
<td>frictionCoefficient</td>
<td>0.3</td>
<td>Friction Coefficient</td>
<td></td>
</tr>
<tr>
<td>maxBrakeForce</td>
<td>1540</td>
<td>Maximum Brake Force</td>
<td></td>
</tr>
<tr>
<td>minMass</td>
<td>160</td>
<td>Minimum Weight</td>
<td></td>
</tr>
<tr>
<td>maxMass</td>
<td>220</td>
<td>Maximum Weight</td>
<td></td>
</tr>
<tr>
<td>turningCircle</td>
<td>4</td>
<td>Minimum Turning Circle</td>
<td></td>
</tr>
<tr>
<td>track</td>
<td>0</td>
<td>Track</td>
<td></td>
</tr>
<tr>
<td>motorcycle</td>
<td>TRUE</td>
<td>Motorcycle model</td>
<td></td>
</tr>
<tr>
<td>designSpeedFactor</td>
<td>1.1</td>
<td>Designed Speed Factor</td>
<td></td>
</tr>
</tbody>
</table>

Profile | Variable number | Default Value | Comments |
---------|-----------------|---------------|----------|
MOTORCYCLE | NumberOfGears | 5 | Number of gears |
gearRatio[1] | 2.615 | Gear 1 Ratio |
<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gearRatio[3]</td>
<td>1.35</td>
<td>Gear 3 Ratio</td>
</tr>
<tr>
<td>gearRatio[4]</td>
<td>1.076</td>
<td>Gear 4 Ratio</td>
</tr>
<tr>
<td>gearRatio[5]</td>
<td>1</td>
<td>Gear 5 Ratio</td>
</tr>
<tr>
<td>finalDriveRatio</td>
<td>(2.666 \times 2.466)</td>
<td>Final Drive Ratio</td>
</tr>
<tr>
<td>engineTorque[0]</td>
<td>25</td>
<td>Engine Torque[rpm]</td>
</tr>
<tr>
<td>engineTorque[1000]</td>
<td>30</td>
<td>Engine Torque[rpm]</td>
</tr>
<tr>
<td>engineTorque[3000]</td>
<td>33</td>
<td>Engine Torque[rpm]</td>
</tr>
<tr>
<td>engineTorque[5500]</td>
<td>(22/(0.105^7))</td>
<td>Engine Torque[rpm]</td>
</tr>
<tr>
<td>wheelRadius</td>
<td>0.3</td>
<td>Wheel Radius</td>
</tr>
<tr>
<td>transmissionEfficiency</td>
<td>0.95</td>
<td>Transmission Efficiency</td>
</tr>
<tr>
<td>frictionCoefficient</td>
<td>0.3</td>
<td>Friction Coefficient</td>
</tr>
<tr>
<td>maxBrakeForce</td>
<td>2990</td>
<td>Maximum Brake Force</td>
</tr>
<tr>
<td>minMass</td>
<td>180</td>
<td>Minimum Weight</td>
</tr>
<tr>
<td>maxMass</td>
<td>230</td>
<td>Maximum Weight</td>
</tr>
<tr>
<td>turningCircle</td>
<td>4.4</td>
<td>Minimum Turning Circle</td>
</tr>
<tr>
<td>track</td>
<td>0</td>
<td>Track</td>
</tr>
<tr>
<td>motorcycle</td>
<td>TRUE</td>
<td>Motorcycle model</td>
</tr>
<tr>
<td>designSpeedFactor</td>
<td>1.2</td>
<td>Designed Speed Factor</td>
</tr>
</tbody>
</table>

### Profile

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable number</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRUCK</td>
<td>based on Daihatsu</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NumberOfGears</td>
<td>5</td>
<td>Number of gears</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[1]</td>
<td>5.339</td>
<td>Gear 1 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[2]</td>
<td>2.793</td>
<td>Gear 2 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[3]</td>
<td>1.593</td>
<td>Gear 3 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[4]</td>
<td>1</td>
<td>Gear 4 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearRatio[5]</td>
<td>0.789</td>
<td>Gear 5 Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>finalDriveRatio</td>
<td>5.857</td>
<td>Final Drive Ratio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>engineTorque[0]</td>
<td>150</td>
<td>Engine Torque[rpm]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>engineTorque[1000]</td>
<td>200</td>
<td>Engine Torque[rpm]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>engineTorque[2000]</td>
<td>240</td>
<td>Engine Torque[rpm]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>engineTorque[3400]</td>
<td>(70/(0.105^3.4))</td>
<td>Engine Torque[rpm]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>wheelRadius</td>
<td>0.381</td>
<td>Wheel Radius</td>
<td></td>
<td></td>
</tr>
<tr>
<td>transmissionEfficiency</td>
<td>0.8</td>
<td>Transmission Efficiency</td>
<td></td>
<td></td>
</tr>
<tr>
<td>frictionCoefficient</td>
<td>0.4</td>
<td>Friction Coefficient</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Variable</td>
<td>Value</td>
<td>Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------------------------</td>
<td>--------</td>
<td>------------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>maxBrakeForce</td>
<td>30000</td>
<td>Maximum Brake Force</td>
<td></td>
<td></td>
</tr>
<tr>
<td>minMass</td>
<td>2150</td>
<td>Minimum Weight</td>
<td></td>
<td></td>
</tr>
<tr>
<td>maxMass</td>
<td>6000</td>
<td>Maximum Weight</td>
<td></td>
<td></td>
</tr>
<tr>
<td>turningCircle</td>
<td>19.5</td>
<td>Minimum Turning Circle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>track</td>
<td>1.6</td>
<td>Track</td>
<td></td>
<td></td>
</tr>
<tr>
<td>motorcycle</td>
<td>FALSE</td>
<td>Not a motorcycle model</td>
<td></td>
<td></td>
</tr>
<tr>
<td>designSpeedFactor</td>
<td>1</td>
<td>Designed Speed Factor</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>BUS</td>
<td>Initially the same as the truck...</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>NumberOfGears</td>
<td>5</td>
<td>Number of gears</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[1]</td>
<td>5.339</td>
<td>Gear 1 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[2]</td>
<td>2.793</td>
<td>Gear 2 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[3]</td>
<td>1.593</td>
<td>Gear 3 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[4]</td>
<td>1</td>
<td>Gear 4 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>gearRatio[5]</td>
<td>0.789</td>
<td>Gear 5 Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>finalDriveRatio</td>
<td>5.857</td>
<td>Final Drive Ratio</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[0]</td>
<td>150</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[1000]</td>
<td>200</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[2000]</td>
<td>240</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>engineTorque[3400]</td>
<td>70/(0.105*3.4)</td>
<td>Engine Torque[rpm]</td>
<td></td>
</tr>
<tr>
<td></td>
<td>wheelRadius</td>
<td>0.381</td>
<td>Wheel Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>transmissionEfficiency</td>
<td>0.8</td>
<td>Transmission Efficiency</td>
<td></td>
</tr>
<tr>
<td></td>
<td>frictionCoefficient</td>
<td>0.4</td>
<td>Friction Coefficient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>maxBrakeForce</td>
<td>30000</td>
<td>Maximum Brake Force</td>
<td></td>
</tr>
<tr>
<td></td>
<td>minMass</td>
<td>2150</td>
<td>Minimum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>maxMass</td>
<td>6000</td>
<td>Maximum Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td>turningCircle</td>
<td>19.5</td>
<td>Minimum Turning Circle</td>
<td></td>
</tr>
<tr>
<td></td>
<td>track</td>
<td>1.6</td>
<td>Track</td>
<td></td>
</tr>
<tr>
<td></td>
<td>motorcycle</td>
<td>FALSE</td>
<td>Not a Motorcycle model</td>
<td></td>
</tr>
<tr>
<td></td>
<td>designSpeedFactor</td>
<td>1</td>
<td>Designed Speed Factor</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Profile</th>
<th>Variable</th>
<th>Default Value</th>
<th>Comments</th>
<th>3D Cockpit model</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAB/TRAC TOR</td>
<td>based on Mack</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>NumberOfGears</td>
<td>12</td>
<td>Number of gears</td>
<td></td>
</tr>
</tbody>
</table>
2. 3D Cockpit

Mount a 3-D cockpit in a driving car to present more realistic landscape views from the car during driving simulation.

(1) Applying an interior model

Select a model from the model list.

Select a 3D cockpit from Edit - Scene - Library - Models Panel.

Select an interior model and click on "Edit".

In "Model" - "Data" tab select "Vehicle Interior" for model type.

It is possible to change the visual contents by moving the mouse on the 3-D display screen.
Setting of 3D Cockpit

The set viewpoint locations of the driver and the passenger are shown as points in 3-D space.

Setting of handle

Select the parts of steering handle in "Parts settings", and specify it as "handle". It rotates around the rotation axis.

- **Yaw**: Right and left inclination of axis
- **Pitch**: Up and down inclination of axis
- **X,Y,Z**: Handle position
- **Max angle**: Angle of rotation of the steering handle

*Steering polygon and not steering polygon are required to be saved in different groups.*
Setting of Mirror

When selecting the mirror parts in Interior Editor, the following parameters can also be set:

- **Settable mirror**: Rear View Mirror, Left Wing Mirror, Right Wing Mirror and Special Mirror (Bus, Track etc.)
- **Yaw**: Rotates the mirror on horizontal plane (-180.00 ~ +180.00)
- **Pitch**: Rotates the mirror on vertical plane (-180.00 ~ +180.00)
- **View angle**: Sets the field of vision in the mirror (0.0 ~ 175.0)

**Display Exterior**: Turns on or off the exteriors of a vehicle viewed from mirrors when driving. If not checked, only the surrounding environment is shown.

*Parts such as mirrors or onboard monitors cannot be defined twice.*

*Image on the mirror is used texture coordinate. So if '3DS' is checked off on the Texture tab in Model tab in the 3D Model Editor form, the mirror function is ignored.*
Setting of monitor

Select the part of monitor in "Parts settings", and specify it as "navi".

*The image of "2D aspect" screen and the spectacle preserved beforehand can be displayed in the monitor
*The content of the display can be switched in the pop-up menu displayed by clicking on the monitor of 3D main screen. Also the display can be switched at an arbitrary position on "Edit of the operation control point" screen.

(2) Application to a driving car

Go to Load 3D model from File.

From the displayed "Model List" screen select a driving car model, and click on the "Edit" button.

Select a cockpit model from "Vehicle Editor" - "Interior" tabs
3. Vehicle Movement Model
When the DS plug-in is effective, the dynamics model in the vehicle can be set. Under this tab, vehicle dynamics models can be chosen. Dynamics model and will affect some detailed vehicle driving characteristics. This has no effect on the cars that are not currently under direct user control. The vehicle engine braking mounted and vehicle mechanics settings for the engine are updated.

As a result, the simulation of "creep", where the vehicle moves without the gas pedal being pressed, is also possible.

**Use UC-win/Road’s car dynamics**: This button enables to use the vehicle dynamics of standard vehicle. When this option is checked, motion platform cannot be used.

**Use CarSim’s car dynamics**: This button enables to use the vehicle dynamics of CarSim. To use dynamics of CarSim, protect of HASP key needs to be added separately.

**CarSim Setting File**: Specify the setting file to use. This file must be created in CarSim. The parameter of the vehicle type and the other information that needs for the initialization of simulation are included in the file.

4. Add custom reference point
A custom reference point enables to adjust view point of mirrors different in size. Click Add custom reference point from Model – Reference point tab and add a reference point of default position and direction. After adding it, it is possible to edit the name, the position and direction.

Items that can be set:
- Number of gears and ratio of each gear
- Torque curve of engine
- Weight range track pitch of vehicle
- Power of brake
- Radius minimum rotation radius of wheel
- Coefficient of vehicle air resistance
- Transmission parameter
Model – Reference point tab

Use this tab to add and edit a reference point or a viewpoint to adjust a position of mirror.

Viewpoint adjustment of side mirror point

This function adjusts images on a side mirror or a room mirror seen from a cockpit. A mirror of 3D cockpit reflects the physical position relation. If the body does not match with the cockpit, the mirror of cockpit may reflect the mirror of body.

In such a case, by adjusting the viewpoint of images, unnecessary parts are not reflected in the mirror, so that the expression is more natural.

Mirror settings

Mirror settings are defined from Vehicle settings – 3D cockpit – Mirror settings. If the cockpit has some mirrors, settings can be defined for each mirror.
Traffic Flow Settings

From the main menu, go to: “Edit” Ribbon - “Traffic” – “Generator”

1. Setting traffic flow and generation

Traffic Generator List

It is possible to generate the traffic flow automatically by editing the following information:

1. (Time) Rate
2. Initial Speed
3. Profile (The mixing rate according to the type of the car)

For intersections, it is possible to express the traffic flow on the intersection by adjusting driving paths within the intersection editor.

Click on “Profile…” in the Traffic Generator List to open the Traffic Profile List Editor. Then clicking on “New” or “Edit” in the List Editor, this Traffic Profile Editor will open.

2. Vehicle Group Setting

Vehicle groups and traffic controls can also be set. Select menu [Edit]-[Vehicle Groups] for Vehicle groups, Performance profiles and Sound profile to show the Vehicle Group Editor. Click the [New] button to create a new group.
Global parameter
The parameters of the vehicle are set here.

Dynamics Model
Define global parameters for Dynamic model, Vehicle and Traffic parameters.
Advanced Dynamics: Check when using the advanced vehicle dynamics model including tire slip. If unchecked, the old model will be used.
Suspension dynamics: Use the suspensions if ticked.

Cross fade parameter:
Driving input switch crossfade duration: Sets the transition time when swapping between automatic driving mode and automatic driving mode. The default value is 1 second.
Model switch crossfade duration: Sets the transition time to smoothly change the acceleration and suspension conditions of the vehicle.
This transition was implemented to stabilize the vehicle when changing its dynamic model.

Vehicle
Type of vehicle: Select a car, two-wheeled vehicle or Motorcycle. (Two wheeled vehicle leans to one side, and the vehicle doesn't).
Drive layout: Select a drive layout: Front-wheel drive layout, Rear-wheel drive layout or All-wheel drive layout.

Traffic parameters
The following value can be set as traffic parameter.
Estimated acceleration: Acceleration amount to be used by driver
Design speed factor: Traffic speed factor

Dynamics Parameters
This tab sets the properties of vehicular dynamics:

Weight(s)
This sets the weight ranges of vehicles. It is useful when simulating situations when vehicles carry different amount of weights.

Inertia
Sets the following inertia parameters: Yaw inertia, Pitch inertia, Roll inertia

Braking System
Max. Brake force: The maximum force car brakes can apply to wheels.
Brake lamp pedal threshold: Sets pedal sensitivity to trigger brake lamps.
Brake lamp deceleration threshold: Sets deceleration speed to trigger brake lamps.
Brake lamp switch time window size: Sets time window to switch smoother brake lamps among exact speed, acceleration and keep distance with simulation car of Scenario.
Anti-lock Braking System: Enables ABS features on brakes. Low / High value: ABS will force a maximum slip within the range defined here.

Engine
The amount of engine torque can be set here.

What's Time window.
UC-win/Road checks speed change, and decides to light on/off a brake lamp within the time window. The time window is used instead of simulations for every flame. The value of Brake lamp deceleration threshold is for the time window, not for every simulation. If 0 is set, it behaves in the same way as the earlier UC-win/Road version.

Engine torque
Torque curve of engine (Torque for PRM) can be defined.
- Engine torque curve (green) represents the torque vs RPM when the acceleration pedal is fully pressed
- Engine resistive torque curve (red) represents the torque vs RPM when the acceleration pedal is fully released.

The actual torque of the engine will be between these two curbs, depending on the position of the pedal.
Click the “Add new point” button to define the correct curve line. This point will be ordered in the ascending order.
Resistive torque curve is automatically generated, but it's definable as well as the torque curve. Click Multiply to multiply all point of the curve and the defined multiplier.

**Engine Inertia**
It is inertia of the engine, used by the tire physics model.

**Transmission**
Transmission and vehicle layout can be set here.

---

**General**

**Transmission mode**
Available only when the tire dynamics are enabled.
Choose between an Automatic transmission (AT), a sequential transmission (SMT) and a fully manual transmission (MT) with control of the clutch.

**Gear box**
The ratio of gear can be defined. Click "Add new gear" button to add the new gear ratio.

Final drive ratio: RPM ratio between the output of the gear box and the wheels.
Dynamics

It is inertia and efficiency coefficient for the transmission.
Clutch Maximum Capacity (used by the semi-automatic and manual transmissions).

- Torque converter

Torque converter has two main parameters: Inverse Capacity and Torque Ratio. Both are edited in a similar fashion:
- Click on any data of interest and simply adjust with the arrows or enter a value directly. Changes are reflected directly on the graph for reference.
- To add new points, click on Add New Point button. Click on the red “X” to delete points.
- The Multiply button will multiply the values set in the corresponding panel by the coefficient define next to the multiply button.

- Upshift / Downshift Schedule

In automatic transmission, gears are shifted based on a certain point along a parameter curve.
The curve indicates the conditions that induce gear shifts, while each curve represents a different gear.
This is edited in a similar fashion as was seen in the torque converter and engine torque.

■ Suspension Dynamics
This tab is displayed only when "Suspension mechanics" is activated on "Global parameter".

- Roll/stiffness
Select spring rate or roll stiffness of the suspension and enter the value of front and rear.

- Spring rate
Set maximum and minimum value of suspension spring rate for front and rear.

- Damping coefficient
Set the damping coefficient of suspension for front and rear.

■ Wheels
This tab is displayed only when "Advanced Dynamics" is activated on "Global parameter".

- Geometry

- Wheel radius: Radius of the wheel
- Grip Factor: Multiplies the slip factor on road surface used for calculation of force generated by tire. Various types of simulation of tire are possible.
- Spin inertia: Inertia moment of wheels
- Rolling resistance factor: Multiplies the rolling resistance factor of the road surface to calculate the force generated by the tire. It allows the simulation of different tire types.
Dashboard
Sets the dashboard items that are displayed when driving:

Dashboard items: Displays all the dashboard elements specified here when driving.

  [New...]: Adds the item selected by the user. The following are currently available:
  Speedometer, digital speedometer, tachometer, current gear, front car speed detector, and trip meter.

  [Test]: Preview all defined dashboard elements.

Name: The name of the item currently selected.

Position
Position: Sets where the meter will be displayed in the main interface.

  Margin X: Fine tunes the position at a pixel level along the horizontal direction. Adding values will put the meter further right, while decreasing the value shifts it left.

  Margin Y: Fine tunes the position along the vertical direction. Increase value for up, and decrease for down.

Speedometer (if a speedometer is currently selected from the list)
Use the unit in the project settings: specifies whether the speedometer will use the unit specified in the project. Unchecking this will allow user to change the unit displayed in the speedometers, independent of the project setting.

Gauge value (only available for needle pointer-based analog meters)
Minimum value: This is the minimum value on the speedometer that will be displayed.

Maximum value: This is the maximum value on the speedometer that will be displayed.
Pointer: The checkbox sets whether or not to show the needle. The color can also be altered.

**Main Dial** (only available for needle pointer-based analog meters)
- **Radius:** This sets the selected meter’s radius in pixel units.
- **Contour:** This sets whether a contour will be displayed and color of the meter. By default it is white.
- **Background:** This sets a transparent color to the meter background.

**Graduations** (only available for needle pointer-based analog meters)
- **Long graduations gap:** Adjusts the graduation gap between 2 long scales.
- **Short graduations gap:** Adjusts the number of graduated scales between long scales.
- **Color:** Changes the color of the scale.
- **Active color:** When this check box is checked, the graduation up to the current speed is drawn with the set color.
- **Font:** Changes the font and font size of the meter.

**Arc** (only available for needle pointer-based analog meters)
- **Start angle:** Sets the angle where the meter graduation starts. 0 is the very bottom of the circle.
- **End angle:** Sets the angle where the meter graduation ends. Can have a value greater than 360.
- **Color:** Sets the color of an arc-shaped strip along the meter.
- **Start value:** Sets the value where the arc starts along the meter.
- *Multiple colors can be added. Simply click on the “+” button and assign colors and values.

**Font** (Digital meters)
- **Font:** Similar to analog dashboards, this sets the font and font size of the information displayed.

**Trip meter** (Only visible when the selected dashboard item has a trip meter)
- **Display unit:** Check to use the length unit set in the project. Otherwise, select the unit (km or miles).

**Copy dashboard settings to all vehicle performance profiles**
- Check this box to also apply the new dashboard changes to all other vehicle performance profiles.
3. Create Sound Profiles

■ Vehicle Sound Profile Editor
Click "New" after selecting any sound profile in the sound profile, the Vehicle Sound Profile Editor will open.

Sound
Engine sound set based on the engine RPM.

Volume
Set the engine volume.

Inside the car (lowpass filter)
Check this item to enable the lowpass filter when testing sounds in the Vehicle Sound Profile Editor. A low-pass filter is used to remove high frequency content from a signal. When it is enabled, the sounds are rendered as if the listener is inside the car. When the lowpass filter is disabled, the sounds are rendered as if the listener is outside the car.

Make sure that the sound card drivers are updated. If there are problems, specify a correct sound device in the Audio Settings window. Click "Edit..." button to update the lowpass filter settings for this sound profile:

Gain: This parameter controls the volume at all frequencies.
Gain HF: This parameter control the volume at high frequencies.
In Tunnel (reverb)
Check this item to enable the reverb when testing sounds in the Vehicle Sound Profile Editor. When the reverb is enabled, the sounds are rendered as if in a tunnel.

Make sure that the sound card drivers are updated. If there are problems, specify a correct sound device in the Audio Settings window.

■ Engine Sound
Engine sound parameter can be set.

Sound: Define the engine sounds files.

Engine Sound: The graph represents the engine sounds curves. Use "Add new volume point" to add points. Move the track bar cursor to play the engine sound at a specified RPM.

Test: Use the "Test" button to test the sound profile engine sound.

PRM: Define PRM.

Volume: Set volume.

■ Wind
Wind parameter can be set.

Sound
Define the wind friction sounds files. Click the "Add new sound..." button to add a new sound. The corresponding vehicle speed must be specified for each sound. Use the third columns buttons to test each sound.

Volume
Define the sound volume curve of the wind friction sounds.
To define a sound volume curve, add any number of points by clicking on the "Add new volume button". The points are organized in the increasing speed order.

Wind Sound
The graph represents the wind friction sounds curves. Use the "Test" button to test the sound profile wind friction sound. Move the track bar cursor to play the wind friction sound at a specified speed.

Pitch at Minimum Speed, Pitch at Maximum Speed
This wind friction sound pitch is modified depending on the vehicle speed.
Start Engine: This defines the engine sound when the engine of the vehicle started.

Stop Engine: This defines the engine sound when the engine of the vehicle stopped.

Surrounding Cars Sound File: This defines the engine sound of the vehicles controlled by the computer.

Horn: This defines the horn sound of the vehicle.

Turn Signals On/Off: This defines the turn signals sound when the vehicle's blinkers or emergency lights are used.

Backup beeper: This defines the warning sound when the vehicle is going backward.
4. Traffic sink/source at arbitrary point

It is possible to put the position of sink or source the traffic vehicles except starting or end point.

This is done in Plan View form opened from the ribbon “Edit” - “Scene” - “Plan View”

Right click on the road. Select [Add]-[Add Traffic Sink/Source to Road Settings can be applied for traffic moving “Forwards” and “backwards” separately.

Right click on the yellow node to edit it. Select [Edit]-[Edit Forward (Backward) Traffic Sink/Source]

- To Sink Traffic

At [Sink], check vehicle group to make sink in Enabled and set its rate by percentage.

Example

0: No sinking at the point
100: All vehicles are sunk

- To Generate Traffic

1) At [Source] check Enabled and add phases by [Add] button
2) Set time, rate and profile of each phase.

**Traffic Profile and Vehicle Group**

The traffic profile sets the percentage of traveling vehicles generated by the traffic flow. In the vehicle group, the driving vehicles generated by the traffic profile are grouped.

Set vehicle rates to be generated by traffic profiles. Vehicle groups and [traffic profiles] can be set for existing vehicles.

“Sink” governs the setting for cars that already exist by editing the Vehicle Group, and “Source” generates new vehicles by basing on the assigned traffic profiles.

- The default is applied when vehicles are not yet grouped.
5. Settings Intersection Drive Paths and Vehicle Counts
At the entry points for each intersection, it is possible to set the rate at which vehicles will travel in different lanes.
In the Plan view, right click on an intersection and select [Edit] - [Edit Intersection]. In the Intersection Editor, go to the [Drive Paths] tab.

1) Click green arrow at each entry point.
2) Check vehicle groups and click [+] to open the route list
3) Set a vehicle weight (rate) for each route
   * If weight is 0, no vehicles drive on the route.

6. Vehicle queue lengths
It is possible to set vehicle queue lengths where traffic begins.
For this setting, in the Plan view, right click on an intersection and select [Edit]-[Edit intersection] and [Queue length] in the Intersection Editor.

1. Click green arrow at each entry point.
2. Set number of vehicle to put there in each path respectively (0 to 50)
3. Select traffic profile for the path.
The setting of “Driving Paths” and “Queue lengths” can be also performed in “Edit intersection” appeared by right clicking after selection the intersection on main screen.

7. Setting the number of vehicles at off ramps
At off ramp position, it is possible to set the number of vehicles at each lane. This setting can be done in the "Vertical Curve Editor" opened in the "Plan View", right-click on a rod, select "Edit" – "Edit road".

1. Open the Transition Change Editor in the Vertical Curve Editor
2. Set rate of vehicle driving to off ramp
   - This value is for the vehicle on lane 1 just before the transition point.

8. Traffic Flow
Click the Start Traffic button.
Vehicles travel according to the traffic and signal settings, the driving paths in intersections, and weights applied to each direction.
It is possible to confirm the situation after a fixed amount of time by going to "Home" – "Simulation" Tab – "Run Traffic".

It enables to shorten the waiting time for generating traffic flow. Click Close during calculation to generate the traffic flow at that point.

Traffic settings etc. can be changed during the simulation. Go to the ribbon, "Edit" – "Traffic" - "Generator"

When the traffic flow has stopped due to congestion, cars that are causing the congestion can be excluded by the method described below. The car can be excluded for Ctrl+Alt+"D" + click.

9. Vehicle Movement Settings

1. Select the vehicle by which they want to run to register of the 3D model.
2. Set the driving condition.
   - Input the direction and other attributes such as repetition in the Driving Instructions window opened right-click the target vehicle on the Models Panel, select "Move on" - "Road/Railroad".
3. Click Start Traffic button and watch the traffic flow.

*The moving vehicles here are not influenced by signal controls.

*For the second vehicle in a train situation, the "Follow another model" function can be applied.

10. Traffic Snapshots (Saving & loading of the current traffic status)

This feature allows the saving of the current traffic for later uses, where it can be resumed from under the exact same condition. This is useful when a certain traffic condition such as accidents that need to be examined repeatedly for research purposes. When a traffic snapshot file is loaded, traffic will continue the same way it was at the time of saving.

**Saving**

1. Go to the ribbon, "Home" – "Simulation" - "Save Traffic Status".
2. By clicking on Save Traffic Status, the current traffic flow condition will be saved. The file format has an extension of ".trs".

**Restoring**
1. If there is still existing traffic flow, it must be turned off first.
2. Go to the ribbon, “Home” – “Simulation” - “Restore Traffic Status”
3. Select and load the previously saved .trs file.
4. Once loaded, the traffic flow will be restarted from the condition restored.

*If traffic already exists on at the time of loading, the condition written in the trs file will be integrated into the current status instead of a full restoration.
*Multiple conditions cannot be loaded simultaneously.
Traffic Light Control Settings

1. Detailed settings for traffic lights

1. Open the Model Editor of the traffic light, and expand the model under the “Texture” tab. It is possible to verify the lights and each object.

2. Open "Traffic Light Editor" tab to display the layer of the model and select a blue light. Set it as "Main Green Light" from the drop-down menu on the right. Repeat it for yellow and red. Set "Right Green Arrow" for the right turn signal. Also, it is possible to test the settings from "Model" "Test" radio button. The signal setting can be adjusted according to the country locations from "Options" – "Application Options".

Hint:
All light colors can also be adjusted on "Magic" property. If the stoplight is difficult to see, it is possible to use Main Light Magic for better visual clarity. For example, the light color can be identified from remote position by changing the color of road surface mark in accordance with the light.
2. Intersection Traffic Light Control

1. Click Arrange Model to open the Model Panel. Select a traffic light and place it near an intersection.

2. Open the edit form from the Plane view, and set stopping positions on the "Stopping point" tab.
   ① Click → on the intersection admission-port.
   ② Click a driving route.
   ③ Uncheck "Use the default". Drag or enter a value. Repeat for each route.

3. Select a signal and an access road on "Traffic control" tab.
   ① Check "control with a signal" on "standard setting".
   ② Click a green line on the intersection admission port.
   ③ Click "Edit list" button and select the traffic light to control the road direction.
   It is displayed in the following color.
   • Choosing: Green
   • Unregistered: Light blue
   • Registered: Red
   • Registered as the other traffic approaching point: Blue
4. To control an intersection without traffic lights, choose a control type of the selected approach point. It is possible to control the priority settings. Check “Override” to overwrite it.

Hint: By checking Control with traffic lights, the signal phase is enabled.

5. On the “Signal Phase” tab, it is possible to change the signal on each road direction by each travel pattern

① Set traffic lights for each phase. Click the colors and the arrows of the traffic lights to change them. Uncheck "Lights Control Traffic" to set up traffic lights and driving paths separately.

② Enter the lighting-up time for each signal on "Signal Time". (Check "Default" or enter optional time)

The settings can be confirmed on the phase list.

<table>
<thead>
<tr>
<th>Phase</th>
<th>Green (secs)</th>
<th>Yellow (secs)</th>
<th>Red (secs)</th>
<th>Total (secs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase 1</td>
<td>10.0</td>
<td>2.0</td>
<td>1.0</td>
<td>13</td>
</tr>
<tr>
<td>Phase 2</td>
<td>3.0</td>
<td>2.0</td>
<td>3.0</td>
<td>8</td>
</tr>
<tr>
<td>Phase 3</td>
<td>10.0</td>
<td>2.0</td>
<td>1.0</td>
<td>13</td>
</tr>
<tr>
<td>Phase 4</td>
<td>3.0</td>
<td>2.0</td>
<td>3.0</td>
<td>8</td>
</tr>
<tr>
<td>Cycle</td>
<td>26.0</td>
<td>8.0</td>
<td>8.0</td>
<td>42</td>
</tr>
</tbody>
</table>

3. Traffic control simulation

Return to the main screen and generate a traffic flow controlled by the signal.
Traffic Connector

By adding Traffic connection point, it is possible to move momentarily from one road to another when driving on the road. In Plan view, right click on a road and chose "Add Traffic Connection Point". Set 2 points: an entrance and an exit.

Go to "Edit" - "Edit Traffic Connection" to edit the connection. And it is possible to add or delete it.

Edit the entrance point and the exit point on this form. The directions and the positions of all connection point are listed.
After settings, the traffic connector is enabled. The traffic moves from point 1 to point 2 instantaneously.

- It is possible to drive on various routes by using a traffic connector even if it is not a manual drive.
- It is possible to divide a long route into several parts so that the manual driving can be done continuously.
*After reaching the edge of the road, the driving can be restarted.

- The connection from the edge of the road to another edge is also possible.
Road Obstructions

Hold Shift+Ctrl+Alt and click on a road to create an obstruction.

Click the collapsed area or go to “Edit - Edit Obstructions”, to open Road Obstruction Editor.

Set up the position, length, Lead In / Out, margins, speed limit, right and left lane as well as the display method.

Examples of road obstructions

A road collapse as a result of a falling bridge girder.  
A road collapse using a highway-regulation model.
Movable Model Settings

It is possible to apply movement settings to models.

Open the Model Editor.
On Movement tab, set the grouping and the model parts to work. Then add actions.

On the Command tab, register the procedure of operations to the command list. And On the Key tab, it is possible to allocate hot keys.
Driving Simulation

*Note

When using controller like steering wheel, axel, brake etc., connect it to PC before activating UC-win/Road.

Go to File - [Application Options] - [Game Controller Options] after activating it.

Operate the joystick, steering wheel and button, and confirm each axis in Driver /Walkers setting to Pilot mapping.

On the Drive Responsiveness Settings opened by clicking "Default" button on the Drive / Walker Settings, input the numerical value in a field of each reactive property such as steering and brake to set the play for each axis. 1.0 is for no play and the bigger the value is the bigger the play becomes.


1. Driving Simulation

Click "Drive" button to open the setting form.

If the "Off road allowed" is checked, it is possible to start driving and drive out of the road.

Select a driving direction from the driving method.

And Set up the following parameters
- Driving lane
- Start driving position
- Limit value of max speed and initial speed
- Lock Speed
- Start with stopped engine
- Start with parking brake ON

Select a car from the dropdown list.

*Model and trailer can be selected separately

To drive on offroad
Edit Offroad to add and edit the start position of offroad. Specify the position and the direction. Then the start position will be added to the list of the driving setting form, and it can be selected during driving. It is also possible to drive on terrain, road, intersection, cutting road, banking road, etc.

2. Traveling Simulation

Click “Travel on road”. After enabling the settings of the driving direction, lane, initial position, initial speed, view point height, it is possible to start travelling on the road. To keep the driving speed stable, check to Lock Speed. Depending on the specifications, it may take a while before the simulation begins. It is possible to make use of the ramp to enter another road. Check Ignore waypoints to ignore waypoints while driving.

Operation keys

While travelling, the travelling speed can be adjusted by pressing "↑" or "↓" on the key board. In addition, the driving lane can be changed by pressing "←" or "→".

Pause, Resume
Start/Restart

[↑] key: 5km/hr acceleration
[↓] key: 5km/hr deceleration
[←] key: Move to the lane on the left
[→] key: Move to the lane on the right
[Ctrl]+[↑] or [↓] key: Look up or look down
[Ctrl]+[←] or [Ctrl]+[→]: Look left and right.
[PageUp] key: Increase viewpoint height
[PageDown] key: Decrease viewpoint height

The number keys: Change the viewpoint by every 45°
Holding down [Ctrl] + [Alt] key and clicking on another vehicle: Move the viewpoint to the vehicle. Holding [Alt] key and clicking on the road: "Road Information" screen will open.
It is possible to change the visual settings with "Visual Options". Refer to "38. Visual Options Tool".

■ Navigation Function

- By going back while rotating a mouse wheel, it is possible to trace a vehicle.
- The target object is enclosed in a green frame. Camera can rotate and satellite around it.
- When dragging the mouse while keeping right-clicking, a white ball appears. And it is possible to rotate around it.

■ Auto Driving/ Manual Driving

1. Click "Traffic Movement"
2. Click a vehicle to ride. The view is from the passenger’s seat.

The direction of the viewpoint can be changed using the number keys.
[1]: Lower left
[2]: Down
[3]: Lower right
[4]: Left
[5]: Front
[6]: Right
[7]: Upper left
[8]: Up
[9]: Upper right

By pressing [Enter] key while sitting on the passenger seat, the view will switch to the view from the driver seat. It is possible to drive with a steering wheel, accelerator and brake.

The interior of the vehicle will change depending on the vehicle. The car, bus, truck and the motorcycle are available.

■ Multi-screen display

It is possible to display driving and pedestrian simulation with multi-screen. It contains of right, left, front and rear view.
The views can be displayed or hided from the ribbon, Views - Left, Right and Rear.
Pedestrian Simulation

Click "Pedestrian" button and open "Pedestrian Settings".

It is possible to simulate walking on terrain, carriage-way, and the sidewalk. It can be moved forwards and backwards using ↑ or ↓ and right or left using ← or →.

Change direction by dragging on the mouse.

Increase the speed by pressing "Shift". Press [J] key and click on the field to jump there. Collision is valid when moving slower than 40 km/h.

Height of obstacles that avatar can jump over can be changed from "Home" Ribbon – "Navigation" – "Navigation options".

*Walking can also be simulated by assigning a flight path for the Pedestrian.

- The MD3 character model displayed as avatar on "Navigation option" screen can be selected.

- It is possible to walk by the operation only with the mouse. The pace of walking is changed when the mouse is moved back and forth while clicking, and it rotates right and left when moving it right and left. The direction of the gaze is set forward of the direction where it walks. The distance with avatar can be adjusted by using the mouse wheel. The setting of the operation can be changed on an optional form of the navigation.

- The game controller is also available. For instance, when the steering wheel is used, the direction where it walks can be controlled by turning the steering wheel right and left. The pace of walking can be adjusted by stepping on the gas pedal.

- When the walking mode is begun, the camera is moved to the height of person's aspect even if whatever it is under the camera. (Therefore, it is not possible to walk on in the air and the geographical features which are not seen.)

Walk and run

The pedestrian can walk on roads, a ground or any models.

It is possible to change the height or the walking speed of the pedestrian in the Navigation Options. The camera can be moved by keyboard and rotated by mouse. The 3D mouse enables to do both, then the mouse wheel is disabled.

It is also possible to run. The running speed changes according to the walking speed and the Boost factor defined in the Navigation Options Form. Over 40km/h the pedestrian can only run over the ground.

Collisions

The pedestrian can notice the obstacles and bump into whatever prevents from going further. He will bump into things higher than him or things too high to be climbed. If a slope is too high, he cannot.

It is possible to bump into obstacle on the side or behind the pedestrian.

The pedestrian can go inside a model which has an entrance way.

If there is an obstacle higher than ground level but lower than the pedestrian, he will bump into it.

Moreover, if the width of the way is lower than 80cm it will not be possible to go further.

(Over 40km/h, the collision check is not available).

Climb and fall

The pedestrian can walk over anything lower than 40% of his own height; therefore what can be climbed depends on the pedestrian's height. Over 40%, the pedestrian will bump into it. If he is on something high, it is possible to fall up to 120% of his own height. Over 120%, the pedestrian will not go further.
Crouch
Crouching enables to go under obstacles that are too high to be climbed and too low normally walk into. When the pedestrian is crouched the height of the camera is Crouched height defined in the Options.
As long as he is under the obstacle, if the pedestrian can bump his head, he will not stand, even if the keyboard key to stand is pressed. When the pedestrian is crouched, changing the height with the keyboard will change the crouched height. To change the height, it will stop changing before the real maximum if there an obstacle.

Jump
If there is a narrow hole in the way of the pedestrian (smaller in distance than the height of the pedestrian), it will be possible to go over it and reach the other side of the hole without falling.
The jump function of the Free mode is also available in the Walk mode. Therefore, the option is available in the popup menu and it is possible to jump to the point clicked with the mouse.
The camera will be moved on the exact position and then be adjusted to allow walking at the new position.
Flight Simulation

It is possible to do “Free Flying” and flight simulation “Fly About”.

Create the flight path on the plan view.

Click "Make Flight Path" or select it on the right-click menu. The settings are the same as the road alignment.

From main screen, go to the ribbon, Edit – Flight path – Make Flight Path or Edit Flight Path.

The driving and walking paths are also possible.

Select a flight path and click on the blue arrow to add it into the flight path sequence. It is possible to select some flight paths and fly along them continuously. Specify the starting and finishing position, flight speed, height offset.

e.g.) For walking path, input 1.5m as the height.

Roll means the inclination during the flight.

The flight speed can be changed at a rate of 5km/hr using [↑] key or [↓] key.

The number keys will allow change of 360 degrees viewpoint for every 45 degrees.
Simulation using Visual Options Tool

Display

There are many variations of visual expressions available. When Visual Options is clicked, a menu pops up and each item on the list can be used during road simulation. The expression can change even while driving.

For the details of settings and parameters, refer to Help – Visual Options.
■ Texture
Set up the method of displaying the texture of terrain, the road, the tree, and the road traffic sign is set in this tab.
Sets ant aliasing, anisotropic filtering and optimize transparency for trees, backdrops, road makings, cockpits.

■ Terrain
Set up the transparent terrain and the altitude/slope.

■ Sky
Set up the sky rendering mode, texture, color, orientation and the number of stars.

■ Clouds
Set up the texture and size.

■ Fog
Set up the color, density, type and starting or end points.

■ Weather
Set up the road condition, type, storm, rain sound, and wind.

■ Water reflection/Water splashing
Set up the road surface reflections and rainwater splashes.

■ Windshield Rainwater
Set up the texture and the noise.

■ Wiper Editor
Set up the position, angle, shape, distance, speed, and the sound.

The lowest noise intensity  The highest noise intensity
The lowest rain intensity

The highest rain intensity

■ Wind
Set up the particles and the sound.

■ Thunder
Set up the direction, height, radius, interval and sound.

■ Temperature
Set up the color indicating the temperature.

■ Time, Lighting
Set up the GMT, daylight, date, and the intensity of Eye light.

■ Shadows
Set up the shadow distance and texture size.

■ Performance
Set up the performance. Objects far away are displayed simply so that the drawing performance will be improved.

■ Miscellaneous
Set up the miscellaneous settings
Various light source settings

- Example: lighting up in the viewpoint direction
- Example: spot light when driving at night

- Example: light based on time settings.
- Example: expression of the sun

- Example: vehicle headlights
Expression of Weather

Example: Rain

Example: Snow

Example: Road surface reflection

Example: Water splashes
**Scenario settings**

Scenario is a series of events from the start to end of the simulation. It consists of two or more events generated according to conditions. By performing simulation based on the scenario which is created to reflect events that may occur in real life, and conditions to test, the intention of VR creation can be attained more effectively.

1. **Scenario function**

**The foundations of an event and a scenario**

An event and a scenario become the following structure.

The event is generated when the vehicle arrives at a certain point. Moreover, message and sound can be generated as a result of the event.

- **Scenario**
  Scenario is the flow from the start to the end of the simulation. It contains two or more events within the simulation.

- **Event**
  In the flow of a scenario, an event means on action that is started and ended by the timing condition and the route condition of the driving vehicles etc. After one event is completed, it progresses to the next event. Triggers can be defined by waypoints and events. Note that since waypoints are placed on roads and contain certain vehicle or model movement parameters defined inside, they are only triggered when directly driving.

A scenario is set up by combining two or more events. Set the condition, action, result, etc to be generated. One scenario is created by adding as many events as needed.
2. Driving according to the scenario
   (1) Initial setting
   Connect hardware, such as steering wheel, to the computer before starting UC-win/Road.
   * Connect the steering wheel and the pedals before starting UC-win/Road.

   (2) Adjustment of Device
   After starting UC-win/Road, configure the game controller in Game Controller Options. The display on the left-hand side, “Game Controller Actions”, will react when the program detects the movement of joysticks, steering wheels, or the buttons, which and the corresponding [axis] and buttons react to the movements on the Game Controller Options window.

   ![Game Controller Options](image)
   Go to File-[Application option] and open [Game Controller option]. Operate joystick steering wheel, and buttons and make sure the axis and buttons on the window respond correctly.

   Flight mapping and driver mapping can be used to set up axis and buttons.

   The play in each axis is set up for the numerical value of "Responsiveness" in the Drive Responsiveness Settings opened by the "Details..." button.

   It becomes nothing in 1.0 of the play, and the Play becomes larger when the value is large.

   (3) Driving by Scenario
   Choose scenario from list in the “scenario” in the ribbon, Home - Scenario on the main screen.

   Push the button of start scenario and operation will start.

   - Start Scenario - End scenario

   - When the "scenario" ribbon is not displayed, "scenario" can be validated in the License Manager opened by File – License Manager in the License Manager opened by File - License Manager
3. Edit Scenario
(1) Edit Scenario

Go to menu-[Edit]-[Edit scenario].

Use the scenario editor to edit scenario. The scenarios are edited by using the edit and add buttons.

Double-click "Scenario manager" - "Edit" to access this window and edit scenario.

Add: Add a new event.
Edit: Edit the event.
Insert: Insert a new event on the selected event.
Copy: Copy the selected event.
Delete: Delete the selected event.
Scenario name: Input the scenario name. The existing scenario names cannot be used.
Event: A scenario consists of series of events; create more than one event.
(2) Transition

The first event that exists in the table begins automatically when the scenario is begun.
It is necessary to set "Exit condition" to link the event with the scenario to trigger other events and scenarios.
Create a new event with "Add" button in the Event list, set the "Number of exit", and select destination event in Exit1, Exit2,…
Then selecting the event set in Exit1, Exit2,…, set exit conditions with the Add button in the "Exit condition" List.
Set each transition 1, transition 2, and the destination of each result of the event.
*It is possible to proceed without destination.

To add the exit to the event
Select the number of transitions to be added in exit number row (Exit Number) or input it.

To make the event that makes other events a target
Create a new event with "Add" button in the Event list, set the "Number of exit", and select destination event in Exit1, Exit2,…
Then selecting the event set in Exit1, Exit2,…, set exit conditions with the Add button in the "Exit condition" List.

To edit the exit
The selected exit exists in the exit row, which displays the exit condition.

(3) Exit condition

Define the condition that the event occurs.
It is possible to specify it from the time condition, the speed condition, the checkpoint street condition, and the crash condition.
Two or more condition settings are also possible. The event is not begun if there is no condition.
The target event or the scenario begins when all requirements are met.
Moreover, when all requirements are met, the behavior of the event and the scenario can be decided.

Add: Add the new condition.
Delete: Delete the selected condition.

Time condition

<table>
<thead>
<tr>
<th>Condition</th>
<th>Target model</th>
<th>Argument 1</th>
<th>Argument 2</th>
<th>Argument 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temporal</td>
<td></td>
<td>&gt;</td>
<td>10 s</td>
<td></td>
</tr>
</tbody>
</table>

The time condition becomes effective before and after the specified time period.
To define the time condition
Click Add in the exit condition. Select Temporal in the condition row.
Choose whether the conditions are less than the specified time period; larger specified time period is necessary in Argument 1.
In Argument 2, input the appropriate time period.

Speed condition

<table>
<thead>
<tr>
<th>Condition</th>
<th>Target model</th>
<th>Argument 1</th>
<th>Argument 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Speed</td>
<td>Simulation vehicle</td>
<td>&gt;</td>
<td>50 km/h</td>
</tr>
</tbody>
</table>

The speed condition becomes effective when it is more than the specified speed condition.

To define the speed condition
Click "Add" in the exit condition area.
In the condition row, choose "Speed".
In the target model row, select the target model from "Select a Target Model form".
Choose whether the conditions are less than the specified time period; larger specified time period is necessary in Argument 1.
In Argument 2, input the appropriate time period.

Conditions for Checkpoint operation at the Waypoints

<table>
<thead>
<tr>
<th>Condition</th>
<th>Target model</th>
<th>Argument 1</th>
<th>Argument 2</th>
<th>Argument 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Checkpoint waypoint</td>
<td>No checkpoint</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

This condition is effective only when Checkpoint operation at the Waypoints is enabled.
Note: Only the drive simulation of this condition is effective.

To add checkpoint operation control point
Go to the ribbon, Edit – Edit plan.
Right-click on the road, and select Add – Add waypoint to the road #.
In "Waypoint Editor", add the checkpoint operation at way point.

To define the condition for checkpoint operation at waypoints
Click "Add" in the transition condition area.
In the condition row, select "Checkpoint waypoint".
In Argument 1, select "Checkpoint".

Collision condition

<table>
<thead>
<tr>
<th>Condition</th>
<th>Target model</th>
<th>Argument 1</th>
<th>Argument 2</th>
<th>Argument 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collision</td>
<td>Simulation vehicle</td>
<td>0 model</td>
<td>True when collide</td>
<td></td>
</tr>
</tbody>
</table>

When the target model collides with another model, this condition becomes effective.

To define the collision condition
1. Click "Add" in the exit condition area.
2. In the condition row, select "Collision".
3. In the target model row, click an ellipse button and select a first target model.
4. In Argument 1, click an ellipse button and open Model Selector window. Select one or some target models and click the edit button. Collisions can be handled in 2D or 3D. In 2D, the height is ignored, so the collision is calculated as a plane.
5. In Argument 2, choose True when collide or True when do not collide

True when collide means the condition is met when a collision occurs. True when do not collide means the condition is met when a collision does not occur. As default, True when collide is selected.

### Road condition

<table>
<thead>
<tr>
<th>Condition</th>
<th>Target Model</th>
<th>Argument 1</th>
<th>Argument 2</th>
<th>Argument 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Road</td>
<td>Simulation Vehicle</td>
<td>Highway</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

When driving the target model on the selected road, this condition becomes effective.

#### Define the road condition

Click "Add" in the exit condition area.

In the condition row, select the road.

In the target model row, select the target model from "Select a Target Model form".

In Argument 1, select the road.

In Argument 2, choose one of the following:

- On the Road: The condition is effective when travelling on the selected road.
- Off the Road: The condition is effective when not travelling on the selected road.

#### Direct distance condition

This condition is effective when the distance between 2 target models is more or less than the specified distance.

#### To define the direct distance condition

Click "Add" in the exit condition area.

In the condition row, choose "Direct distance".

In the target model row, click an ellipse button and select a first target model.

In Argument 1, choose whether the condition is more or less than the specified distance.

In Argument 2, select the distance.

In Argument 3, click an ellipse button and select a second target model.
Travel distance to condition

This condition is effective when a distance to the second target models is more or less than the specified distance.

Travel distance means the road distance from a position of the first target model to the second target model. Select a movable model as a first model such as simulation vehicle or event moving model.

Travel distance is calculated as bellows:

- 2 target models on the same road: Travel distance is a distance between 2 models.
- 2 target models on the different roads or intersections: When the second model is driving in front of the first model at an angle of less than 15, the travel distance is based on the travel direction of the first model.

Any other cases: Travel distance is a distance between 2 models.

To define the travel distance condition

Click "Add" in the exit condition area.
In the condition row, choose "Travel distance to".
In the target model row, click an ellipse button and select a first target model.
In Argument 1, choose whether the condition is less or lager the specified distance.
In Argument 2, select the distance.
Note: it is possible to input a negative value. It means the second model is driving behind the first model.
In Argument 3, click an ellipse button and select a second target model.

Time to reach condition

This condition is effective when a first model reaches a second model sooner or later than a specified time.
Note: Select a movable model as a first model such as simulation vehicle or event moving model.

To define the arrival time condition

1. Click "Add" in the exit condition area.
2. In the condition row, choose "Time to reach".
3. In the target model row, click an ellipse button and select a first target model.
4. In Argument 1, choose whether the condition is less or lager the specified time period.
5. In Argument 2, input the time.
6. In Argument 3, click an ellipse button and select a second target model.

Key event condition

Press or release a specified key of a keyboard to enable this condition.

To define the key event condition
1. Click "Add" in the exit condition area.
2. In the condition row, choose "Key event".
3. In the Argument 1, click an ellipse button and select a key.
4. In Argument 2, select the key event type: "On Key Up" means it is effective when releasing a specified key. "On Key Down" means it is effective when pressing a specified key.

Note: the specified key may be used for the other function on UC-win/Road.

Controller button event condition

This condition is effective when pressing or releasing a specified button of a joystick, game controller or steering.

To define the controller button event condition
1. Click "Add" in the exit condition area.
2. In the condition row, choose "Controller button event".
3. In the Argument 1, click an ellipse button and select a button.
4. In the Argument 2, select a type of button event: "On Button Up" means it is effective when releasing a specified button. "On Button Down" means it is effective when pressing a specified button.

Car driver input condition

This condition is effective when a control value of steering, throttle or brake is more or less than a specified value.

To define the car driver condition
1. Click "Add" in the exit condition area.
2. In the condition row, choose "Car driver input".
3. In the target model row, click an ellipse button and select a target model.
   Note: Select a movable model as a target model such as simulation vehicle or event model(vehicle model).
4. In Argument 1, select a driver’s type:
   • Steering
   • Throttle
   • Brake
5. In Argument 2, choose whether the condition is less or larger than the specified value.
6. In Argument 3, input the value.
   - Steering: Input range is [-100% ~ +100%].
   - Throttle and brake: Input range is [0 ~ +100%].

**Model click condition**

<table>
<thead>
<tr>
<th>条件</th>
<th>ターゲットモデル</th>
<th>項目 1</th>
<th>項目 2</th>
<th>項目 3</th>
<th>項目 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>モデルクリック</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

This condition is effective by clicking a specified model.

**To define the model click condition**
1. Click "Add" in the exit condition area.
2. In the condition row, choose "Model click".
3. In Argument 1, click an ellipse button and select a model for model click.

**Driving Control Mode condition**

<table>
<thead>
<tr>
<th>条件</th>
<th>ターゲットモデル</th>
<th>項目 1</th>
<th>項目 2</th>
<th>項目 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>実際の運転状態量</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ハンドル</td>
<td></td>
<td>&gt;</td>
<td>0%</td>
<td></td>
</tr>
</tbody>
</table>

This condition becomes effective by changing the driving mode for the driving simulation vehicle.

**To define the Driving Control Mode condition**
1. Click "Add" in the exit condition area.
2. In the condition row, select Driving Control Mode.
3. Set a target model as a simulation vehicle.
   - Hint: This condition is effective only for the driving simulation vehicle.
4. In Argument 1, choose a mode after the transition from the followings.
   - Manual mode
   - Force automatic mode
   - Adaptive Cruise Control
   - Passenger mode

**User Variable**

This condition becomes effective by comparing the user variable.

**To define a User Variable condition**

In the Exit Condition area, click Add.
In the Condition column, select User Variable.
In the Argument 1, set the index for the target user variable.
In the Argument 2, select the compare condition. It is compared with the value set in Argument 3.
   - = (equal)
   - <= (not greater than)
   - > (more than)
   - >= (not less than)
   - != (not equal)

In the Argument 3, set the real value to compare.
**Raw driving input**
This becomes active when using the actual operation value such as steering wheel and accelerator pedal as a condition on automatic driving or using ACC.

**To define a Raw driving input condition**
In the Exit Condition area, click Add.
In the Condition column, select Raw driving input.
In the Argument 1, select target devices for condition setting.
  - Steering wheel
  - Acceleration pedal
  - Brake pedal
  - Parking brake

In the Argument 2, set the condition to be greater or less than the specified value.
In the Argument 3, input the value. The range is [0, 100%].

**When exit condition are fulfilled**

**End event**
When all conditions are met:
  - The condition target is an event: The event ends when the exit target event occurs. (Refer to (4)Calling condition).
  - The condition target is a scenario: The event ends when the exit target scenario starts.
  - An event or scenario is not selected as a target: The event will end soon.
For details refer to [Manage 3D models...].

**End scenario**
Select an action in the combo box:
  - To end the event scenario: End this scenario
  - To end another scenario: select a scenario to stop.
  - To end all scenarios: End all the scenarios

**When all conditions are fulfilled:**
  - The condition targets are events: the selected action is executed when the exit target event occurs. (Refer to (4)Calling condition).
  - The condition target are scenarios: the selected action is executed when the exit target event stops.
  - Events or scenarios are not selected as a target: the selected action is executed immediately.

[Manage 3D models...] button
When the event ends, the dynamic model can be managed by "Event model form".
The model that is stopped or deleted can be selected.
Note: This button can only be used when "End Event" is checked.

(4) Calling condition
When the event is from multiple event targets, the call condition can be defined.
Select the event to display the call condition of the event.

In the following image, "Watch Speed" is stems from both "Start" and "Toll gate".

In the calling method, the combination of conditions can be specified.
In this case, select OR means either "Start" or "Toll gate" to trigger the event.
In this case, select AND means both "Start" and "Toll gate" to trigger the event.
(5) Event edit

User simulation tab

Manage the drive simulation on this tab.

Simulation command

Select it from the following command.

- Do nothing: This maintains no change and is a default command.
- Launch new vehicle: The new drive simulation is generated.
- Leave the vehicle: Even if the drive simulation ends when driving, the vehicle remains.
- Delete the vehicle: Delete the vehicle when the drive simulation ends when it drives.
- Start walking: Start a new walking simulation.
- End walking: End the walking simulation.
- Change ACC settings: Change ACC settings.
- Get in a new bicyclist model: Start a new bicycle simulation.
- Delete the bicyclist model: End the bicycle simulation.
- Start auto pilot: Start flight simulation on the selected flight route.
- End auto pilot: End the flight simulation.

New vehicle

Define the parameter of the drive simulation.

Previous vehicle

The previous vehicle can be defined before the new drive simulation begins. Select it as follows.

- Leave the vehicle: Even if the drive simulation ends, the vehicle remains.
- Delete the vehicle: Delete the vehicle when the drive simulation ends.

Add cluster client target

When a function of multi driver by cluster is enabled, a cluster can be added as a target on driving simulation and walking simulation.
Click [Add cluster client as a target] button to open a setting form for client. Set up driving or walking simulation and input a machine name displayed on the cluster master screen. For details, refer to the chapter “Cluster option plugin”.

It is possible to choose View Mode from do nothing, Interior View, Back View (close), Back View, Back View (far) or Top View.

<table>
<thead>
<tr>
<th>View Mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>Back View (far)</td>
</tr>
<tr>
<td>(do nothing)</td>
</tr>
<tr>
<td>Interior View</td>
</tr>
<tr>
<td>Back View (close)</td>
</tr>
<tr>
<td>Back View</td>
</tr>
<tr>
<td>Back View (far)</td>
</tr>
<tr>
<td>Top View</td>
</tr>
</tbody>
</table>

- **Moving Models tab**

Choose the travel model to operate.

There are five kinds of travel models.

- **Vehicle model**: Vehicle model that travels on road
- **Driving model**: A model used to travel on road. Any model is possible.
- **Flight model**: Model that flies along flight paths.
- **MD3 character model**: MD3 character model that travels along flight paths.
- **FBX character model**: FBX character model that travels along flight paths.
**Explanation of row**

**Model type**
Select the kind of dynamic model to be added.

There are 4 types which can be used in event. The model to be used should be setup and arranged in VR space.

<table>
<thead>
<tr>
<th>Movement</th>
<th>Contents</th>
<th>Preparation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moving vehicle</td>
<td>The vehicle model which travels along a road is added. It can be made to travel from arbitrary positions.</td>
<td>The model registered with the attribute of a car is required.</td>
</tr>
<tr>
<td>Driving model</td>
<td>Model which moves on a road. It can be made to start from arbitrary positions.</td>
<td>(All models can be set up.)</td>
</tr>
<tr>
<td>Flight model</td>
<td>Flying object along flight paths can be set up. The movement of bicycles, cars etc. can be expressed.</td>
<td>A flight route is set up. (All models can be set up.)</td>
</tr>
<tr>
<td>Character</td>
<td>Define a flight path, and set up MD3 character and FBX character. Pedestrians jumping out to a road, other pedestrians, animals etc. can be visualized.</td>
<td>Characters need a setup of a flight route, and to be registered.</td>
</tr>
</tbody>
</table>

**Model:** Select a model.

**Road/flight path:** Select a model to be travel on the road or the flight path.

**Lane:** Choose a lane that the model will travel on. This can be applied only to the driving model with the vehicle model.

**Driving From:** Choose the direction to move the model to. This can be applied only to the vehicle model, the driving model, and the flight model.

**Start position:** Specify the default position of the model and the road in the flight path.

**Initial speed:** Specify the initial speed of the model.

**Wanted speed:** Specify the speed at which they want to reach the model. This can only be assigned to vehicle model.

**Frequency:** Specify how often the model is generated.

For instance, 360/h means the same model is generated 360 times every ten seconds in an hour. 0 means generation only once.

**Visibility On Scenario Start:** Define the model. Check to maintain the model geostationary.

**Rotate:** Spin the model when it travels along the flight path. Only applicable to flight models.

**Movement:** If it is a character model, it is also possible to select other pre-defined movement animation from the list. This can be done for both MD3 and FBX models.

**Auto custom ID:** When checked, custom ID are added automatically. When unchecked, manually.

**Custom ID:** Select custom ID manually. For details, refer to Custom ID.

**Model Control**

![Model Control](image)
Control Target

[Add]: Add the vehicle controlled in this event.
The default is the simulation vehicle. Double-click in the Model Type of the added item and select a model in the select target model form. Especially, if selected the 'Event Type', it is possible to select one event vehicle that has already been registered.

[Insert]: Insert a target model in the current position.

After done that, how to select the target model is the same method in above-mentioned, Add button.

[Delete]: Delete the current selected target model.

Setting Parameters

Set the parameters for each target model respectively.

Note: In the case of flight simulation, only Target speed on Behavior can be defined. Change lane, Lane Offset, Stop Sound and Play Sound are disabled.

Behavior

- normal (auto): resume the normal AI behavior
- target speed: modifies the vehicle speed. Note: TTC is given priority
- exact speed: the vehicle will reach the exact speed at the specified time
- acceleration: defines the acceleration of the vehicle
- keep distance with simulation car: the vehicle tries to keep the specified distance without going over the specified acceleration and deceleration values. Once the distance is reached it will keep the same speed than the simulation vehicle.

Note: it isn’t the direct distance between the two vehicles but the distance between each vehicle along the road.

The position of each vehicle used for the distance calculation can be specified as follows:

_ default: the vehicle default position
_ front: the vehicle front edge
_ rear: the vehicle rear edge
_ frontLeft: the vehicle front left edge
_ frontRight: the vehicle front right edge
_ rearLeft: the vehicle rear left edge
_ rearRight: the vehicle rear right edge

Notes:

- In intersections, the distance of a vehicle along the road cannot be calculated so it must be approximated. Therefore the vehicle behavior cannot be guaranteed to be exact in intersections.
- Both vehicles must be on the same road, or a ramp belonging to the road.
- The vehicle cannot move backward.

Change Lane:

To Right Lane, To Left Lane: Select to which lane the vehicle move to

Distance: Set the distance necessary for the lane change after the event is generated (input range 10 - 200m)

Immediate lane change (can collide with other vehicles): Check if the lane is changed immediately after the event is generated.

[Hint]
The selected vehicle moves to right or left lane by this setting. At this time, set the distance in consideration of the distance for the lane change and the movement timing as well as the idea of the waypoint setting. Set more than the distance that moves at about almost two seconds because it becomes unnatural movement according to the speed when the distance necessary for the lane change is too short.

If the [Immediate lane change] is checked, the lane is changed even if there is another car in the lane changing ahead. If checked
off the option, according to the vehicle condition in the changing lane a head, after adjusted the vehicle speed to keep the safe following distance, the lane is changed.

**Lane Offset:** Sets maximum and minimum value of the amount of the offset. (Input range: -100.0 - 100.0 m)

[Hint] The maximum and minimum value of offset from the center of the lane is set. The left side is the negative value and the right side is the positive one toward the moving direction. An actual amount of the offset is set at random between the maximum value and minimum value.

**Stop Sound:** Stop the sound played.

**Play Sound:** Plays the selected sound. Select a sound with [...] button. See Sound Model Editor for further details

**CustomID:** Check Custom ID to assign an custom ID to a model.

Note: Same custome IDs can be assigned to some models.

- **Command Execution**

Use the Command Execution tab to specify the scene models to be controlled. For example, it is possible to have traffic lights blink, railway crossing gates open, balloons fly, etc...

---

**Explanation of row**

**Model:** Select the scene model to be a movable model.

To make a model scriptable, right click on the model and in the Model Tool Form select Scriptable.

**Action:** Select a model command.
Multimedia model tab

Define the multimedia model that reproduces.

There are four kinds of multimedia models.

Sound model: Reproduce the sound.
Image model: Display the image in the view.
Message model: Display message on the screen.
Virtual Display model: Display the virtual display model.

Transparency Change: Change the transparency of the image.

Note: All the multimedia models stop when the event ends.

Model type: Choose a type from Sound, Video, Image, message, Virtual Display or Transparency Change.

File: This is the information of the model.
The Sound Model Editor, the Image Model Editor,
The Message Model Editor, the Video Model Editor and the Virtual Display Manager can be opened.

Start: Specify time for the multimedia model playback.

Duration: Set the duration of multimedia model playback. 0 means to continue playing and displaying until the event ends.

Note:
The multiple sound models can be activated simultaneously.
Only one image model can be displayed at one time. A new image model is always given priority over the previous one.
Only one message model can be displayed at one time. A new message model is always given priority over the previous one.
Only one animation model can be played at one time. A new animation model is always given priority over the previous one.
Traffic Signals tab
Use the Traffic Control tab to modify the traffic phases. Only intersection controlled by signal and in which one phase of the signal is set at least is displayed in the list.

Extension
There are several additional sub-tabs under the Extension tab. This is mainly used in conjunction with other plugins to incorporate them into events.
The tabs are as follows:
・ EXODUS
・ Micro Simulation Player
・ xpswmm
・ Replay
・ AVI
・ Traffic Snapshot
・ ECO Drive (available when Eco Drive plugin is activated)
・ Log (available when Log Export plugin is activated)
EXODUS

Under EXODUS tab, the animation assigned to the event currently editing can be further manipulated.

*If EXODUS plugin is not enabled, the tab will become gray and unavailable.

Enabled: Enable the EXODSU feature.

EXODUS commands

- **Play**: Play the animation
- **Play in loop**: Repeat the animation.
- **Stop**: Stop the animation playback.
- **Pause**: Pause the animation.
- **Next Step**: Move forward to the next animation step.
- **Previous Step**: Move backward to the previous animation step.
- **Change Play Speed**: Alter playback speed to the percentage value specified.
- **Jump**: Jump to a certain playback time on the animation.
- **Show Trajectory**: Shows the trajectory. Select from All / selected / none from the pull down menu...
- **Change camera mode**: changes the camera mode.
  - **Camera mode**: Select from the following camera modes: Free / Helicopter / Tracking View / Avatar View / Exploration
  - **Selected avatar**: Specify the avatar’s ID for use in Avatar View.
  - **Tracking direction**: Specifies the direction during Tracking View...
Micro Simulation Player

The Micro Simulation Player tab grants control of animation playback controls in events.
*If Micro Simulation Player plugin is not enabled, the tab will become gray and unavailable.

- **Enabled:** Enable the Micro Simulation Player
- **Microsim Commands:**
  - **Play:** Start playing the animation normally.
  - **Play in Loop:** Play the animation repeatedly.
  - **Stop:** Stop the animation playback.
  - **Pause:** Pause the animation.
  - **Next Step:** Move animation to the next step.
  - **Previous Step:** Move animation back to the previous step.
  - **Change play speed:** Alter playback speed to the percentage value specified
**xpswmm tab**

On the xpswmm tab, it is possible to set the controls of xpswmm plug-in. It is required to load the analysis data of xpswmm to use this tab.

*If xpswmm plugin is not enabled, the tab will become gray and unavailable.

---

**Enabled:** Enable the xpswmm functions

- **Play:** Play XPSWMM Plugin by the following parameters.
  - Play speed: set the speed of the play.
  - Start step: set the start step of the play.
  - End step: set the final step of the play.
  - Play in loop: Repeat the play.

- **Stop:** Stop XPSWMM Plugin playback.

- **Pause:** Pauses XPSWMM Plugin playback.

- **Previous step:** Go back to the previous step.

- **Next step:** Go to the next step.

- **Change draw:** Set the detailed elements to be drawn:
  
  - **Draw water level:** Draw the water surface.
  - **Height contours:** Draw the water surface based on contour of heights.
  - **Water reflection:** Draw the simulation result water surface with water reflection.
  - **Draw velocity vectors:** Show the velocity vectors of water flow.
  - **Draw pipework:** Draw the pipe networks.
  - **Draw time:** Show time.
Replay Tab

On Replay tab, users can record the situation encountered during the event, and replay for later use.

*If Replay plugin is not enabled, the tab will become gray and unavailable.

Enabled: Enable Replay

[Record and replay]

* Command

- **Start Recording**: Starts recording vehicle, pedestrians, or traffic signals. If data has already been recorded, this command will overwrite previous recording. Use Restart Recording for no overwriting.

- **Stop Recording**: Stop recording.

* Pause Recording: Pause recording.

- **Restart Recording**: Start recording. If records already exist, the recording will continue from the previously recorded replay.

- **Replay**: Playback the record. All scenarios are stopped during replays.

  Replays are played prior to other events (e.g. mobile model generation, context changes). For example, assigning “launch a new vehicle” from the Driving Walking Simulation tab, will immediately take the user into a car and resume the driving after all replay has been done. Replay will make AI pedestrians and cars disappear. Thus after a replay it is recommended that the traffic gets manually regenerated.

* Method of Replay

- **Replay from beginning**: Replay from when the recording event started.

- **Replay last**: Replay from n seconds before the recording. Specify the time directly in the box.

* View Point:

  - **Cockpit View**: Use the mouse to manipulate the camera angle. The mouse wheel can be used to switch to the third person rear view.

  - **Rear View**: Similarly, the mouse can be used to manipulate the camera angles and during replay, and the mouse wheel can switch back to cockpit view.

* Other

- **Restart recording on replay’s end**: Restart recording after the replay has ended.
AVI
Start/stop AVI recording.

[Command] combobox
Select the command, and start AVI recording or stop AVI recording. If the field is empty, nothing will happen.

Traffic Snapshot
Use the Traffic Snapshot tab to make an event on which the traffic snapshot registered is executed in the Traffic Snapshot List Editor preliminary
Pull down the combo box and select a traffic snapshot to execute on this event.
**On DSPlugin tab**

On DSPlugin tab, it is possible to control the lock function for the seatbelt on Driving Simulator by the scenario, and define the aux variables to send to CarSim by the event.

Note: If DSPlugin tab is not on the Extension tab page, check whether the DS Plug-in has been loaded correctly or not in the License Manager.

**Aux variables for CarSim by Scenario**

It is possible to define the aux variables to send to CarSim by the event function of Scenario of UC-win/Road.

Hint: This function is available with CarSim 8.1 or later.

**Seatbelt lock function**

This function simulates the fastening the seat belt at the sudden braking etc. driving with the seatbelt fastened. Control it in the event of the scenario. After locked it once, release it.

Hint: This function is available for SIMREX simulator

The command of the seatbelt control is as follows;

- **(blank)**: The seatbelt command is not executed
- **Lock**: Locks the seatbelt
- **Release**: Releases the seatbelt

When driving finishes, Release is automatically done as for the lock function of the seat belt.

**Change Value**: The value of each variable is sent to CarSim when the event occurs.

Hint

- It is required to set up settings in CarSim preliminarily to send the aux variables to it.
- When once the event that variable are changed, UC-win/Road continues to send the same values to CarSim by the value changed in the next changing event.
**ECO Drive**

Use the ECO drive tab to put the border model for the border of carriageway on the intersection when logged on driving of vehicle by ECO drive plug-in.

Note: This tab is displayed when the ECO drive plug-in (none free) has been loaded. So if it is not displayed on the page, load it in the License Manager.

The shortest distance between the moving vehicle and the border of the carriageway on both sides of the road is output to the log when driving by ECO drive plug-in. However, the border does not exist at the Intersections. Placing a 3D model defined as a border model between the borders of the road on the intersection will allow the system to continue logging as the border model substitutes the border of the carriageway.

**To Create the 3D Model**

Create a long and thin cuboids for the 3D model.

The model can be created selected the menu, New - Building model on the Models panel form that opens by selecting Library form the ribbon, Edit - Scene.

Hint: The model can be also created by another tool for creating the models.

**How to put and set for the border model**

Select the model used for the border model on this page. See The Border Model Use for more details to set for it.
With the Log tab, users can control data log outputs. Users can also assign a model that records the distance from the current driving car (e.g. parking lot gate).

*If Log plugin is not enabled, the tab will become gray and unavailable. For more details on data logging, see [Log Export Plugin].

**Command**

Select the command that will occur during events.

- Set Profile: Open the profile setting dialogue for event logs.
- Start Logs: Initiate logging.
- Stop Logs: End logging.

**Distance to**

Assigns models used for distance measuring.

- **Add**: Add a scene or event model that measures and records the distance between itself and the car currently driving. After clicking add, if Scene Model is selected for Model Type, users must also define the specific model for distance measuring from the list in Argument 1, and the event model's ID in Argument 2

- **Delete**: Delete the currently selected distance measuring object.

**Note** This feature will not work when using UDP Stream Export features.
User Variables

It is possible to rewrite the values of the elements of the user variable array at any timing in the scenario. From the Event Editor form, open the "User Variables" tab and select the command. Selectable commands are assignment and clear.

[Assign]: Substitute the specified value for the checked element.
[Clear]: When executing the event, all values in the user variable array are set to "0".

Other tab

[Simulation] box
Reset the traffic: Reset the traffic on the command executing
Context name: Select the context of event to use. If the field is empty, nothing will happen.
Vehicle behavior randomization: There are three choices, "Do not change", "Enable randomization", and "Disable randomization". When selecting the "Enable randomization", we make each vehicle's behavior random in the traffic simulation. The maximum speed, the offset from the lane center and the vehicle weight at the time of generation are set randomly. When selecting the "Disable randomization", the randomness of behavior of each vehicle will not occur in the traffic simulation. The vehicle moves as same conditions.
[Screen Effect Settings] box

Screen effect Type: Select an effect type
- No effect: This is the default.
- Fill screen: Fill the screen in a single color
- Fade in: After filling the screen with a single color, the color of the screen will be gradually transmitted. Then finally, 3D screen view is displayed.
- Fade out: Gradually fills the screen in a single color.
- Stop effect: Stops the current effect.
  - Effect time: Set the time from occurring the event to the end. Unit: 0.01s
  - Effect color: Set the color to represent the effect.
  - Max alpha: Set the alpha value when the effect becomes maximum.
- Screen effect is displayed over the HUD: When HUD object is displayed, the effect will be reflected to the over the object.

[Load Plugins] box

Load and unload the plug-in on executing a scenario. It is used when switching the scenario-dependent function extensions, for example, the plug-in used for each scenario for advanced control and switching the control pattern,
  - Add button: add the executing plug-in to the list
  - Delete button: delete the unneeded plug-in from the list

When the event is activated, the BPL file of selected plug-in is loaded and runs in the same way as a normal plug-in. While loaded, the original BPL file is write-protected and cannot be deleted.

The plug-in is unloaded at the end of the scenario. After that, it is possible to overwrite the plug-in's BPL file.

Note:
  - The plug-in must exist in the same folder or subfolder of the RD of UC-win/Road.
  - The plug-in will not be unloaded at the end of the EVENT.

Edit shortcut

An unused button of a game controller can be used as a shortcut key to start scenario.
Go to Ribbon Home – Scenario to open Scenario Manager, and click Edit Shortcut.

Pushed button: The button number is displayed. It can be used for a shortcut button.
Add: Add an item.
Action: Start scenario.
Button: Select button to start scenario. Assign a button number displayed on “Pushed button”.
Object (Target scenario): Select a scenario to start.
button: Delete the settings.
4. Setup event route
Although multiple data input of the route can be carried out, it cannot pass through the same place due to the event setup.

[Setup for event control point]
Motion control point (Way Point) can be set at the position where event is triggered at the position where the event ends. On the Plan View, right-click on the horizontal road alignment, and select the menu, [Add] - [Add way point road #]. Select the command, "CHECKPOINT"

Example
Appearance of pedestrian jumping out  Collision occurs
Micro Simulation Player

The micro simulation player can reproduce the simulation result from other applications in UC-win/Road. Microsimulation of traffic, construction management and pedestrian movement etc. can be reproduced by using 3D models in UC-win/Road. Moreover, the traffic signal cycle, vehicles and the movement of characters can be recorded. And it is also possible to record the traffic in UC-win/Road as simulation and reproduce it in this plug-in, and to display movements of objects that are not on roads, such as those on pavements and flight path.

1. The micro simulation editor
Go to ribbon Record/Play Ribbon - Micro Simulation - Editor/Player.

Load the simulation data
It is possible to load XML files in the OpenMicroSim format, VISSIM files, S-PARAMICS files and Legion XML files.

List of simulation
Each simulation list from each loaded file is listed.

When a fixed element exists in the simulation, it is listed as a child node. (A fixed element can only be allocated to the existing model of UC-win/Road)

Each fixed element can be allocated to the model. Select a default model from the list in the existing model by clicking the display under the element to allocate the model to a fixed element.

When the model is not allocated to the fixed element at all, "The model is not allocated" is displayed. Click it in the list to separate a fixed element and the model. (The element allocation cannot be corrected while the animation is playing)

2. Record traffic with the micro simulation recorder
Go to Record/Play Ribbon - Micro Simulation – Recorder. The models’ movements are saved in the Open Micro Simulation format (.xml), which can then be used in a simulation. It is possible to record vehicles, characters or traffic signals.

Begin the recording and stop it at arbitrary time.
Check object(s) to be recorded. (It cannot be reselected under the reproduction)

Then click [Begin]. It changes into [Stop], and the length of the recording time is displayed while recording.

* For the format of XML file, refer to "Help"-"Technical note"-"Micro Simulation Player"-“File Format Support” and “Open MicroSim Schema”.

The information site of “Open MicroSim Schema” is opened. http://openmicrosim.org/
3. Load Simulation

Click Add/Edit simulation icon to load the xml file.

**List of simulation:**
- Each simulation list for each loaded file is listed
- Click the radio button to select the simulation.
- Each simulation becomes a route node. When a fixed element exists in the simulation, it is listed as a child node.
- A fixed element can be allocated to the model in the tree view.
- Click on the line under the element to select a model from the list of the existing models.
- Click "No Model" to separate a fixed element and the model. (It cannot be modified while the animation is playing)

**2D view**

The models and 2D view of the road are displayed in the preview.

**Tab**

**Space movement tab**

<table>
<thead>
<tr>
<th>Size:</th>
<th>Translation (x, y, z):</th>
</tr>
</thead>
<tbody>
<tr>
<td>100%</td>
<td>0.000 m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Rotation:</th>
<th>0.000 deg</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.000 m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>South → North</th>
<th>Coordinates system:</th>
</tr>
</thead>
<tbody>
<tr>
<td>-Y → Y</td>
<td>Local</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>West → East</th>
</tr>
</thead>
<tbody>
<tr>
<td>X → X</td>
</tr>
</tbody>
</table>

There are all transformation parameters in this tab. It is possible to resize, rotate, and translate the simulation.

The movement of elements can be displayed by moving the slide bar of time under 2D view.

Click ○ mark, the state of the element is displayed on the lower tab. It is updated according to the time.

Click the radio button to add the selected simulation to the playlist.

"Define a target zone" can rotate the simulation of selected range or change the scale of it.
4. Profile creation

Create profiles. The profile can be allocated to each type of simulation. 
Go to "Manage profiles and types".

5. Assigning of simulation type to the profile

The profile created in step 3 can be assigned to the simulation on "Assigning of simulation type to the profile" tab. 

*Weight (1-99) can be set to each model.
*The profile cannot be corrected while animation is reproducing.

Element type of all simulations is categorized.

The type and the profile can be separated by clicking
"Undefined profile" by the profile list.
When there is no profile set to the type, the element that is not allocated will not be displayed in the playback.

When types are allocated, all elements not allocated are allocated to the models from the profile at random. The instance of the model is generated when animation is begun in UC-win/Road, and it is related to the element.
6. Simulation Playback

Click "Add to the playlist" on add/edit tab of the simulation to add the selected simulation to the playlist.

The added simulation can be confirmed on the Simulation Playback Tab.

**Playlist contents**

**List of simulation:**

**Time line:**
- Each bar shows one simulation. The name is displayed on the left, and revokable to drag the bar with the mouse to change the start time. (The color of the bar can be changed.)
- The track bar shows the run time of the animation playback. The red line also shows run time and follows the movement of the track bar. Click the timeline to update. (The line turns to red during playback.)
- Display color can be selected from the color chart.
- Add to the current simulation. The popup Normal, Invisible and Solo are shown.
- Select a time unit.
- Delete the simulation from the playlist.
- Zoom in/out.
- Restore the size.

**Outline tab:**

In this tab, the following information is displayed.
- Ratio between simulations in the playlist
- Position of the track bar when invisible due to zoom
- The part of animation that is currently visible in the timeline (The part invisible is displayed in lighter color)

**Controls**
- Previous step: Back to the previous step.
- Play: Play animation.
- Pause: Stop the animation temporarily.
- Stop: Stop the animation.
- Next step: Skip to the next step.
- Repeat: Repeat the animation.
- Ignore Traffic Generation: New traffic flow is not generated if Traffic Generation is inactive.
- Playback time: Run time of animation
- Speed: Change the reproduction speed in %.
7. Toolbar

Microsimulation player's toolbar can be opened by right-clicking on the main screen of UC-win/Road and clicking "Display the micro simulation player toolbar" from the pop-up menu.
**Replay Plug-in**

This plug-in saves movements of vehicles and pedestrians in UC-win/Road, and replays them. It is possible to confirm scenes of an accident from the driver's perspective or outside the vehicle, and observe traffic simulation at an intersection.

### 1. Operation Panel

Go to Record/Play - Replay - Player to open the operation panel.

<table>
<thead>
<tr>
<th>Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Open replay data file" /></td>
<td>Open replay data file (*.RR). This button is inactive while recording or replaying data.</td>
</tr>
<tr>
<td><img src="image" alt="Save data" /></td>
<td>Save the data into a file (*.RR). This button is inactive while recording or replaying data or having no recorded data.</td>
</tr>
<tr>
<td><img src="image" alt="Jump" /></td>
<td>Jump to the start of the data.</td>
</tr>
<tr>
<td><img src="image" alt="Play backward (Slow)" /></td>
<td>Play backward (Slow).</td>
</tr>
<tr>
<td><img src="image" alt="Play backward (Fast)" /></td>
<td>Play backward (Fast).</td>
</tr>
<tr>
<td><img src="image" alt="Play backward" /></td>
<td>Play backward.</td>
</tr>
<tr>
<td><img src="image" alt="Play" /></td>
<td>Play.</td>
</tr>
<tr>
<td><img src="image" alt="Pause" /></td>
<td>Pause recording or playing.</td>
</tr>
<tr>
<td><img src="image" alt="Stop" /></td>
<td>Stop recording or playing. Stop recording to save it. When [Automatic recording on drive simulation] options is checked, it is automatically stopped at the end of drive simulation.</td>
</tr>
<tr>
<td><img src="image" alt="Start recording" /></td>
<td>Start recording. When [Automatic recording on drive simulation] option is checked, the record starts automatically when starting a drive simulation.</td>
</tr>
<tr>
<td><img src="image" alt="Fast Forward" /></td>
<td>Fast Forward.</td>
</tr>
<tr>
<td><img src="image" alt="Play slowly" /></td>
<td>Play slowly.</td>
</tr>
<tr>
<td><img src="image" alt="Jump" /></td>
<td>Jump to the end of the replay.</td>
</tr>
<tr>
<td><img src="image" alt="Open Option form" /></td>
<td>Open the Option form.</td>
</tr>
<tr>
<td><img src="image" alt="Help" /></td>
<td>Open the Help.</td>
</tr>
<tr>
<td><img src="image" alt="Pin" /></td>
<td>Pin the Replay plug-in control panel.</td>
</tr>
</tbody>
</table>
2. Option
Go to Tools-Replay plug-in, and click Options.

<table>
<thead>
<tr>
<th>Record</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Automatic recording on drive simulation: Recording starts automatically when start driving.</td>
<td></td>
</tr>
<tr>
<td>- Records in an area from the camera: Record movements within a certain range. The file size can be reduced.</td>
<td></td>
</tr>
<tr>
<td>- Specify the recording time: Record only for a certain time from the start or the end of the recording.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Auto Save</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Automatically save the file after recording: The data is automatically saved. The file name is as follows. YYYYY-MM-DD_HH-MM-SS_[Scenario name]_[RD filename]_Replay.rr</td>
<td></td>
</tr>
<tr>
<td>For driving without scenario, the scenario name is set to [NoScenario]. For new project without saved, the RD filename is set to [NoFilename].</td>
<td></td>
</tr>
<tr>
<td>- Create date folder: The data folder is created in the auto save folder, where the replay file is saved.</td>
<td></td>
</tr>
<tr>
<td>- Auto save folder: Specify the auto save folder. If it is blank, the data is output to [User data folder]¥ReplayFiles.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Play</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Disable on playing</td>
<td></td>
</tr>
<tr>
<td>Delete models which loaded already: the models (vehicles and characters) are temporarily deleted. Disable the new traffic generation: New traffic is not generated during the playback.</td>
<td></td>
</tr>
<tr>
<td>- Camera mode on traffic simulation: Make a view from my cockpit: The cockpit view is enabled during the replay. Follow my vehicle: A view from the rear of vehicle is enabled during the replay. Note: Without checking the camera mode, the camera position does not move.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Others</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>Working folder: Specify a folder to store the temporary recorded data. As the data might be bigger than 1GB, specify a drive with enough free space.</td>
<td></td>
</tr>
</tbody>
</table>
3. Replay Search form
Go to Record/Play-Replay - Search. It is possible to search and replay the recorded playback file on this form.

Hint
It is possible to sort and refine the columns, [Date], [Time], [RD file] and [Scenario]. Click on the top of the column to open the pop-up window. They can be sorted in "Ascending Order" or "Descending Order", and refined with substring. Items which are unique are listed with checkbox. Check it off to exclude for the target.

4. Toolbar (Replay Option)
Right-click on the main screen, and click [Show Replay Tool] to open the toolbar.
* It is enabled only when "Replay plug-in" is active.
Log Export Plug-in

The log export plug-in enables to export information of simulation as a csv file or through a UDP stream to another machine on the same network. It is possible to export information about user’s vehicle, the vehicle in front of the user’s vehicle and other moving objects. The operation can be done manually or synchronized with a scenario. When exporting to CSV, it is possible to add fixed objects selected in the scenario to the log content, which exports a distance between objects and the user’s vehicle.

Setup the UC-win/Road environment

License Manager

Check if Log Export plugin is in Loaded plugins list from File – License Manager.
Select LogExportPlugin.bpl in Available plugins list and click Load. Repeat the operations for ScenarioPlugin.bpl as needed.

Note: If LogExportPlugin.bpl or ScenarioPlugin.bpl is not found, find it in UC-win/Road plugins directory(¥FORUM 8¥UCwinRoad¥Plugins¥), copy and paste them in this directory.

Output log file directory

The output log file will be saved to<User data directory>¥Log¥ folder.
It can be changed at “Folder and files” section in the Application Default windows opened by File menu - Application Options - Application Default.

Log option screen

Go to Diving Sim Ribbon - Log Export - Options
The other options are as followings.

**Vehicle in front**: Export the information of the vehicle in front of the user’s vehicle. It is automatically detected.

**Surrounding moving objects**: Export the information of all the surrounding moving objects (vehicle, pedestrians, trains etc…) inside a limit radius which is set in **Radius for surrounding objects**.

**Other moving objects**: Export the information of all the moving objects in the scene. This option can slow down the simulation. Set an appropriate radius to maintain high FPS.

**Pedestrian density**: Output pedestrian density. Arrange an object to measure the density on a pass.

**Pedestrian flux**: Output pedestrian flux. Arrange an object to measure the flux on a pass.

**Interval**: Timing of log output.

**Time precision**: The number of digits after a decimal point of the time.

**Radius for surrounding objects**: Radius in which area the objects are exported to the log information.

The log files depend on the setting of profile as followings.

1. In case of single user
   "Log_" + YYYYMMDD + HHMMSS + "." + RoadName + "." + UserNumber + "." + Gender + "." + Age + "." + DrivingExperience + ".csv"

2. In case of Multi user
   - User log file name: log_filename + ".User_" + machine name + ".csv"
     The machine name is "master". For the other machine, the machine name on the client cluster setting will be used.
   - Other vehicle, log filename: Log_filename + ".OtherMovingObjects.csv”.

**Basic functions**

**Start logs**
Start logs by clicking 💚 or triggering an event which has a Start Logs command selected in the Log tab of the Event Editor.

**Stop logs**
Stop logs by clicking 🚫 or triggering an event which has a Stop Logs command selected in the Log tab of the Event Editor.

Note: Log stops automatically when stopping driving or when the scenario which contains the Start Logs event ends.
Set profile (Optional)

Set a profile which will be used in the output log file name.

Open this form by clicking or triggering an event which has a Set Profile command selected in the Log tab of the Event Editor.

The log file name: "Log_" + YYYYMMDD + HHMMSS + "_" + RoadName + "_" + UserNumber + "_" + Gender + "_" + Age + "_" + DrivingExperience + "_.csv".

For example, the output log file name is Log_20120315093000_Road 1_10_0_24_5.csv means it has done in 03/15/2012 at 9:30:00 am on Road 1.

And the profile is as follows.
User number = 10, Gender = Male, Age = 24 years old, Driving experience = 5 years

Note: This function is not effective on UDP output.

How to use the Log Export with Scenario

It is possible to configure an event which will launch the log start/stop/edit during a scenario.

Setup a scenario

Create a scenario from ribbon Home - Scenario. Click on Add in the Scenario Manager form. Select a Scenario and click on Edit to open the Scenario Editor.

Setup a Start Logs event

It is possible to create an event which will start logging.
Click Add to create a new event in the Scenario Editor, and select the one and click on Edit. In the Event Editor, select the
Extensions and Log tab.

And select **Start Logs** in the command list.

Setup a Stop Logs event
It is possible to stop logging by events. Create a new event and connect it with the previous one. Then choose Stop Logs in the command list.

Setup an event which opens the profile editor
It is possible to open the profile editor by events. Create a new event and connect it with the previous one. Then choose Set Profile in the command list. Refer to Set profile for details.

Select the object to export to the log file
Choose one or more models which will be used in the distance computation.

- To save the distance between a driving vehicle and a scriptable scene model in the log file, choose Scene Model as a model type, and select a scriptable scene model. Refer to make a scriptable scene model for details.
To have the distance between a driving vehicle and a moving object created during an event, choose Event Model as a model type and select the ID number of the moving object model. Refer to how to find the ID number of a moving object model in a specified event for details.

Note: In the output log file, it is possible to get the distance between the driving vehicle and the selected object only when the event is executed. When the event ends, calculation will stop, and the distances are not exported anymore. It is possible to activate several events at the same time in a scenario.

Note: This function is not usable for the UDP stream output.

How to make a scriptable scene model
It is possible to set scriptable scene models in the Argument1 when selecting Scene Model as Model Type on the Log tab in the Event Editor. This operation needs LogExportPlugin.bpl.

1. Left-click a model in the 3D view.
2. On the Model tool form, check Scriptable on the Options tab.
3. Click OK.

How to find the ID number of a moving object model in a specified event
Select an event which has moving object models, and find the ID numbers on the Moving Model tab in Event Editor.
Distance between your vehicle and a model instance

Set a model instance on the Log tab in Event Editor to export the distance between a driving car and a model. If the model instance is moving, the instance position is the front center of the car.

If the model instance is not a vehicle or is not moving, the instance position is the center of the model.
VISSIM Link Plug-in
This is a plugin for the cooperation of UC-win/Road and VISSIM in real time. This plugin can send information of a driving vehicle in UC-win/Road to VISSIM in real time using DLL and execute a traffic simulation of the vehicle in operation and surrounding vehicles on VISSIM. The result of VISSIM traffic simulation (calculation results of surrounding vehicles and traffic lights) are sent to and visualized in UC-win/Road.

Hint: The VISSIM Ver.11 or later is supported.

The following data can be sent to VISSIM and visualized in UC-win/Road.
- A vehicle in operation (position and speed)
- Surrounding vehicles (position and speed)
- Traffic signals

Simulation flow
The information of vehicles and signals are mapped between UC-win/Road and VISSIM, so that the information can be sent to and calculated in VISSIM, then it can be reflected in UC-win/Road.

How to use
(1) On VISSIM
- The VISSIM ver.11 or later is supported.
- VISSIM uses COM interface to get setting information. Set up VISSIM as COM server. Refer to the operation manual of VISSIM for details.
- At first, check Driving simulator active on Network settings form on VISSIM, and select the one of the vehicle type.

(2) License Manager
Enable VISSIM Link Plugin(VISSIM Ver.11)[VISSIMPlugin.bpl] on License Manager.
On Simulation Link ribbon, VISSIM group is added.

(3) Prepare VISSIM data
Prepare VISSIM data to link with UC-win/Road.

(4) Create a project
Create a project to link with UC-win/Road. As needed, set up the following settings.
- Register a vehicle model to use.
- Register a pedestrian model to use.
- Add a road or signal to use.
Hint: Save the project to save the working time from next time.

(5) Settings
Click Settings icon on VISSIM group to open the settings form.

(6) Signal settings
Click a signal model to open the Model Tool. Set up the Signal Controller ID and Signal Group ID.

(7) Start connection with VISSIM
Click Connect on VISSIM group to start connecting and open VISSIM road network data.
Hint:
- If the VISSIM network data has not been assigned, assign it on file dialog box.
- If VISSIM runs with the command line –automation, connect to the VISSIM and open the data. Otherwise, run VISSIM, connect it and open the data.

(8) After connection

Click \[ \text{button} \] from the main ribbon Home – Simulation to start the data exchange. Click \[ \text{to stop} \], and click \[ \text{to pause} \].

When driving starts on UC-win/Road, the information of driving vehicle is sent to VISSIM. Then information of other vehicles and signal color, etc. are reflected on UC-win/Road.

Hint: If the connection does not work, disconnect it and confirm whether Driving simulator is enabled and whether the vehicle type is selected on Network settings of VISSIM.

(9) Disconnect

Click Disconnect to end the connection.

Restriction

VISSIM information contains articulated bus, cab(trailer) and train with some cars, however, only the segment 1, which is a vehicle model or a first vehicle model, can be reflected.

Settings

Set up items for connection between UC-win/Road and VISSIM

Click Settings \[ \text{from ribbon Simulation Link – VISSIM} \].

Setting items

- VISSIM data name: Select data from Open.
- VISSIM Vehicles: Import vehicles from VISSIM. Click it to open VISSIM Vehicles Editor, and click Get from VISSIM
- Road Vehicle List: Register vehicles or pedestrians on Road Vehicle List Editor.
Mapping (VISSIM vehicles <-> UC-win/Road vehicles)
After importing VISSIM vehicles and UC-win/Road vehicles, select UC-win/Road vehicles mapped to VISSIM vehicles.

Hint: Vehicles, which are not mapped, are displayed as a vehicle model registered first.

VISSIM → UC-win/Road
Offset
Set up offsets for X, Y and Height. Information from VISSIM is imported in the UC-win/Road local coordinate, which is X for east and Y for north, so that the imported information are not displayed at the center of project.

UC-win/Road → VISSIM
Apply vehicle height
Check it to apply vehicle height during driving simulation. The vehicle height information are sent to VISSIM.

Connection Condupex to VISSIM
Set up parameters sent to VISSIM.
- **Simulation Frequency**: Set up FPS of UC-win/Road in Hz.
- **Visibility Radius**: Set up visibility radius. Set 0 or less means entire range.
- **Max VISSIM Vehicles**: Set up the max number of VISSIM vehicles to be imported.
- **Max VISSIM Pedestrians**: Set up the max number of VISSIM pedestrians to be imported.
- **Max VISSIM Signal Groups**: Set up the max number of VISSIM signal groups to be imported.

Simulation Time Option
- **Fetch Result Interval**: Set up fetch result interval in seconds, which is the interval to get information. The default is 0.025.
- **Moving Delay Time**: Set up moving delay time in seconds. The default is 0.15.

Hint: Click OK to confirm the settings. Then save RD file to save it to the project. However, the parameters on Simulation time option are saved to the registry.

VISSIM Vehicle Editor
Set up VISSIM vehicles. Click VISSIM Vehicles on Settings form.

**Add**: Add a vehicle.
- **No**: Vehicle ID
- **Segment No**: Segment Number
  1 for single model.
  1,2,3,... for articulated model such as cab(tractor), trailer, tram.
- **File name**: File name corresponding with VehicleID

**Delete**: Deleted the selected vehicle

**Get from VISSIM**: Get information from VISSIM automatically.
Road Vehicle List Editor
Register UC-win/Road vehicles to be mapped.
Hint: It is recommended to register necessary models in advance in UC-win/Road
Click Road Vehicles on Settings form.

Add: Add a vehicle or pedestrian, and select a thumbnail for it.
Delete: Delete the selected vehicle or pedestrian.
Load: Load a new vehicle model(*.Rm).
Download: Download a vehicle model(*.Rm) from RoadDB.

Hint: Only vehicle models can be loaded and downloaded.

Signal Settings
Assign Signal Controller ID and Signal Group to the signal models on the project.

Click a signal model to open Model Tool, and go to VISSIM tab.
Simulator
ECO Drive Plug-in

1. About the ECO drive

This plug-in calculates the fuel consumption and the carbon-dioxide emissions of a vehicle during driving simulation, and produces graphs from data logged by UC-win/Road. It is possible to use it for Eco-driving training and ITS research with driving simulator.

\[ E = K_e \left( 0.3T + 0.028D + 0.056 \sum_k \delta_k (v_k^2 - v_{k-1}^2) \right) \cdots \text{式(1)} \]

ここで、
E: 旅行時間Tに対する二酸化炭素排出量 (kg-C)
T: 旅行時間 (sec)
\( v_k \): 第k点における走行速度 (m/sec)
D: 旅行距離 (m)
\( K_e \): 排出係数 (CO2 0.00231kg-C / ガソリン cc)

※1 大口・片倉・谷口都市部道路交通における自動車の二酸化炭素排出量推定モデル
土木学会論文集 No.695/IV-54,125-138,2002.1

2. ECO drive plug-in toolbar

The following operations can be done with buttons from Driving Sim - ECO Drive.

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Begin Logging</td>
<td>Begin the data logging.</td>
</tr>
<tr>
<td>End Logging</td>
<td>End the data logging.</td>
</tr>
<tr>
<td>Display Results</td>
<td>Display the ECO drive analysis results.</td>
</tr>
<tr>
<td>Log output setting</td>
<td>Settings for ECO optional drive.</td>
</tr>
</tbody>
</table>

3. Setting for ECO drive option screen

Change settings by clicking on Options in ECO Drive under Driving Sim Ribbon. The menu is displayed when ECO drive plug-in is added.
(1) Selection of result screen
Select “ECO drive analysis viewer” to display the graphs of results after logging, and select “Ranking ECO Drive viewer” to display the ECO drive diagnosis.

1) Start with drive: Begin logging when driving simulation is started, and end logging when the driving simulation ends.
2) Start with scenario: Data logging is started simultaneously with scenario, when used with the scenario function.

* Data logging can also be started with and stopped with.

3) Auto save log: Save automatically the data when the logging ends.

Parameter 1: Change the coefficient of travel time (T) in the calculation formula for carbon-dioxide emissions. Default is 0.3.
Parameter 2: Change the coefficient of travel distance (D) in the calculation formula for carbon-dioxide emissions. Default is 0.028.
Parameter 3: Change the coefficient in the calculation formula for carbon-dioxide emissions in clause 3 (clause Σ). Default is 0.056.
Parameter 4: Change carbon dioxide exhaust coefficient Kc f in the calculation formula for carbon-dioxide emissions. Default is 0.00231.

(2) Ranking ECO Drive viewer options
Change the option for the ECO drive diagnosis

<table>
<thead>
<tr>
<th>Variable identifier</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Target fuel cost</td>
<td>Optimum fuel consumption, unit in Km/L. The greater the value, the closer the result is to the optimum fuel consumption.</td>
</tr>
<tr>
<td>Amount of CO₂ ideal exhaust</td>
<td>Optimum carbon emission, unit in Kg. Penalty is added as the CO₂ emission exceeds the optimum amount.</td>
</tr>
<tr>
<td>Exhaust amount penalty coefficient</td>
<td>Coefficient for how excess CO₂ emission penalty is calculated. When this coefficient is kco2, Penalty = (CO₂ excess: kg-C) × kco2.</td>
</tr>
<tr>
<td>Dangerous acceleration</td>
<td>It becomes the subject to penalty.</td>
</tr>
<tr>
<td>Dangerous acceleration penalty</td>
<td>When the coefficient to be kacc, Penalty = (frequency that exceeds it) × kacc.</td>
</tr>
<tr>
<td>Dangerous deceleration</td>
<td>It becomes the subject to penalty.</td>
</tr>
<tr>
<td>Dangerous deceleration penalty</td>
<td>When the coefficient is kbrk, Penalty = (frequency that exceeds it) × kbrk.</td>
</tr>
<tr>
<td>Rank A acceptance line</td>
<td>Passing grade for rank A.</td>
</tr>
<tr>
<td>Rank B acceptance line</td>
<td>Passing grade for rank B (must be &lt; rank A/B Border) Anything below Rank B is ranked as C.</td>
</tr>
<tr>
<td>Standard Annual Mileage</td>
<td>Used to calculate the total fuel consumed in one year.</td>
</tr>
<tr>
<td>Currency sign</td>
<td>Currency used.</td>
</tr>
<tr>
<td>Gas price</td>
<td>Price of gasoline per liter. Use it to calculate the gasoline costs of one year.</td>
</tr>
</tbody>
</table>

Display "Print" button: Display Print button on the ranking result screen to print the ECO drive diagnosis result.
Reset button: Reset to the default values and the parameters.
OK button: Execute CO2 emission calculation by shutting the screen using the displayed parameter from the Eco drive log file. “CO2 emission calculation” button does not need to be pressed.
Cancel button: Invalidate the change, and close the screen.
4. ECO drive analysis viewer
Load the CSV output for the calculation of carbon dioxide emissions. The carbon dioxide emissions are calculated and graphed.
When the ECO drive analysis viewer is selected on the main menu, the results are displayed after the data logging.

Or click on will also display the results.

Open log file
Open the data file (*.CSV) file for the calculation of carbon dioxide emissions.
The following values are displayed.
- Log file name
- Driving time, T
- Driving distance, D
- Gasoline consumption, Q
- CO2 emission level, E
Moreover, the graph displays the data, such as acceleration, speed, gasoline consumption, total gasoline consumption, CO2 emission level and total CO2 emission level.

Change Calculation parameter
Change the parameters of the calculation formula. It will be applied to the results immediately.

Save the analytical result
Export the following items to CSV file:
- Time (s)
- Vehicle name
- Road name
- Travelled Distance (m)
- Speed (km/h)
- Steering Wheel Angle (1…1)
- Acceleration (m/s^2)
- Accelerate (0..1)
- Brake (0..1)
- X Pos
- Y Pos
- Z Pos
- Distance to the left border (m)
- Distance to the right border (m)
- Measurement point Number
- Gasoline Consumption (cc/sec)
- Carbon Dioxide Emission (kg/sec)
- Progressive Gasoline Consumption (cc)
- Progressive Carbon Dioxide Emission (kg)
5. ECO drive diagnosis result

The driving result is diagnosed based on the amount of exhaust carbon dioxide emission (fuel consumption).

After driving, the results are automatically displayed if the "Selection of the result screen" is set to "ECO drive ranking" on the ECO drive option screen.

Contents

Rank:

Shows the rank of A, B, or C, based on the scored from the diagnostic results, and displays the position in relation to the top score in percentage.

Score:

The score on gasoline consumption is the fundamental score. Penalty points on carbon dioxide, dangerous acceleration, and dangerous deceleration will be deducted from this fundamental score. The final score = 100 points - penalty points.

The scores are calculated as follows:

Total score = fuel consumption score - (CO₂ penalty + bad acceleration penalty + bad braking penalty)

Fuel consumption score = (spending on fuel/ideal fuel cost) x 100

CO₂ penalty = excess CO₂ emission (kg-C) x CO₂ emissions penalty coefficient (Pt/kg-C)

Bad acceleration penalty = frequency of excess bad acceleration x bad acceleration penalty coefficient (Pt/time)

Bad braking penalty = frequency of excess bad braking x bad braking penalty coefficient (Pt/time)

Speed: The driving speed is displayed as a graph

CO₂ emissions: CO₂ emissions during driving is displayed as a graph

Gasoline consumption: Display the consumption of gasoline converted into the mileage in a standard year by the unit of L.

Annual Gas Price: Display the total cost on fuel in a standard year based on the set gas price.

Amount of money saved: Display amount saved during year compared to when driven at the ideal fuel consumption rate.
Cluster Option Plug-in
The cluster system has been developed for use with large-scale and complex driving simulators.
Until recently, the cluster system was a system in which two or more PCs were synchronized and one of these PCs outputs the resultant image to all monitors. PCs were all dependent upon each other. For this reason, a decrease in the system's performance was inevitable when the number of channels increased.
This new system synchronizes the work of two or more PC's and outputs the resultant image to multiple monitors, thus it becomes possible to maintain a constantly high level of performance and frame rate irrespective of the number of channels since each one is displayed on an independent PC.

Starting from Ver.9 and on, the Cluster System Option now allows all Client PCs to perform walking or driving simulation as well. Previously these were limited to the Master PC.
Two different modes can be assigned to Clients PCs:

・Display Only Mode: Display only the information shown in the Master PC.
・Free Control Mode: Client PCs can freely navigate, drive/walk, drive. Master PC now also has the option to track individual client PC's camera.

To use these features, the Master PC must have the Clustering options installed and enabled, while the Client PCs require the multi-user client option.

1. Structure of cluster system
Cluster system is composed of a master machine and multiple client machines.

■Master Machine
Master machine is linked with client machine. As for master machine, perform usual calculation (movement of model, animation and camera movement etc.) of UC-win/Road and send the required data to synchronize the dynamic data (camera view, traffic model, drawing setting, setting of weather and performance etc.) of UC-win/Road project.

■Client Machine
Client machine is used only for rendering. Required data to synchronize the dynamic data of UC-win/Road project from master machine is received.

■Program
Program to use the cluster function is as follows.
Master: UCwinRoad.exe + ClusterPlugin.bpl
Client: UCwinRoad.exe, or UCwinRoadClusterClient.exe, or UCwinRoadMultiUserClient.exe + ClusterPlugin

2. Hardware Setting
Set IP address to all PCs. Those are required to be static IP address (Please ask network manager before allocating it.).
All machines of cluster machine have to be connected each other in local private network. Set the network speed to make it enough speed to avoid performance degradation
Make the speed of network card of all computers 100Mbps.
3. Cluster Setting

3-1. Cluster Master setting
Activate UC-win/Road on one machine, select Home Ribbon - Cluster - Master and switch to cluster master machine. Master setting screen is displayed on the bottom of the main screen of UC-win/Road.

[1] Actions
Synchronize the cluster machine from here.
[Search] button searches cluster client machine and display it. Only machines in client mode will be displayed.

<table>
<thead>
<tr>
<th>Load a file</th>
<th>Select a file and checked machine imports the file from where the selected file is saved. We strongly recommend using the same file where all machines are saved in the shared folder and network folder for cluster synchronization. All machines have to have access right to the selected file.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load a file from UC-win/Road save data directory</td>
<td>Select a file of save data directly of UC-win/Road. Checked machine imports a file from the individual save data directly.</td>
</tr>
<tr>
<td>Close UC-win/Road.</td>
<td>Checked machine finishes UC-win/Road.</td>
</tr>
<tr>
<td>Restart UC-win/Road.</td>
<td>Checked machine restarts UC-win/Road.</td>
</tr>
<tr>
<td>Shut down Windows.</td>
<td>Checked machine shuts Windows down.</td>
</tr>
<tr>
<td>Restart Windows</td>
<td>Checked machine restarts Windows.</td>
</tr>
<tr>
<td>Load xpswmm files</td>
<td>Load xpswmm file from a chosen save space. Since all computers are synchronized together with clustering, it is highly recommended that a shared folder or network is created beforehand. All client devices require the same access authorization to the same file.</td>
</tr>
<tr>
<td>Load xpswmm file from UC-win/Road’s save directory</td>
<td>Select and load xpswmm files from UC-win/Road’s save directory. The checked devices will each read from the data directory.</td>
</tr>
<tr>
<td>Load debris-avalanche file</td>
<td>Load debris-avalanche analysis file from a chosen save space. Since all computers are synchronized together with clustering, it is highly recommended that a shared folder or network is created beforehand. All client devices require the same access authorization to the same file.</td>
</tr>
<tr>
<td>Load debris-avalanche file from UC-win/Road’s save directory</td>
<td>Select and load debris-avalanche analysis files from UC-win/Road’s save directory. The checked devices will each read from the data directory.</td>
</tr>
</tbody>
</table>
[2] Cluster Settings

Each client and master is listed. Change of display setting of each client is possible. To synchronize between clusters, click on Start Synchronization and all computers registered on the list will be registered.

To avoid synchronization errors due to insufficient data, make sure that all connected machines have had already loaded the same exact files.

When a machine is selected on the synchronized client list, "View", "Window", "Rendering, Media" and "Virtual Display" tabs are displayed.

View

Scene type: Select a scene type in the following to be displayed on the client selected in the "Active clients".
- simulation screen
- bird view
- left mirror
- right mirror
- rear mirror
- camera view

Set parameters based on the selected "Scene type".
• simulation screens
Selection: Select the name of the simulation screen that set FOV applied to select above. Set the FOV for each screen on the Simulation Screen Editor displayed by the ribbon [View] - [FOV Options].

• bird view
Yaw, Pitch, Roll: Input each angle based on the downward direction to change the direction of view.
Height: Input the offset upward value based on the position of the camera.

• camera view
Camera view: This parameter is displayed when camera view is selected in the Scene type. When selected it, the setting of selected camera position will be reflected to the selected client.
It is possible to select a camera position saved in the project opened in the master PC. In advance, set the camera position in the Camera Positions opened from the ribbon [Home] - [Camera Position].

• Window
Displayed in the case of client. Decide screen position and screen size.
Use this client for frame delay adjustment: Check this box on the client that should be used as reference for delay adjustment.
- Rendering, Media

Rendering option
Disable HUD rendering: Use this check box to activate or deactivate all HUD rendering on the selected client.
This option does not affect the display of the Virtual Displays.
Disable scene rendering: Use this check box to activate or deactivate the rendering of the 3D scene on the selected client.
Disable vehicle mirrors: Use this check box to activate or deactivate the rendering of the mirrors of the 3D cockpit of the vehicle on the selected client.

Media
Use the checkboxes to activate or deactivate the following elements found in the Scenario:
- Display Script Messages
- Display Scenario Messages
- Display Scenario Images
- Play Scenario Videos
- Play sounds as on the master PC

- Virtual display
For each client it is possible to select virtual displays to be drawn.
Use this setting according to the Visibility Settings of the virtual display and activate the virtual display option in the visual options.
- If the virtual displays are deactivated in the visual options, no virtual display will be shown.
- The visibility of a virtual display must be set to ON to be shown.
- The virtual display of a client must be selected in the Virtual Displays tab to be shown.
- While synchronizing the cluster, no virtual display is shown on the master PC.

Check **Automatically switch the virtual displays global display option on and off when the cluster starts and stops** so the visual option virtual displays is set automatically ON when the cluster synchronization starts and OFF when the synchronization is stopped.

[3] Connection Settings
Master sends the data to all clients through multi cast protocol. The same multi cast IP and port has to be set to whole cluster computer.
*Note: Editing can only be done when disconnected. These are also saved inside “ClusterMasterSettings.ini” file, found inside UC-win/Road User Data Folder.

### Master -> Clients

**[Multicast IP Address]**
Range of multicast IP address is 224.0.0.0 - 239.255.255.255. However 239.0.0.0 - 239.255.255.255 is recommended for use in intranet.
The selected address has to be set not to crash the other multicast address on network.

**[Multicast Port]**
Port range is 1025 - 49150. When firewall is used, make sure the port is not blocked.
Set the number of multi cast port which is different from the number of master port.

**[Packet Size Limit]**
Maximum size of packet sent to clients.
The value of limited packet size must not be greater than MTU. When using the huge frame, it is possible to set to greater value up to MTU value.

**[Compress Data]**
The master sends the compress data to the clients.

**[Software Swap Buffer Sync]**
Just before called the OpenGL swap buffer in the main window, cluster master synchronizes the swap buffer call of the entire cluster.

### Clients -> Master

**[Master IP Address]**
When the several network cards stick in computer, select IP address of network card to use.

**[Master Port]**
Port range is 1025 - 49150. Please ask network manager before allocating the port.
When firewall is used, make sure the port is not blocked.
Set the number of master port so that it is different from the number of multi cast port.

**Response timeout**
During the simulation, any client not responding after the specified time will be considered as not answering and displayed with the status “Timeout”.

**[Warn on client problems]**
If one or more client is in timeout or has a low FPS a warning is displayed in the event console of the application. The threshold setting for the warning on low FPS can be setup in the following parameters.
The parameters "Warn when one client FPS is under:" and "When FPS stays low for this number of frames:" are used together to determine if a client is having FPS slowdown issues. If for the given number of frame the client refresh rate stays under the given FPS a warning will be displayed.

[Auto connect when switching to cluster mode]
When checked, the master will automatically connect to the client at the same time when it comes to cluster mode.

[Reset Settings]
Press to reset the settings.

[Connect/Disconnect] button
Press Connect to connect the master with the parameters of this screen.
Press Disconnect to disconnect the master.

3-2. Cluster client setting
Activate UC-win/Road on the other PC, select Home Ribbon - Cluster - Client display or Client free to switch to the cluster client.
Right-click on a main screen as client mode, and select pop-up menu / Cluster Settings.

■ UC-win/Road of all cluster machine must be the same version. When the all machines are connected, all clients are set to active and all PCs import UC-win/Road project. Master starts synchronization of cluster machine.
■ During synchronization, the dynamic data of whole client machine is synchronized with master and the dynamic data will be updated so that the dynamic data will be the same in cluster machine.
4. **Client (Free Mode)**
Upon successful synchronization, choosing the Client Free Mode will enable some tool bars features and information that otherwise were unavailable. Client PCs can freely select different camera modes from the ribbon, and manipulate the camera angle with keyboard and mouse, as if using UC-win/Road normally.

![Client Free Mode](image1)

5. **Tracking**
The Cluster Master can track a Cluster Client’s (who is in Client Free Mode) current view camera and the driving or navigational conditions.
From the Action tab in Cluster Master setting window, selecting “Follow a client” will display a list of tractable clients.

![Tracking](image2)

Upon selecting a client from the list, two view options become available: Track Video or Follow Model. Track View synchronizes the Master’s camera with that of the selected Client, while Follow Model tracks the model the Client is currently controlling (e.g. pedestrian, cars). If the Client is using Free Camera Mode and not controlling anything, Follow Model will only switch the Master’s Camera to the Client’s screen once, and will stop tracking further.
Tracking view will sync the view point to the same as the Client. If Tracking Model is selected, using the mouse wheel can switch to external third-person view.

To end tracking, select the click on the Free View button, or go to the ribbon Navigation - Free.
6. Adding Client Target from Scenario Events

It is possible to assign driving or walking to a specific client (who is in Client Free Mode) during a scenario event. To do this, on the Event editor, go to the User Simulation tab, and click on “Add cluster client target” to reach the following window:

Click on the button above to add a new client tab. Under this new tab, the Master can assign driving or walking simulation to a particular Client. The Machine Name should match the client’s name as was shown in Cluster Master. To modify these, refer to the previous sections on how to use the Cluster Master.

*The following scenario commands cannot be assigned to a Client:
- Resetting the traffic flow (there is no traffic flow on the Client)
- Leading car in front of the driver (The Client’s car is always deleted at the end of driving).
Virtual Display Plug-in

1. Virtual Display and Mask function

Virtual Displays can either be 2D overlays (Picture in Picture, PinP) displayed over the 3D view, or 3D objects placed within the scene. They can be used to display solid colors, images or camera views that are updated in real time. Virtual displays can also be made into any shape by applying a mask.

In the following example image shows the inclined PinP picture on the front surface of cockpit dashboard, or the virtual display whose rearview mirror is covered with mask.

Views can be combined by adding a mask. Image mask and polygon mask are supported, and in both mask modes, virtual display will be transparent in dark part of mask, and in transparent in bright part. This virtual display can be displayed as multimedia model when running scenario.

Please refer to "Overview of Virtual Display Plug-in" for details.

2. Overview

Virtual Displays have been introduced in UC-win/Road V12. Virtual Displays can either be 2D overlays displayed over the 3D view, or 3D objects placed within the scene. They can be used to display solid colors, images or camera views that are updated in real time.

In the following example the virtual display highlighted in red is an image 2D overlay (with alpha). The virtual display highlighted in blue is a camera view 3D object.
3. Operation Flow

(1) Activation
Activate "VirtualDisplaysPlugin.bpl" from the menu "File" - "License Manager".

(2) Setting of Virtual Display
Click the image button from the "Virtual Display" group on the ribbon "View" on the main screen to open the Virtual Display Editor.

(3) Display of virtual display
When "Virtual display" is checked on "Visual options" - "Display", the set virtual display is displayed.

(4) Display in scenario
It is possible to add virtual display events with scenario events. Select "Virtual Display" on the "Multimedia Model" tab of the Event Editor. Open Virtual Display Manager, which is the setting form of virtual display, from "Open file".

3. Virtual Display Editor
Once virtual displays are set, they will be displayed for all projects.

Click Virtual Display icon from the ribbon, Views – Virtual Displays to open this form.

■ Display list
The list on the left displays all the displays. A new display can be added by clicking on Add new, and deleted them by clicking on the red crosses that appear on the list. The order in which the displays are rendered can also be changed using the blue arrows.
Finally it is possible to show/hide each display by clicking on the eye icon.
The settings of the display selected in the list will be shown on the right side of the window.

■ Import/Export
The Import and Export buttons are used to load and save all display configurations from files. It is possible to choose between the Import/Export of the configuration itself or the visibility preset.
Basic settings

Name
This is the name of the display. It is not rendered in the 3D view.

Type
Defines how the display will be rendered. It is possible to choose between a 2D overlay and a 3D object.

Source
Define the content of the display. It is possible to choose between a solid color, an image or a camera view.

Source resolution
Define the quality of the display. Using a higher resolution will increase quality but will also increase the memory usage. In most cases it is best to let UC-win/Road manage the resolution to get the best quality/memory use ratio (it is possible to do so by checking the "Auto adjust" checkbox).

Mask
Masks are optional to blend the display nicely. It is possible to choose between an image based mask and a polygon mask. In either mode, the dark parts of the mask will make the display more translucent and the lighter areas will make the display more opaque (pure white areas will be fully opaque, pure black areas will be fully transparent).

Mask resolution
Define the quality of the mask. Just like with sources, using a higher resolution will increase quality but will also increase the memory usage.

2D Overlay Settings
If the display has been set to be a 2D overlay type in the basic settings, the display will be shown as an overlay on top of the 3D scene. The following options are available:

Position/size units
With this option, it is possible to choose if the placement and sizing of the overlay is relative to the main 3D view or if it is set to a defined number of pixels.

Size
The size of the overlay, either in pixels or percentage, depending on the unit set.

Position
The size of the overlay, either in pixels or percentage, depending on the unit set.
3D Object Settings
If the Type has been set to be a 3D object in the basic settings, the display will be shown as a 3D object within the 3D scene. The following options are available:

- **Size**
The size is in meters.

- **Reference CS**
The reference coordinate system used to place the display within the 3D scene. It can be set to Absolute, relative to the viewpoint (Viewpoint) or relative to the driving vehicle (Car body (only shown during drive)) in the case of Car body, when the user is not driving the placement will default to relative to the viewpoint).

- **Position**
If the Reference CS is absolute, this is the absolute position within the 3D scene. If the placement is relative, this is the offset used to place the display relative to the reference (either the viewpoint or the driving vehicle).

- **Orientation**
This is the rotation relative to the reference CS (like the position).

- **Window options**
Since the 3D object is part of the scene it will be drawn like any other object, but it is possible to choose here to ignore depth:
  - Ignore depth: Check if the display should be displayed on top of other elements of the 3D scene regardless of its distance.
  - Ignore light: Check if the display should be fully lit, regardless of the position of the sun

Color Source Settings
If the source of the display has been set to be a single color in the basic settings, the display will simply be drawn with a solid color. The following options are available:

- **Color**
This is the color to be used to draw the display. If the light is ignored on 3D objects, the color that will actually be drawn on screen will change depending on the orientation of the display and the sun.

Image Source Settings
If the source of the display has been set to be an image in the basic settings, the display will show an image. The following options are available:
Camera Source Settings
If the source of the display has been set to be the Camera view in the basic settings, the display will show the scene as seen from the set camera. The following options are available:

Camera
This is the camera from which the scene is rendered. It is possible to set a custom camera. The camera parameter of the custom camera can be modified by clicking on Edit...

Visual options
The visual options used to render the scene. These settings make it possible to disable the rendering of some elements of the scene.

While driving, draw car
Define what part of the car should be drawn while driving.

Clipping
Define the near and far clipping planes (any object outside this range will not be displayed).

Ignore stereo
During stereo rendering, the camera position is slightly modified to take into account the position of the left and right eye. If this option is Yes, the position of the eyes will be ignored (both eyes will be drawn from the same position). This option has no effect when not using stereo rendering.

Image Mask Settings
If the mask of the display has been set to be an image in the basic settings, the display will be blended based on an image. The following options are available:
Image
This is the image to be used when blending the display. Click Load... to choose the image from the hard drive.

Image Mask Settings (Polygon)
If the mask of the display has been set to be an image in the basic settings, the display will be blended based on a polygonal image. The following options are available:

Image
This is the image to be used when blending the display. This image is generated automatically from polygons. The polygons can be edited by clicking on Edit...

Polygon Mask Editor
The polygon mask editor is used to define the polygons, as well as few other options, used to generate the image that will be used as a mask to render the display. The following options are available:

Background Opacity
This defines the background of the mask, between fully transparent (0) and fully opaque.
**Edge Blending**
This defines the blending of the background on each side of the image. The Outer Opacity parameter defines the opacity at the border of the image. The left width, right width, top height and bottom height parameters define the size of the blending on each side of the image (relatively to the width/height of the image).

**Style**
This defines the blending style. It is possible to choose between typical gradient functions or a custom one. If custom gradient is selected, it is possible to edit it by clicking on "...", the gradient editor will be shown:

![Gradient Editor](image)

Using the gradient editor it is possible to define the shape of a curve from 0 to 1 that will be used to blend the edges of the polygons and of the sides of the mask. Control points can be added and removed by right clicking on the curve area. Once the curve is set, clicking on the Save button will close the curve editor and return to the polygon mask editor.

**Polygon list**
Polygons are shown in the polygon list (initially empty). New polygons can be added by clicking on New and deleted by clicking on Delete. During the creation of a new polygon, polygon points can be added by clicking on the preview area. Once the polygon is fully defined, clicking on the Save polygon button will save the polygon.
Existing polygons can be edited by selecting them in the polygon list. Once a polygon is selected, its points will be shown in the preview and can be dragged to change their positions. The polygon Operation (addition or subtraction) can be set as well as its Strength. Finally the size of the blending can be set (relatively to the mask width/height).
SimLink Linkage Plugin

1. Overview
SimLink Plugin is for Real-time Link between UC-win/Road and third party application by exchanging data. It is possible to send information on UC-win/Road about the vehicle, simulation such as other vehicles or traffic light information, or accident. And it is also possible to operate simulation on UC-win/Road by command from third party application.

During connection, it is possible to check the simulation information on operation panel of this plug-in.

Basic functions
- **UC-win/Road Plugin function**
  It contains plug-in activation/deactivation and activation by license manager.
- **Data send function**
  It sends simulation information to third party application.
- **Command receive function**
  It operates the simulation on UC-win/Road by receiving commands from third party application.
- **Operation panel**
  It edits settings for connection with another application or to check simulation information. All operations in this plug-in can be done on this panel.
- **Accident function**
  It saves automatically to Accident List as an accident.

Operation environment
UC-win/Road and third party application are connected with TCP/IP communication.

Note:
This plug-in generates automatically four files in UC-win/Road data folder¥Plugins¥RTSimLinkPlugin¥; Image.ini, LogValue.ini, Setting.ini and Text.in. When they are deleted, restart this plug-in to regenerate it. All parameters are initialized to default settings.

2. Operation flow

(1) **Activate**
Go to License Manager from UC-win/Road Menu. Check the checkbox of Real-Time Simulation link Plugin and activate the license. After activation on license manager, Real-time Simulink Link group is added to ribbon “Driving Sim”.

(2) **Prepare for connection with third party application**
Start the plug-in and third party application. Click Operator form from ribbon Driving Sim - RT Simulink.

(3) **Connect**
Click Connect on the form. During connection, it is possible to send simulation information on UC-win/Road and receive commands from third party application.

And on simulation view panel or Simulation information display panel, it is possible to edit Accident settings and to check simulation information from UC-win/Road. By click Connect button once, it switches to Disconnect.

(4) Disconnect
Click Disconnect button at the bottom.

3. Operation panel
There are 3 tabs on operation panel.

   (1) Setting forms
- Network setting tab
This is available only when network is not connected. Settings are saved to Setteings.ini file and are loaded automatically. The network settings are saved after closing UC-win/Road.

![Setting forms image]

**Simulation index**
When Command server is enable, values of **Simulation index** are sent to third party application after starting connection. The data is 1 byte and Byte type. Use this value to edit command from third party application on UC-win/Roadside.

**Export server**
This is the settings for sending information from UC-win/Road to third party application. Switch Enable/Disable, and input IP address and Port number. Input same IP address and Port as third party application. It enables the Data send function.

**Command server**
This is the settings for receiving command from third party application. Switch Enable/Disable, and input IP address and Port number. Input same IP address and Port as third party application. It enables the Command receive function.
- **Simulation setting tab**
  This tab is for settings about simulation information which is sent from UC-win/Road to third party application.

- **Visual Effects tab**
  This tab is for settings about visual effect on simulation view.

---

**Reset Simulation Time**
Enable/disable reset simulation time when starting connection with third party application.

**Radius of simulation**
Set up radius of main camera on UC-win/Road to get information from. Data out of radius is neither sent to third party application nor displayed on simulation view.

**Accident interval**
Set up interval of accidents. When multiple accidents occur within a specified time, they are treated as a one accident.

---

**Visibility Of Vehicle’s Visual Effects**: Enable/Disable vehicle’s visual effects on simulation view.

**Vehicle’s Parameter**: Set up parameters of vehicle’s visual effect.

**Line**: Set up the width and color of line.

**Line Width**
**Color**

**Information**: Set up the visual effects of text and background.

**Text Size**
**Text Color**

**Background Color**

**Background Transparency**

**Visibility Of Pedestrian’s Visual Effects**: Enable/Disable pedestrian’s visual effects on simulation view.

**Pedestrian’s Parameter**

**Traffic Light’s Visual Effects**: Enable/Disable Traffic Light’s visual effects on simulation view.

**Traffic Light’s Parameter**
(2)Simulation view panel
On simulation view panel, it is possible to operate simulation view or edit settings of Manual Accident.

![Simulation view panel](image)

Click each icon to change mode to Rotate Zoom, Move, Fly, Spin, Satellite or Follow.

Manual Accident settings
Click Manual Accident button on upper right on simulation view panel and click a point to generate Accident. This point is registered as a Manual Accident and it is processed like other accident, for example the crash accident between vehicles. It is possible to check the Accident history on simulation information tab.

1. Click Manual Accident button.
2. Click any point on simulation view.
3. A pencil icon appears at the point.
4. Click Commit button.

The pencil icon changes to an exclamation mark icon. Then it is registered as an accident.

(3)Simulation information tab
Accident History tab
It is possible to show the accident history or manual accident history.

![Simulation information tab](image)

Checkbox
Switch Show/Hide the accident. Uncheck it to hide the vehicle information when accident occurs and '!' mark for manual accident.

- **Viewpoint moving**: Click the eye mark to move the view camera to the accident point automatically.
- **Position**: Show the position of accidents at the local coordinate.
- **Date**: Show the data of accident
- **Type**: Show the accident type. It shows Auto when an accident occurs by user vehicle or auto-generated vehicle. It shows Manual when an accident occurs by manual accident on view screen.
- **Delete**: Delete an accident.
User Vehicle Information tab

- **ID:** shows vehicle model instance ID.
- **Speed:** shows driving speed.
- **Position:** shows the local coordinate. X is X coordinate and Y is Y coordinate.

Road List tab

It contains all roads in the project on UC-win/Road. ID is assigned automatically when road is generated. It is possible to export this list as CSV file using Export to CSV File.

Intersection List tab

It contains all intersections in the project on UC-win/Road. ID is assigned automatically. It is possible to export this list as CSV file using Export to CSV File.

Vehicle Performance Profile List tab

It contains all vehicles profiles in the project on UC-win/Road. ID is assigned automatically. It is possible to export this list as CSV file using Export to CSV File.
Sample
Forum8 offers a sample program with RTSimLinkPlugin. Refer to Help for details.

4. Technical note

Data send function
It is possible to change sending format of simulation information by editing LogValue.ini file. LogValue.ini file is saved to UC-win/Road data folder¥Plugin¥RTSimLinkPlugin¥, and it is loaded automatically after running UC-win/Road. So after editing LogValue.ini, it is required to restart UC-win/Road.
Refer to send data list about simulation information sent to third party application,

(1)Send data items
The following data are sent from UC-win/Road to third party application.

- **Vehicle information**
  it contains coordinate of vehicle position and parameters such as acceleration and brake. After starting simulation, it is sent to third party application every second.
- **Traffic light information**
  it contains coordinate of traffic light position and parameters such as light types. After starting simulation, it is sent to third party application every second.
- **Accident information**
  it contains coordinate of accident position and parameters such as accident type. It is sent when an accident occurs or user generates a manual accident.
- **Delete information**
  this sends object information deleted on UC-win/Road during connecting with third party application. It is used to match with data of third party application and it contains items deleted on UC-win/Road and the date. It is sent when vehicles, traffic lights or accidents are deleted on UC-win/Road.

(2)Send data editing
It is possible to change sending format of simulation information by editing LogValue.ini file.

LogValue.ini file is divided by data items, an item is called a section, which has 2 types; “Offset scaling section” and “Coordinate section”. It is possible to change send format by editing variable of section.

```
[Section name]
```
Each section is preceded by Section name which is enclosed in []. Section name has a prefix. And when it has a suffix, it is possible to identify which axis of the section is.

<table>
<thead>
<tr>
<th>Prefix</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>VI</em></td>
<td>Vehicle information</td>
</tr>
<tr>
<td><em>TLI</em></td>
<td>Traffic light information</td>
</tr>
<tr>
<td><em>AI</em></td>
<td>Accident information</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Suffix</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>.X</td>
<td>X axis</td>
</tr>
</tbody>
</table>
Offset scaling section

Define the offset and scaling for the coordinate on UC-win/Road. The suffix of the section is used to determine the axis.

Example: Offset scaling section

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OriginalValue</td>
<td>Original Value on UC-win/Road.</td>
</tr>
<tr>
<td>OriginalOffset</td>
<td>Edit onLogValue.ini file (Default is 0).</td>
</tr>
<tr>
<td>OriginalToDestinationScale</td>
<td>Edit onLogValue.ini file (Default is 0).</td>
</tr>
<tr>
<td>DestinationOffset</td>
<td>Edit onLogValue.ini file (Default is 0).</td>
</tr>
<tr>
<td>DestinationValue</td>
<td>Value sent to third party application.</td>
</tr>
</tbody>
</table>

Values are calculated as bellows.

\[
\text{Destination Value} = (\text{OriginalValue} + \text{OriginalOffset}) \times \text{OriginalToDestinationScale} + \text{DestinationOffset}
\]

Coordinate axis section

Define the order in which the coordinates on UC-win/Road are stores in a network packet. It is defined using the variable Order, and the default is X=1, Y=2, Z=3.

The entry example below shows the order in which the coordinate is stored.

- UC-win/Road X coordinate > stored in the third
- UC-win/Road Y coordinate> stored in the first
- UC-win/Road Z coordinate> stored in the second

Example: Coordinate axis section

Command receive function

It is possible to operate simulation on UC-win/Road partially by receiving commands from third party application via TCP/IP communication.

Refer to data format of receive command about data format, byte count, details of editing.ch

Emergency brake command

When receiving emergency brake command, it is possible to operate, edit the pressure of brake and enable/disable emergency brake notification. And by editing variables of Text.ini and Image.ini file, text or image can be displayed as HUD (Head Up Display) on simulation view.

Text.ini settings

On Text.ini, it is possible to set up texts when receiving emergency brake command.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[Section name]</td>
<td>Section name of Text.ini file. Ini file is divided to sections and it is possible to set up parameters. And section name should be above parameter and be different by each section</td>
</tr>
<tr>
<td>Index</td>
<td>Text index. It should be different by each section. This is correspondence to textID of emergency command.</td>
</tr>
<tr>
<td>Text</td>
<td>Texts on UC-win/Road.</td>
</tr>
<tr>
<td>DisplayTime</td>
<td>Time to display after get commands in sec.</td>
</tr>
<tr>
<td>FontName</td>
<td>Font name. It is possible to use a font installed on PC running UC-win/Road</td>
</tr>
<tr>
<td>FontBold</td>
<td>Font bold setting. True/False</td>
</tr>
<tr>
<td>FontItalic</td>
<td>Font italic setting. True/False</td>
</tr>
</tbody>
</table>
### Font size
Font size.

### Font color
- **FontColorRed**: Font color. Set up 0~255 of RGB.
- **FontColorGreen**: Font color. Set up 0~255 of RGB.
- **FontColorBlue**: Font color. Set up 0~255 of RGB.

### Text opacity
- **FontColorAlpha**: Percentage of text opacity. Transparent at 100, opaque at 0%.

### Text alignment
- **Alignment**: Text alignment.
  - _alignLeft_: Left-aligned
  - _alignCenter_: Center-aligned
  - _alignRight_: Right-aligned

### Background color
- **BackgroundColorRed**: Background color of text. Set up 0~255 of RGB.
- **BackgroundColorGreen**: Background color of text. Set up 0~255 of RGB.
- **BackgroundColorBlue**: Background color of text. Set up 0~255 of RGB.

### Background opacity
- **BackgroundAlpha**: Percentage of background opacity. Opaque at 100, transparent at 0%.

### Background size adjustment
- **BackgroundMode**: Background size adjustment.
  - _bgmText_: adjust to text size
  - _bgmDisplay_: adjust to window side
  - _bgmCustom_: adjust to the size according to CustomWidth and CustomHeight

### Background width and height
- **CustomWidth**: When BackgroundMode is _bgmCustom mode, background is adjusted to the size according to CustomWidth and CustomHeight
- **CustomHeight**: CustomHeight in pix.

### Horizontal position
- **HorizontalPosition**: Horizontal position of text.
  - _hpLeft_: Text is displayed on right side of window.
  - _hpCenter_: Text is displayed on center of window.
  - _hpRight_: Text is displayed on right side of window.

### Vertical position
- **VerticalPosition**: Vertical position of text.
  - _vpTop_: Text is displayed on upper part of window.
  - _vpCenter_: Text is displayed on center part of window.
  - _vpBottom_: Text is displayed on bottom part of window.

### Horizontal margin
- **HorizontalMargin**: Adds horizontal or vertical margin to text and background (if it is enabled.) in pix... When text position is center, these parameters are not applied.

### Vertical margin
- **VerticalMargin**: parameters are not applied.

### Text blinking effect
- **BlinkingEffect**: Text blinking effect. True/False

### Blinking interval
- **BlinkingTime**: Blinking interval in sec.

### Text movement effect
- **MovingEffect**: Text movement effect. True/False

### Moving speed
- **MovingSpeed**: When MovingEffect is enable, set up moving speed. When the positive value, text moves from right to left, when the negative, from left to right.

### 3D text effect
- **3DEffect**: 3D text effect. True/False

### 3D text depth
- **3DDepth**: 3D text depth. (Unit is decimal)

### 3D rotation angle
- **3DAngle X**: Rotation angle of 3D text in X axis in deg.
- **3DAngle Y**: Rotation angle of 3D text in Y axis in deg.

### Image.ini settings
On Image.ini, it is possible to set up images when receiving emergency brake command.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[Section name ]</td>
<td>Section name of Text.ini file. Ini file is divided to sections and it is possible to set up parameters to each section. And section name should be above parameter and be different by each section</td>
</tr>
<tr>
<td>Index</td>
<td>Text index. It should be different by each section. This is correspondence to textID of emergency command.</td>
</tr>
</tbody>
</table>
**Filename**
Filename of image. It should have an absolute path or a relative path to a folder where ini file is saved. Available file is bmp, jpg or png file. In case of png, alpha channel is used.

**DisplayTime**
Time to display after get commands in sec.

**View**
Display setting.
- _ViewMain: Display on main display.
- _View2D: Display in 2D viewpoint
- _ViewLeft: Display on left view
- _ViewRight: Display on right view
- _ViewRear: Display on rear view

**Top**
Top of image on window in percentage(0~100)

**Left**
Left end of image on window in percentage(0~100)

**Height**
Height of image on window in percentage(0~100)

**Width**
Width of image on window in percentage(0~100)

**KeepImageRatio**
Ignore height/width settings above and calculate image ratio depending to image. True/False

**Transparency**
Overall transparency. The range is 0~100. This multiplies alpha channel of png file.

### HUD command
After receiving HUD command, it is possible to generate text, image or sound set up by command on UC-win/Road. It is not the same as emergency brake command. On the third party application side, all parameters should be sent to UC-win/Road.

#### Send data list

**(1) Vehicle information**

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range/System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Object ID. It is assigned automatically.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>Time</td>
<td>Elapsed time. When simulation step is calculated, time is incremented.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Date</td>
<td>Date when TCP/IP packet was sent.</td>
<td>DateTime</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>User Vehicle</td>
<td>User’s vehicle or not</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>Simulation index</td>
<td>UC-win/Road ID. It should be assigned on operation form or Setting.ini file.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>Type</td>
<td>Vehicle type. The Profile index on data file. Index on the file list is displayed on operation form.</td>
<td>Type ID</td>
<td>0..65535</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>position X</td>
<td>Position of vehicle. It is the center of front wheel.</td>
<td>meter</td>
<td>East direction +X</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>position Y</td>
<td></td>
<td></td>
<td>Upward +Y</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>position Z</td>
<td></td>
<td></td>
<td>North direction +Z</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>Variable</td>
<td>Description</td>
<td>Type</td>
<td>Size</td>
<td>Default Value</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--------------------------------</td>
<td>------------------------------------------------------------------</td>
<td>---------</td>
<td>-------</td>
<td>---------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>latitude</td>
<td>Position of accident. Liner conversion applied to Cartesian coordinate</td>
<td>decimal degree</td>
<td>North &gt; 0</td>
<td>NO DOUBLE 8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>longitude</td>
<td></td>
<td></td>
<td>South &lt; 0</td>
<td>NO DOUBLE 8</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>East &gt; 0</td>
<td>NO DOUBLE 8</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>West &lt; 0</td>
<td>NO DOUBLE 8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yaw angle</td>
<td>Yaw angle of vehicle</td>
<td>radian</td>
<td>South direction= 0</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Counterclockwise</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pitch angle</td>
<td>Pitch angle of vehicle</td>
<td>radian</td>
<td>Horizon = 0</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Down ward = + Rotation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roll angle</td>
<td>Roll angle of vehicle</td>
<td>radian</td>
<td>Horizon = 0</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Right Inclination = + Rotation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>direction X</td>
<td>Vehicle travel direction</td>
<td>unit vector (no dimension)</td>
<td>East direction +X</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>direction Y</td>
<td></td>
<td></td>
<td>Upward +Y</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>direction Z</td>
<td></td>
<td></td>
<td>North direction +Z</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>bodyPitchAngle</td>
<td>Pitch angle ofVehicleBody (the upper part of suspension).</td>
<td>radian</td>
<td>Horizon = 0</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Downward = + Rotation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>bodyRollAngle</td>
<td>Roll angle of VehicleBody (the upper part of suspension).</td>
<td>radian</td>
<td>Horizon = 0</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Right Inclination = + Rotation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RPM</td>
<td>RPM of engine.</td>
<td>Root Per Minute (integer)</td>
<td>0..INF</td>
<td>NO WORD 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>gearNumber</td>
<td>Gear number</td>
<td>integer</td>
<td>0..Gear number (0=Neutral)</td>
<td>NO BYTE 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>speedInKmPerHour</td>
<td>Vehicle speed</td>
<td>km/h</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>speedInMetresPerSecond</td>
<td></td>
<td>m/s</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>distanceTravelled</td>
<td>Travelled distance</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>steering</td>
<td>Steering input (applicable to other vehicle in traffic flow.)</td>
<td>ratio</td>
<td>[-1..+1]</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-1: Left max.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0: Center (straight)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>+1: Right max.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>steeringVelocity</td>
<td>Steering rotation rate.</td>
<td>1/s</td>
<td>NO (Same as steering unit and range.)</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>turningCurvature</td>
<td>Steering rotation rate.</td>
<td>1/m</td>
<td>Right rotation = +</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>throttle</td>
<td>Throttle input (applicable to other vehicle in traffic flow.)</td>
<td>ratio</td>
<td>[0..+1]</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0: no throttle</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1: Full throttle</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>brake</td>
<td>Brake input (applicable to other vehicle in traffic flow.)</td>
<td>ratio</td>
<td>[0..+1]</td>
<td>YES FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0: no brake</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1: Max brake</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>lightState</td>
<td>Vehicle’s light state</td>
<td>enumeration</td>
<td>Example</td>
<td>NO BYTE 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Field</td>
<td>Description</td>
<td>Type</td>
<td>Default</td>
<td>Units</td>
<td>Notes</td>
<td></td>
</tr>
<tr>
<td>-----------------------</td>
<td>-------------------------------------------------------</td>
<td>------------</td>
<td>---------</td>
<td>-------</td>
<td>-------</td>
<td></td>
</tr>
<tr>
<td>automaticControl</td>
<td>Automatic control</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL 1</td>
<td></td>
</tr>
<tr>
<td>mass</td>
<td>Mass weight of vehicle</td>
<td>kg</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>wheelBase</td>
<td>Wheel base</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>centerOfGravityHeight</td>
<td>Height of center-of-gravity</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>centerOfGravityPosition</td>
<td>Position of center-of-gravity</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rollAxisHeight</td>
<td>Height of roll axis</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>trailerAngle</td>
<td>Yaw angle of trailer</td>
<td>rad</td>
<td>YES</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>trailerPitchAngle</td>
<td>Pitch angle of trailer</td>
<td>rad</td>
<td>YES</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>trailerWheelbase</td>
<td>Wheel base of trailer</td>
<td>m</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>inIntersection</td>
<td>In an intersection or not. When True, it is not possible to get information about traffic lane.</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL 1</td>
<td></td>
</tr>
<tr>
<td>road</td>
<td>Road name</td>
<td>Unicode string</td>
<td>May be blank.</td>
<td>NO</td>
<td>*CHAR</td>
<td></td>
</tr>
<tr>
<td>distanceAlongRoad</td>
<td>Distance from the road origin defined on VR data along the road.</td>
<td>meter</td>
<td>0..The road distance.</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
</tr>
<tr>
<td>distanceToLeftBorder</td>
<td>Shortest distance to the left border. In an intersection, this means the point obtained from vertical distance to the vehicle.</td>
<td>meter</td>
<td>On the left lane, the value is +. Out of the left lane,.</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
</tr>
<tr>
<td>distanceToRightBorder</td>
<td>Shortest distance to the right border.</td>
<td>meter</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>carriagewayWidth</td>
<td>Width of carriageway. Up to 2 carriageways can be defined.</td>
<td>meter</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>offsetFromRoadCenter</td>
<td>Offset from the center of</td>
<td>meter</td>
<td>Left : -</td>
<td>NO</td>
<td>FLOAT 4</td>
<td></td>
</tr>
<tr>
<td>Variable</td>
<td>Description</td>
<td>Type</td>
<td>Default</td>
<td>Precision</td>
<td>Unit</td>
<td></td>
</tr>
<tr>
<td>--------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------</td>
<td>---------</td>
<td>-----------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>offsetFromLaneCenter</td>
<td>Offset from the center of lane.</td>
<td>meter</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>roadLongitudinalSlope</td>
<td>Longitudinal slope of the road.</td>
<td>percent</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>roadLateralSlope</td>
<td>Lateral slope of the road.</td>
<td>percent</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>laneNumber</td>
<td>The number of the lane on which the vehicle is travelling.</td>
<td>integer</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>laneWidth</td>
<td>The width of the lane on which the vehicle is travelling.</td>
<td>meter</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>laneDirection X</td>
<td>Direction of the lane.</td>
<td>unit vector</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>laneDirection Y</td>
<td>Direction of the lane.</td>
<td>unit vector</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>laneDirection Z</td>
<td>Direction of the lane.</td>
<td>unit vector</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>laneCurvature</td>
<td>Curvature of the lane.</td>
<td>1/m</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>drivingForwards</td>
<td>There are vehicles driving forward or not.</td>
<td>boolean</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>leftLaneOverLap</td>
<td>The ratio of the width of vehicle to the width of overlap with the left lane.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>rightLaneOverLap</td>
<td>The ratio of the width of vehicle to the width of overlap with the right lane.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>collisionWithUser</td>
<td>A vehicle or pedestrian model collide with another object or not.</td>
<td>boolean</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>slipFL X</td>
<td>Slip of left front wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipFL Y</td>
<td>Slip of left front wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipFL Z</td>
<td>Slip of left front wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipFR X</td>
<td>Slip of right front wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipFR Y</td>
<td>Slip of right front wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
</tbody>
</table>
| Variable  | Description                  | Type  | Unit            | Mode | Precision | 4
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>slipFR Z</td>
<td>Z = Longitudinal acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipRL X</td>
<td>Slip of rear left wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>slipRL Y</td>
<td>X = Lateral acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipRL Z</td>
<td>Y = Vertical acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipRR X</td>
<td>Z = Longitudinal acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>slipRR Y</td>
<td>Slip of rear right wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>slipRR Z</td>
<td>X = Lateral acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>wheelSpeed FL</td>
<td>Wheel’s speed.</td>
<td>rad / s</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>wheelSpeed FR</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>wheelSpeed RL</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>wheelSpeed RR</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>localAccel X</td>
<td>VehicleAcceleration.</td>
<td>m / s2</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>localAccel Y</td>
<td>X = Lateral acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>localAccel Z</td>
<td>Y = Vertical acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>speedVect X</td>
<td>Z = Longitudinal acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>speedVect Y</td>
<td>Slip of rear right wheel.</td>
<td>double</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>speedVect Z</td>
<td>X = Lateral acceleration</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>yawAccel</td>
<td>Vehicle angular acceleration</td>
<td>rad / s2</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>pitchAccel</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>rollAccel</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>yawSpeed</td>
<td>Vehicle angular speed</td>
<td>rad / s</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>pitchSpeed</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>rollSpeed</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>bodyYawAccel</td>
<td>Body angular acceleration</td>
<td>rad / s2</td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>bodyPitchAccel</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>bodyRollAccel</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>bodyYawSpeed</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>bodyPitchSpeed</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>bodyRollSpeed</td>
<td></td>
<td>NO</td>
<td>FLOAT</td>
<td>4</td>
<td></td>
</tr>
</tbody>
</table>

*Refer to Notes about Text data sending/receiving.

(2) Traffic light information
<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range /System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Object ID. It is assigned automatically.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>Time</td>
<td>Elapsed time. When simulation step is calculated, time is incremented.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Date</td>
<td>Date when TCP/IP packet was sent</td>
<td>DateTime</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Type</td>
<td>Traffic light type (3 main lights and arrows)</td>
<td>Type ID</td>
<td>Example</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>position X</td>
<td>Traffic light position</td>
<td>meter</td>
<td>East direction +X</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>position Y</td>
<td></td>
<td></td>
<td>Upward +Y</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>position Z</td>
<td></td>
<td></td>
<td>North direction +Z</td>
<td>YES</td>
<td>FLOAT</td>
<td>4</td>
</tr>
<tr>
<td>latitude</td>
<td>Position of accident. Liner conversion is applied to Cartesian coordinate.</td>
<td>decimal degree</td>
<td>North &gt; 0</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>longitude</td>
<td></td>
<td></td>
<td>East &gt; 0</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Main light color</td>
<td>Blue/Yellow/Red</td>
<td>enumeration</td>
<td>Example</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>Straight arrow color</td>
<td>Blue/Yellow/Red</td>
<td>enumeration</td>
<td>Example</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>Left arrow color</td>
<td>Blue/Yellow/Red</td>
<td>enumeration</td>
<td>Example</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>Right arrow color</td>
<td>Blue/Yellow/Red</td>
<td>enumeration</td>
<td>Example</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>color</td>
<td>on</td>
<td>bit0 :  Red</td>
<td>bit1 :  Yellow</td>
<td>bit2 :  Blue</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>----</td>
<td>-------------</td>
<td>---------------</td>
<td>-------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Current Phase Number</td>
<td>When the traffic does not start, 0. When the first phase starts, 1.</td>
<td>number</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>Phases Cycle Offset</td>
<td>Same as intersection editor.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Current time in phase</td>
<td>Elapsed time in this phase.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Time before next phase</td>
<td>Time remaining before next phase.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Time before next color</td>
<td>Time remaining before next color.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Road ID</td>
<td>Road ID. It is assigned automatically.</td>
<td>Type ID</td>
<td>0..65535</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>Road name</td>
<td>A name of road with traffic light.</td>
<td>Unicode string</td>
<td>May be blank.</td>
<td>NO</td>
<td>*CHAR</td>
<td>*</td>
</tr>
<tr>
<td>Road direction</td>
<td>Direction of road with traffic light.</td>
<td>True :  Ahead</td>
<td>False :  Rear</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>Distance along the road</td>
<td>Distance from the road origin along the road</td>
<td>meter</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Intersection ID</td>
<td>Intersection ID.</td>
<td>Type ID</td>
<td>0..65535</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>Intersection name</td>
<td>A name of intersection with traffic light.</td>
<td>Unicode string</td>
<td>May be blank.</td>
<td>NO</td>
<td>*CHAR</td>
<td>*</td>
</tr>
</tbody>
</table>

*Refer to Notes about Text data sending/receiving.

(3) Accident information

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range/System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Object ID. It is assigned automatically</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>Time</td>
<td>Elapsed time. When simulation step is calculated, time is incremented.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Date</td>
<td>Date when TCP/IP packet was sent</td>
<td>DateTime</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Type</td>
<td>Accident type</td>
<td>Type ID</td>
<td>Example</td>
<td>bit0: Collision with another vehicle.</td>
<td>NO</td>
<td>BYTE</td>
</tr>
</tbody>
</table>
### involving pedestrian
- **bit2**: Collusion involving several entities.
- **bit3**: Manual Accident

| position X | Accident position | meter | East direction +X | YES | FLOAT | 4 |
| position Y | | | Upward +Y | YES | FLOAT | 4 |
| position Z | | | North direction +Z | YES | FLOAT | 4 |

| latitude | Position of accident. Liner conversion is applied to Cartesian coordinate. | decimal degree | North > 0 | NO | DOUBLE | 8 |
| longitude | | | South < 0 | | | |
| | | | East > 0 | NO | DOUBLE | 8 |
| | | | West < 0 | NO | FLOAT | 4 |

| Road ID | Road ID. It is assigned automatically. | Type ID | 0..65535 | NO | WORD | 2 |
| Road name | A name of road where an accident occurred. | Unicode string | May be blank. | NO | "CHAR" | |
| Road direction | A road direction where an accident occurred. | True : Front | NO | BOOL | 1 |
| | | False : Back | | | |
| Intersection ID | Intersection ID. It is assigned automatically. | Type ID | 0..65535 | NO | WORD | 2 |
| Intersection name | A name of intersection where an accident occurred. | Unicode string | May be blank. | NO | "CHAR" | |

*Refer to Notes about Text data sending/receiving.*

### (4)Delete information

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range/System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Object ID. It is assigned automatically.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>Type</td>
<td>Instance type deleted on simulation.</td>
<td>Instance Type ID</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td>Elapsed time. When simulation step is calculated, time is incremented.</td>
<td>second</td>
<td>0..INF</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
</tr>
<tr>
<td>Date</td>
<td>Date when TCP/IP packet was sent</td>
<td>DateTime</td>
<td>NO</td>
<td>DOUBLE</td>
<td>8</td>
<td></td>
</tr>
</tbody>
</table>

#### Receive command data list

### (1)Emergency command

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range/System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item</td>
<td>Description</td>
<td>Unit</td>
<td>Standard range /System</td>
<td>Standard range/Editable or not</td>
<td>Data type</td>
<td>Byte</td>
</tr>
<tr>
<td>-------------</td>
<td>---------------------------------------------------------</td>
<td>--------------</td>
<td>------------------------</td>
<td>--------------------------------</td>
<td>-----------</td>
<td>------</td>
</tr>
<tr>
<td>objectID</td>
<td>Object ID. It is assigned automatically.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>textID</td>
<td>Text ID. The text is displayed when it matches index parameter of Text.ini file.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>imageID</td>
<td>Image ID. The image is displayed when it matches index parameter of Text.ini file.</td>
<td>integer</td>
<td>0..65535</td>
<td>NO</td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>control</td>
<td>Enable Emergency brake notification</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>pressure</td>
<td>Pressure when stepping on brake.</td>
<td>integer</td>
<td>0 = Minimum</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
</tbody>
</table>

(2) HUD command

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>Unit</th>
<th>Standard range /System</th>
<th>Standard range/Editable or not</th>
<th>Data type</th>
<th>Byte</th>
</tr>
</thead>
<tbody>
<tr>
<td>text</td>
<td>Text on UC-win/Road</td>
<td>Unicode</td>
<td>May be blank.</td>
<td>NO</td>
<td>*CHAR</td>
<td>*</td>
</tr>
<tr>
<td>textDisplayTime</td>
<td>Time to displaying text.</td>
<td>ms</td>
<td>0..INF</td>
<td>NO</td>
<td>CARDINAL</td>
<td>4</td>
</tr>
<tr>
<td>textFontName</td>
<td>Font name</td>
<td>Unicode</td>
<td>It is possible to use a font installed on the PC.</td>
<td>NO</td>
<td>*CHAR</td>
<td>*</td>
</tr>
<tr>
<td>textFontBold</td>
<td>Font bold</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>textFontItalic</td>
<td>Font italic</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>textFontSize</td>
<td>Font size</td>
<td>integer</td>
<td>1..72</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textFontColorRed</td>
<td>Text color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textFontColorGreen</td>
<td>Text color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textFontColorBlue</td>
<td>Text color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textFontColorAlpha</td>
<td>Text color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textAlignment</td>
<td>Text alignment</td>
<td>text alignment type</td>
<td>Text alignment</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textEnableBackground</td>
<td>Enable text background.</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
<td>1</td>
</tr>
<tr>
<td>textBackgroundColorRed</td>
<td>Text’s background color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textBackgroundColorGreen</td>
<td>Text’s background color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textBackgroundColorBlue</td>
<td>Text’s background color and transparency.</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
</tbody>
</table>

-349-
<table>
<thead>
<tr>
<th>textBackgroundMode</th>
<th>Text background size</th>
<th>Background mode</th>
<th>NO</th>
<th>BYTE</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>textCustomWidth</td>
<td>Text with custom size</td>
<td>pixels</td>
<td>0.65535</td>
<td>NO</td>
<td>WORD</td>
</tr>
<tr>
<td>textCustomHeight</td>
<td></td>
<td></td>
<td></td>
<td>WORD</td>
<td>2</td>
</tr>
<tr>
<td>textHorizontalPosition</td>
<td>Text position</td>
<td>Horizontal position</td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textVerticalPosition</td>
<td>Vertical position</td>
<td></td>
<td>NO</td>
<td>BYTE</td>
<td>1</td>
</tr>
<tr>
<td>textHorizontalMargin</td>
<td>Horizontal or Vertical margins</td>
<td>pixels</td>
<td>When textHorizontalPosition is center, this is not applied.</td>
<td>NO</td>
<td>WORD</td>
</tr>
<tr>
<td>textVerticalMargin</td>
<td></td>
<td></td>
<td>When textVerticalPosition is center, this is not applied.</td>
<td>NO</td>
<td>WORD</td>
</tr>
<tr>
<td>textBlinkingEffect</td>
<td>Text blinking effect</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
</tr>
<tr>
<td>textBlinkingTime</td>
<td>Blinking interval</td>
<td>s</td>
<td>0.1..5.0</td>
<td>NO</td>
<td>FLOAT</td>
</tr>
<tr>
<td>textMovingEffect</td>
<td>Moving effect</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
</tr>
<tr>
<td>textMovingSpeed</td>
<td>Moving effect speed</td>
<td>s / screen</td>
<td>-100..+100</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>If &gt; 0:  Right to left</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>If 0:  without movement</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>if &lt; 0:  Left to right</td>
<td></td>
<td></td>
</tr>
<tr>
<td>text3DEffect</td>
<td>3D text effect</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
</tr>
<tr>
<td>text3DDepth</td>
<td>3D text depth</td>
<td>decimal</td>
<td>0.1..1</td>
<td>NO</td>
<td>FLOAT</td>
</tr>
<tr>
<td>text3DAngle X</td>
<td>3D text rotation</td>
<td>deg</td>
<td>-45..45</td>
<td>NO</td>
<td>WORD</td>
</tr>
<tr>
<td>text3DAngle Y</td>
<td>X axis or Y axis</td>
<td></td>
<td></td>
<td>NO</td>
<td>WORD</td>
</tr>
<tr>
<td>soundFilename</td>
<td>Sound file name</td>
<td>Unicode string</td>
<td>Assign the relative path from UC-win/Road data folder. When the folder name is blank, it may not work correctly.</td>
<td>NO</td>
<td>*CHAR</td>
</tr>
<tr>
<td>soundPlayTime</td>
<td>Time to play sound</td>
<td>ms</td>
<td>0..INF</td>
<td>NO</td>
<td>CARDINAL</td>
</tr>
<tr>
<td>soundVolume</td>
<td>Sound volume</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>BYTE</td>
</tr>
<tr>
<td>soundLoop</td>
<td>Sound loop effect</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>BOOL</td>
</tr>
<tr>
<td>imageFilename</td>
<td>File name of image</td>
<td>Unicode string</td>
<td>Assign the full path of image file. When the folder name is blank, it may not work correctly.</td>
<td>NO</td>
<td>*CHAR</td>
</tr>
<tr>
<td>imageDisplayTime</td>
<td>Time to display image</td>
<td>ms</td>
<td>0..INF</td>
<td>NO</td>
<td>CARDINAL</td>
</tr>
<tr>
<td>imageView</td>
<td>View type where to display</td>
<td>View type</td>
<td>Assign a display.</td>
<td>NO</td>
<td>BYTE</td>
</tr>
<tr>
<td>Variable</td>
<td>Description</td>
<td>Type</td>
<td>Default</td>
<td>Access</td>
<td>Size</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------------------------</td>
<td>--------</td>
<td>---------</td>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>imageTop</td>
<td>Top of image</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>imageLeft</td>
<td>Left side of image</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>imageHeight</td>
<td>Height of image</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>imageWidth</td>
<td>Width of image</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>imageKeepImageRatio</td>
<td>Ignore height/width settings above and calculate image ratio depending to image.</td>
<td>boolean</td>
<td>True or False</td>
<td>NO</td>
<td>1</td>
</tr>
<tr>
<td>imageTransparency</td>
<td>Transparency of image</td>
<td>byte</td>
<td>0..255</td>
<td>NO</td>
<td>1</td>
</tr>
</tbody>
</table>

*Refer to Notes about Text data sending/receiving.*

**Notes about text data sending/receiving**

When sending/receiving text data, it should be stored in network packet of TCP/IP communication.

<table>
<thead>
<tr>
<th>Data type</th>
<th>Format</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Byte</td>
<td>L</td>
<td>N+1 (Byte type), N: the number of letters in text (WideChar type), #0: Null (WideChar type)</td>
</tr>
</tbody>
</table>

(Example) L: N+1 (Byte type), N: the number of letters in text (WideChar type), #0: Null (WideChar type)
Gaze Tracking Plug-in

This plugin is to connect UC-win/Road and any eye tracking device. The data is converted to the data format for UC-win/Road.

Hint: This plugin can be linked with Object Detection Plug-in. It is possible to detect the tracked objects.

Connect with eye tracking device

(1) Driver

In most cases, a driver suitable for the type of eye tracking device is required to be developed to connect with UC-win/Road and convert the tracking data to the data format used in UC-win/Road.

(2) Communication format

Refer to the chapter Communication format.

(3) Receiving gaze tracking information

The tracking information is transferred to UC-win/Road via UDP. Click Start Gaze Tracking to start receiving the data based on the setting of Host IP and Port. For details, refer to Settings.

Use mouse point

It is possible to use mouse point to simulation the eye tracking. The points pointed by a mouse pointer are treated as the points gazed toward. Refer to Eye tracking simulation with mouse point.

Ribbon menu

After enabling Gaze Tracking Plug-in, the gaze tracking menus will be added to the Device ribbon.
Host IP: 192.168.1.80  
Port: 3000

**Host IP, Port:** Set up the host IP address and port number.

**Use Mouse Point:** Use mouse point to adjust the eye tracking data.

**Options:** Open the setting option form.

### Receiving gaze tracking information

The gaze tracking information is transmitted to UC-win/Road via UDP. Click Start Gaze Tracking to start receiving the information. And UC-win/Road displays it on the screen depending on the settings.

![Gaze tracking information on UC-win/Road](image)

**Hint:** Once the icon Start Gaze Tracking is clicked, it is changed to Stop Gaze Tracking, and then it is not possible to change the host IP and port number.

### Settings

1) **Host IP**
   - Choose a Host IP listed on the combo box. It is possible to assign a loop back address manually too.

2) **Port**
   - Assign a port to receive the tracking information. When the port is already used, an error message will be displayed. 4000 is assigned as default. The input range is from 1024 to 49151.
Eye tracking simulation with mouse point

Click Use Mouse Point to start eye tracking simulation with mouse pointer. The points pointed by a mouse pointer are treated as the points gazed toward.
However, it is not possible to use in the following cases
- While the actual eye tracking information is being receiving.
- When the camera movement is not “Rotate”.

Gaze tracking options

Here adjust the gaze tracking information position and set up marker settings.

Go to the ribbon Server – Gaze tracking, and click Options.

General Adjustment
It is possible to adjust the received eye tracking information using offsets or scaling.
Measurement Reliability
It is a reliability of the received tracing information. It depends on eye tracking devices. So that set up maximum reliability on UC-win/Road side to normalize the received information within the range 0-1.
If the reliability is less, some processes will be skipped. For example, in the case that the reliability is 0, it is not possible to use eye tracking information in Object detection plugin. How to treat this reliability depends on the connected plugin.

Received tracking information
It shows the received tracking information in real time.

Marker
It is possible to display the actual eye sight with marker. Set up the color, size and type.

- **Type**: Choose one from 2D Circle, 3D Circle, 2D Cross or 3D Cross. Check Fill to fill the circle with the marker color.
- **Size**: Set up the size of marker.
- **Color**: Choose a color of maker.
Communication format

EyePositionX, EyePositionY, EyePositionZ
There are information of eye position in X, Y, Z.

EyeDirectionX, EyeDirectionY, EyeDirectionZ
These are information about direction of eye sight line.

HeadPositionX, HeadPositionY, HeadPositionZ
These are information about the head position.

HeadDirectionX, HeadDirectionY, HeadDirectionZ
These are information about the head direction.

MeasurementReliability
This is a value of measurement reliability. This must be more than 0.
Object Detection Plug-in

It is possible to acquire and utilize the object information detected in UC-win/Road. The apex angle and distance defines a cone-shaped detection range as an object sensor, and the sensor detects objects in the range.

The detection results can be used in the simulation real time function. The simulation information including the object detection result can be transferred to the cooperated application via TCP/IP in real time and the application sends vehicle control commands based on that information to execute simulations of in-vehicle devices combining UC-win/Road and external applications.

How to use
Go to License Manager, and enable Object Detection Plug-in

Menus
Once the plugin is enabled, the menus will be added to the ribbon menu.

Start/Stop Detection: Click this icon to start object detection.

Object Sensor: Open object sensor setting form. Refer to Object Sensor settings.

Cockpit Object: Opens cockpit object setting form. Refer to Cockpit Object Settings.

Options: Open Option form. Refer to Object Detection Options.

Start detection
Start object detection. At first, click Object Sensor, Cockpit Object and Options to set up each setting.

Object sensor settings
Object sensor means a cone shaped detection range. It is possible to register object sensors as needed.
Object Sensor Settings
Set up a size of cone shaped detection range. Each object sensor has each ID.

- Add: Add a new object sensor.
- Delete: Delete the selected object sensor
- Enable Sensor: Turn on/off the sensor.
- Sensor ID: It is a unique ID for each sensor.
- Detection distance: Set up the detection distance in meters.
- Apex angle: Set up the apex angle.
- Color: Set up a color to display the detected objects.

Sensor type
Set up the position and direction of the sensor. Choose one from the following 3 types.

- Relative to main camera: Sensor moves relative to the main camera of UC-win/Road.
- Relative to vehicle chassis: Sensor moves relative to the direction of vehicle. It can be enable only during driving.

Hint: Furthermore, it is possible to adjust the position and direction of sensor with offsets. This is a sensor mounted on the vehicle, the position of sensor must be within ±20m.

- Relative to eye position: This is a function connected with Gaze Tracking Plugin. Sensor moves relative to the eye positions received from an eye tracking device. Without the eye tracking data, sensor moves relative to the main camera.
Cockpit Object settings
It is possible to register cockpit objects as detection range. For example, without cockpit object settings, objects off side of the mirror, which it cannot be seen actually, are detected if the driver gaze into a mirror. By setting the mirror as a cockpit object, if the driver gaze into a mirror, the mirror can be detected.

Click Cockpit Object.

Setting items
- Add: Add a new cockpit object.
- Delete: Delete the selected object.
- ID: Assign the ID number to cockpit. It is not a unique number, so it is possible to assign same number to the cockpits.
- Type: Select a cockpit type to be detected. Now only Actual Screen is available.
- Screen: Select a screen to be linked with cockpit object.
- Client IP: Input a client IP which displays a simulation screen to be detected.
- The screens on which cockpit objects can be detected screens other than MAINVIEW.
- Gradation: set up the range of gradation display when cockpit objects are detected. Input the percentage in vertical and horizontal direction.
Object Detection Options

It is possible to select objects to be detected. Click Options icon.

Category

It is possible to filter the objects, such as vehicles, characters, other movable models, signals, 3D models, signs, markings and cockpit objects, to be detected. Check it on/off to turn the detection on/off.

The settings are applied automatically.

Static object

Set up this options to detect 3D models arranged on the project. It is possible to specify the target 3D models by Custom ID. Disable the object type to be detected on Category tab, and then register the Custom ID for the target object type on Static Object tab, so that it is possible to detect only the object with same custom ID.

Dynamic object

Set up this options to detect 3D models arranged on the project. It is possible to specify the target 3D models by Custom ID. Disable the object type to be detected on Category tab, and then register the Custom ID for the target object type on Dynamic Object tab, so that it is possible to detect only the object with same custom ID.

It is recommended to assign as big custom ID number as possible.

Note: Reuse the Custom IDs
The unique IDs are automatically assigned to the dynamic objects. But a model with Custom ID has deleted from the project, the Custom ID becomes a unique ID. If the unique ID is assigned to a new model, it may be a target for detection.

Display
Able or disable the display of object sensor, bounding box and screen respectively.
Hint: If the displays are turned off, the detected object information contains the results. And the display settings are applied for the next time automatically because it is saved to the registry.
Camera Sensor Base Plug-in

1. Overview
Camera Sensor Basic plug-in provides the function of simulating image distortion by fisheye lens etc. The lens distortion parameters support basic projection methods such as equidistant projection, isometric projection, stereographic projection, and orthographic projection besides polynomials, parameter specifications in distortion table files. These lens simulations can be applied to any camera view as well as the main screen.

2. From Main view
Right click on the main screen and select Renderer Settings. Select Camera Sensor Renderer and click Apply. The renderer setting form is opened. Edit the parameters and click OK.

3. From Camera view
Open the camera view editor and select Camera Sensor Renderer as Camera Renderer. Edit the parameters on the right side and then click OK. The camera view changes according to it.
4. Setting items

■ Cube map option

Texture size
Select texture size when generating the cube map. The size is specified size in +X, -X, +Y, -Y, +Z, and -Z directions respectively. Select a power of 2 from 128 to 4096. Depending on the performance of the video card, images may not be generated.

■ Camera lens function

Lens type
Select a lens type from the followings.
- Polynomial (Enter the constant term and the 6 coefficients.)
- Distortion table file (Click Open to load a distortion table file.)
- Equidistant projection
- Equal solid angle projection
- 3D projection
- Orthographic

■ Camera parameters

Set up the parameters of the camera.

Pixels: Pixel size of horizontal and vertical.
Center: Pixel size of the center position.
Cell size: Size of 1 pixel.
Focal length: Focal length of the lens. This is not available when "Lens type" is Polynomial or Distortion table file
Cutoff angle: Angle at which drawing can be overlooked. If selecting Distortion table file, this is not available. And the maximum incident angle of view in the distortion table is the cutoff angle.
DS Course Converter plug-in

1. Overview
The DS course conversion plug-in exports data of roads or lanes. Center coordinates, width, curvature radius etc. can be exported as a CSV format at a fixed interval or at each curvature variation. Furthermore, the coordinate system ("Local", "XY", "TKY", and "DD") is exported so that the coordinates of roads can be read in different software. The graph of curvature change can be built for road data exports with a fixed interval.

2. Operation flow
1. Activation
Go to "File" – "License Manager" and activate the plug-in.

2. Road selection
Load a project. Then go to File – Export – Export DS Courses to select roads to be exported.

3. Export data settings
Click "Settings..." to configure data export.

4. Data exporting
Click Export button in the road selecting form to export the data.
5. Data format
If "Road" is selected as the Export Target, one csv file is created per road, and if "Lanes" is selected, a csv file is created per lane. Each item in a csv file is separated by comma (,). The first row shows a name of item, and data for corresponding distance is displayed from the second row. For position items ("Center position", "Left position", and "Right position"), each X, Y and altitude value are displayed with the coordinate system in parenthesis. For example, if "Center position" is exported in "Local" coordinate system, three columns are created: of Center X (Local), Center Y (Local), and CenterHeight.

Length, CenterX (Local), CenterY (Local), CenterHeight, Alignment, Structure type
0, 500, 1000, 610,..., Begin, Normal,
100, 600, 1000, 610,..., Straight, Normal,
200, 700, 1000, 615,..., Curve, Normal,
..., ..., ..., ..., ..., ...
1200, 800, 600, 612,..., End, Normal,

Selected items are exported by columns from left to right.

6. Exceptions
Depending on the lane file, there are items where data is not exported for the number of lane changed. For example, there is a case that a road exists, whose lane number changes from 2 to 1 at a distance of 100m. At that point, because the lane 2 disappears, the data for lane 2 is not exported after 100m (the distance will be done.).

7. Coordinate position
When the target is a road, the left side of the road direction is the left carriageway and the right side is the right carriageway. If the target is in each lane, the lane of the left carriageway depends on the road direction, but the lane of the right carriageway depends on the opposite of the road direction. The coordinates of the left edge and the right edge for each lane are as shown in the figure below, respectively.

8. Coordinates of roads and lanes
The lane coordinates is exported based on the lane center of UC-win/Road. When exporting with the special VR data such as data created by roads that overlap with invisible roads, the coordinate different from appearance may be exported. To get the
appearance coordinate, manually edit the exported coordinate data. The figure below shows an example of exporting the
invisible lane coordinates of Highway.rd bundled with UC-win/Road (White sphere objects are the position of export.).

9. About Transition section
The width of each lane will be a constant value in the section where the section changes. Because "Lane left edge" and "Lane
right edge" are calculated using the "Lane width", it is not guaranteed that those value are accurate.
Speed meter display plug-in
This is a function to display a meter panel, which shows the speed of driving vehicle, the rotating speed of engine, the direction indicator and so on.

It is displayed on a separate form, different from the Main Form. In driving mode, the components react to the driving situation: warning lights, speedometer, tachometer etc.

Eight instruments are shown:
1. Background image
2. Speedometer
3. Tachometer
4. Turn signals (hazard warning)
5. Gearstick position indicator
6. Hand brake warning
7. Full beam indicator
8. Wiper indicator
9. Seat belt warning

It is possible to customize the dashboard by changing image, position and each size:

About input device
The instruments react according to the input device. Available input devices depend on the status of the drive simulator plug-in.

- DS plug-in is used: input data from DS is reflected on the meter panel.
- DS plug-in is not used: Input data from keyboard or game controller is reflected on the meter panel.

Hint: When using keyboard or game controller as input device, seatbelt and hand brake warning are always off.
Simulink Connection Plug-in

1. Overview

Simulink connection plug-in is to operate the own car driving by using the vehicle model external to UC-win/Road. The external vehicle model and Simulink connection plugin are linked via UDP communication. Normally the external vehicle model needs to be prepared, however we provide a sample here, which contains a connection block that is always used in Simulink and also a simple vehicle model. It is possible to check the operation immediately. And if you follow the communication protocol, it is not necessary to use Simulink to realize the external vehicle model.

Basic Function

- It is possible to select between the UC-win/Road's vehicle model and the external vehicle model.
- It is possible to define IP address and port (excluding transmission port) used for UDP communication.
- It is possible to define the communication interval time on transmission.
- Transmission and reception are done asynchronously.
- It is not guaranteed that all received packets will be processed.
- Logs of communication data can be saved in CSV file.

2. Environment

Since the external vehicle model and UC-win/Road are linked via UDP communication, there is no restrictions on the execution environment for the external vehicle model as long as it follows the communication specifications. Besides external hardware and PC, it can also be executed on the local PC. The environment required for the operation of the external vehicle model depends on the model. The MATLAB / Simulink license required to run the sample model is as follows.

- MATLAB
- Simulink
- Simulink Desktop Real-Time

It is recommended to use the following licenses as options.

- Simulink Coder (Required for operation in external mode, not required in normal mode.)
- MATLAB Coder (It is an operation requirement of Simulink Coder, unnecessary in normal mode.)

The sample model is confirmed with MATLAB / Simulink version R2016b, R2017a. MATLAB / Simulink can operate on the same PC which runs UC-win/Road or on another PC. When operating on the same PC, use the local IP address for communication and set port numbers that do not overlap. If more than one network can be used, please define a physical IP address instead of local IP address 127.0.0.1.

When operating on a different PC, Mac OS X is also possible in addition to Windows OS. Linux does not support Simulink Desktop Real-Time, so it is not possible to use the sample model.
3. Operation flow

(1) Prepare the execution environment
With MATLAB / Simulink environment, it is possible to use the sample model to check the operation of the plugin.

(2) Connect the external vehicle model with UC-win/Road.
When operating the external vehicle model on the PC which does not operate UC-win/Road, connect the devices by LANs. The sample model works with MATLAB / Simulink environment on the same PC or another PC.

(3) Activate Simulink connection plugin on UC-win/Road.

(4) Check "Auto connect when driving" on the option screen of Simulink connection plugin.
The Options screen is opened by the ribbon, Driving Sim -> Simulink – Options.

(5) Similarly set parameters such as logs and vehicle specifications as needed in Option screen.

(6) Communication settings in UC-win/Road
Set the "Listen on IP" and "Send to IP" in the Option screen of UC-win/Road.
For the "Listen on IP", select it from the drop down list.
When executing the external vehicle model on the same PC with UC-win/Road, both of the transmission and reception IP address can be defined to be the same one. When executing on an external device, set the IP address of the external device as the transmission IP address.

(7) Communication settings on the side of the external vehicle model
Define it to be the IP address of PC which operates the UC-win/Road on the side of the external vehicle model.
When using the sample model, set the transmission and reception IP address according to the “Communication module settings”. The IP address on the side of external vehicle model depends on the model configuration, however it refers to the PC settings when using PC.

(8) Start running the external vehicle model.

(9) Start driving on UC-win/Road.

(10) Stop driving on UC-win/Road, then stop executing the external exercise model.
BIM/CIM
**DWG Tool**
The DWG tool plugin enables to import and load DWG format CAD data into UC-win/Road and to export UC-win/Road projects into DWG format. It is possible to import 3D Model and Road Cross-section and export road and terrain data. It needs DWGToolPlugin and ExportScenePlugin.

1. Exporting from UC-win/Road to DWG
1-1. Go to File - Export - Export DWG.
1-2. Enter the criteria the DWG Export window. (See Below)
1-3. Click on the Export button and specify a destination folder.
1-4. Export starts.
1-5. A message “Finished exporting the scene” appears. Export is completed.

**DWG Export setting window.**

- **Output Unit**
  - Each objects: Each object is output.
  - Whole objects: Selected objects are output together.
  - Each kinds: Each kind, such as road, intersection, model and terrain is output.

- **Color setting**
  - White: All objects are output in white.
  - By layer: it is possible to change the colors.
    - The road layer : red
    - The intersection layer : yellow
    - The model layer : green
    - The terrain layer : blue
  - Use UC-win/Road material: Use the colors based on UC-win/Road materials.

- **Filter option**
  Check objects to export.

- **Coordinate system**
  - World coordinate
  - UC-win/Road local coordinate
  - Objects coordinate
  This is available only when “Each objects” is selected as an output unit. The coordinate system for each model.

*Moreover, check “Use offset” check box to give the offset to the coordinate.

  - East - West
    Enter the offset of east-west direction in m. Enter a positive value to move a model to east, a negative value to west.
  - North - South
    Enter the offset of south - north in m. Enter a positive value to move a model to north, a negative value to south.
  - Height
    Enter the offset of the vertical direction in m. Enter a positive value to move a model up, a negative value down.
2. Importing from DWG to UC-win/Road

There are 2 methods (*all data is in meters).

(A) Transforming into 3D objects

Import DWG data as a 3D model data which is blocked by 3DFACE and is output to wblock export command line. Only one color is supported to each surface. Textures are not supported.

(B) Transforming into 2D Images

Import DWG data as a cross-sectional for road which is edited with Lines (excluding polylines) regardless of whether it is a closed shape. Do not include any shapes other than the horizontal cross-sectional outline.

2-1. Go to File - Import DWG.
2-2. Select the DWG file to be imported.
2-3. Set the necessary properties at the setting window.

A. Importing as a 3D Object

A-1. Enter the name displayed in UC-win/Road.
A-2. Select a Model Group Type and click OK.
A-3. Find a model from the Model Panel

*Right click any position to open the pop-up menu to sort models. The imported model can be found easily by sorting by "modification date".

Left click on the imported model to place it on a main screen.

B. Importing as a 2D Shape

B-1. Enter the cross-sectional name displayed in UC-win/Road.
B-2. Data type is always general cross-sectional parts.
B-3. Select a section type from road, railway bed or railway tracks.
B-4. Select whether this is a road, bridge, or tunnel. Although a file contains all 3, it still necessary to specify only one.
B-5. Select a layer of DWG file.
B-6. Click OK. The Section List will open.
B-7. Display and edit it as necessary.
Import / Export LandXML

1. Preparation
Go to “File” - “Import” - “Import LandXML” - “Options”.

Setting to change from LandXML to UC-win/Road

**Options for LandXML**

Convert surface as:
- Terrain Patch
- Convert surface as: convert it as model.

Smart Transitions
Generate transition depending on the number of the panel points.
Uncheck to generate transition without revising the number of points.

Default length for transitions
If it is set at 0, the transition section starts at 1cm behind the section.

Maximum number of structures in a section
Specify the number of elements to be included in the structures. With a positive value, an element from the internal structure is included. With a negative value, an external element is deleted.

If "2" is entered, element "4" and "3" are included in the cross section. If "-2" is entered, element "1" and "2" are deleted.

Default length of intersections legs
The size of imported intersections. (Input range: 6m - 500m)
Note: When LandXML has no intersection information, UC-win/Road uses this value to create intersections.

Use the section defined in this Options
Check to use the section defined here when there is no road sections in the LandXML. Unchecked to use the default one.

Standard section option
Edit the parameters such as texture, size and angle etc. of default shape, carriageway marking and cut soil/banking.

Note: When LandXML has no intersection information, UC-win/Road uses this value to create intersections.
Standard Section Option

Changing setting from UC-win/Road to LandXML

Margin for terrain export: output range from the road
Correct sections before export: Check to delete unnecessary elements prior to export. Unchecked to export without correction.

[Manage section attributes ...] button: Open "Cross sections attribute management" form. It is possible to set the cross section's attribute for LandXML's import and export.

Settings for Cross section's attribute
"LandXML to UC-win/Road" tab

Edit the following parameters.
Color, texture, and size
- YTI: LandXML cross section which is used for HICAD (Yokokawa Gijutu Joho)
- MTC: LandXML cross section which is used for APS-Mark IVWin (MTC)
The defined cross section's attribute data is saved to ¥Data¥XMLSurfaces.cfg
Copy it when importing LandXML to another PC. It can be added, copied, and deleted by buttons on right side.

Select the attribute of UC-win/Road and set up the texture and size. Then Add the attribute element by clicking "Add" button.
"UC-win/Road to LandXML" tab
Set Name/Description of the attribute of UC-win/Road for the cross section's attribute of LandXML

2. LandXML Import

1. Go to "File" - "Import" - "Import LandXML".
2. Click [Import as new] and select a file.
3. Set up import options.
   Specify the coordinate system when there is no information of the coordinate number (1-19) in the LandXML file.
4. Click [Merge].
   Click [Import as new] to generate data newly. Check "Select elements" option to select elements to import.
5. Click [Done].
3. Exporting LandXML

Go to [File] Ribbon - [Export] - [Export LandXML] and specify the file name. When checking "Correct sections before exporting" option, select a section to export, and correct the shape on "Correct cross sections" form.

Select or auto adjust a section for export.
Checking Auto sort allows for self-adjustment.
Uncheck Auto sort to select sections to be exported.

How to remove unnecessary sections
The red dotted line is the cut line, which can be moved by dragging. Then the section which crosses a line is cut.
Checking "Cut at intersection points" will cut line at the intersection point. This can be applied for other cross sections by right mouse-clicking.

Refer to the following Help for the details for changing LandXML.
・Operation - Option - "LandXML options" screen (plug in)
・Beginner's guide (to do -?) - LandXML data conversion
・Technical note - About LandXML data conversion
IFC Plug-in

1. Import IFC file

The geographical features defined by IfcSite in IFC format is imported in UC-win/Road. When importing, the geographical features data is converted into the geographical features patch data of UC-win/Road.

Go to "File" - "Import" - "Import IFC file".

- Load IFC file

Click Open icon, select the IFC file (*.ifc) to be loaded, and Click Open in the open dialog. IfcSite entity will be displayed in the box if it is found in the IFC file.

- IfcSite entities

The IfcSite entity information is displayed. Select the entities to convert into a UC-win/Road terrain patch. The regions of the selected ones will be indicated in the plan view on the right side in red boxes. The entities not selected will be indicated in blue.

- Latitude and longitude

Select the latitude and longitude. It is possible only when "New project, create mesh terrain from DTM data" is selected from the Import Methods. Click the button to reset the IFC file to the initial state.

- Import method

The following import methods are available.

○ New project, create blank mesh terrain at default height

Create a new project and generate a mesh terrain where the height of each mesh is the initial height set in the options below. The center of the entire terrain of the IFC file corresponds to the center of the mesh terrain.

○ New project, create mesh terrain from DTM data.

Create a new project and generate a mesh terrain using the height data found in the conditions set in the options below. UC-win/Road terrain patch is placed on the mesh terrain in the position that is calculated from the latitude and longitude information.
in the IfcSite data.

○ Merge with existing project
The IfcSite data is converted and placed on the mesh terrain at the position that was defined in the Option below.

- **Merge method for overlapping terrain patches**
  Terrain patches cannot be placed on top the existing terrain patch, therefore, select the method to overlap terrain patches.

○ Do not merge overlapping terrain patches
The overlapping terrains are merged as a new single terrain.
○ New terrain patches have priority when overlapping
The existing terrain is replaced by the newly imported terrain where there is an overlap.
○ Old created patches have priority when overlapping
The existing terrain is utilized instead of the newly imported terrain, therefore, the overlapping regions of the newly imported terrain are deleted.

Option
(1) Click "New project and create blank mesh terrain at default height" in the import method above.

- **Blank terrain settings**  Default height
Define the vertex height of the blank terrain mesh. When the IFC file is loaded, the default height of all IfcSite data is converted to the smallest possible value. Click the back button to reset it.

- **Select location**
Select the location information of the project.

(2) Select Import method "New project and create mesh terrain from DTM data".

- **Latitude and longitude**
The smallest the latitude and longitude values in all IfcSite entities are displayed. Click to reset it to the smallest values.
  *Input degree, minute and then second in fraction.

- **Select country**
Select the country of the terrain file. Copy the terrain file to the folder C:\UCwinRoadData XX.X\Data
- Select location
Select a location for the project.

- Terrain Type
Select from the following terrain mesh. 10km × 10km □ 20km × 10km □ 10km × 20km □ 20km × 20km

(3) Merge with existing project

![Option]

Placement
Set the location of geographical features patch generated by IfcSite data. Local coordinate system of UC-win/Road is used.

- Elevation offset
Set the offset value in the direction of the altitude when the IfcSite data is converted into geographical features.

Click "OK" to start importing. After import is completed, "Finished importing" message appears.

2 Export IFC file
Geographical features and the arranged model can be output from UC-win/Road as IFC data. It can be imported by the application that supports other IFC.
Export IFC file from File - "Export" - "Export IFC file".

Set items before exporting, on the export settings window opened by above menu.
Check Export Terrain Data and select model to export.
Click OK to start exporting. Click Cancel to close the window without exporting.

Click "OK" to start exporting, and specify the file name of the destination file.
Soil calculation
This function calculates the amount of cutting and banking from the difference of the shape before and after construction.

Soil calculation of road earthwork
On the soil calculation of road earthwork, the amount of soil is calculated from the difference between the surface of the road earthwork and the surface of the terrain. First create the shape of the road earthwork using the soil section set in the road section. After specifying sections for the calculation case, the shape of soil section is created only at the sections.

Next, delete the area that does not overlap when looking at the earthwork surface of road and the terrain surface from above.

Then, compare the earthwork surface of road and the terrain surface, and divide it into the area where the earthwork surface of road is over than the terrain surface or under. The former is judged as banking and the latter is judged as cutting.

After that, considering the prismatic column from the surface to the lowest height for each of earthwork surface of the road and the terrain surface, the volume of the prismatic column is calculated.
Calculation flow

(1) Set the soil section on the road section
Set the soil section on the road section or use the default soil section.
See Soil section tab in the Section Editor form for further details.

(2) Generate a road
Create a road just like as a normal road, and apply the road section with a soil section on the Vertical curve Editor form.

(3) Open the Soil Calculator by ribbon on the main form
From the ribbon “Analysis”-“Soil calculation”, click calculator button to open the Soil Calculator

(4) Define the calculation case of soil
Define the calculation case of soil. See Soil Calculator form for further details.

(5) Execute the calculation
Click “Calculate” button in the Soil Calculator form.

(6) Check the calculation result and export it
It is possible to check the calculation result, export the table to the file and display the result on the main form.
4Dsimulation

1. Overview
The 4D Simulation plug-in offers different functionalities to manage a schedule starting from creating the tasks of the schedule all the way to launching the simulation to see the progression of your project.
To use the plug-in, go to “Record / Play” tab and click on Gantt chart icon of 4D Simulation. This will open the plug-in main window:

![Gantt chart](image)

2. Gantt-chart
2.1 Schedule

After opening the 4D Simulation main screen, clicking a pull-down menu that can be used for 4D simulation is displayed.

![Pull-down menu](image)

The Gantt chart provides a visualization for the schedule. It contains a calendar that shows the working hours and the holidays. In the gantt chart, the list of tasks of the schedule are displayed.
Using the zoom, it is possible to display different levels of details of the schedule. Drag it to see a smooth expansion or contraction of the span of the calendar.
2.2 Calendar settings

To change the settings of the calendar, select the menu on the Gantt chart, Setting - "Holiday Settings". This will open the following window:
Check the days of the week as weekdays in the Weekdays Settings. Basically, the day without check becomes a holiday. It is also possible to set the Starting Time and End Time of the working hours of the day. Moreover, any day can be individually set as a holiday. Select a holiday date from the calendar and click the "+" button to add a new holiday to the list. To delete a set holiday, select a date in the list and click the "-" button.

2.3 Edit the schedule
2.3.1. Add a Task
Click the following button to add a task.

2.3.2. Add a Subtask
Click the following button to add a subtask.

2.3.3. Edit the task
Click the following button to open Edit Task window.

The following items can be edited for the selected task.
- First Day
- Start Time
- Last Day
- End Time
- Color
- Parent Task

2.3.4 Delete the task
Click a task to delete and then click the following button. Deleting a task will delete all its child tasks.
2.3.5. Move a task to the right / left / up / down
First, select the task to move to the left, right, up or down, and then click on the following button. Moving a task to the left will make it leave the parent task.

3. Simulation

3.1 3D models
When linking a model to a task, it is possible to choose to display the model in different intervals depending on the start and end of the task and the start and end of the schedule.

The start of display can be from:
- Schedule start
- Task start
- Task end

The end of the display can be from:
- Task start
- Task end
- Schedule end

3.2 Setting Resources and Moving Objects

3.2.1 Resources
To add new resources to the project, select "Resources" in the menu. The following window will open.

On this screen, the unique resource name used in the Gantt chart is associated with the corresponding 3D model on UC-win / Road. Add one or more resources using the 'Add' button, then select the 3D model for the added resource name from "3D Model".
3.2.2 Moving Objects

If the model resources used in the simulation is movable, it can be moved for a specific task. The movement can be a rotation, translation or trip on a road or flight path.

On this screen, set the type of movement of the selected model resource and the operation for the type of movement.

Hint:
Model instances are not included in the UC-win/Road project (.rd file) and are created while the simulation is in progress. Task can contain multiple models that perform different actions.

To open the screen above, select a task in the task list of the Gantt chart, then right-click and select "Edit moving objects".

Descriptions:
• Task Name: Displays the name of the task. This item cannot be changed.
• Start Date / Time: Set the start date and time.
• Finish Date / Time: Set the end date and time of the task.

Hint:
The task start date and time can be changed directly on the screen as well as the task end. The duration of the task must include the duration of all actions. Therefore, when changing the date of a task, you may see the following warning:

Hint:
To edit the resource name, select the name and press “F2” key to edit. Set a unique name and press the “Enter” key or change the focus to another place. Use the “Delete” button to delete the selected resource name.
Note:
When changing the span of a task in the edit screen or task list, if the task contains movable objects, the duration of the task must include the duration of all moving objects. Otherwise, the date cannot be changed.

The movement settings include pre-defined movement names in the 3D model editor, movement along roads and flight paths, and sequence of points defined in the 3D coordinates x, y, and z. It is possible to perform multiple operations with the same resource.

The following describes the setting items after clicking the "Add" button:

- **Start Date**: Sets the start date of the movement. Specify the number of days relative to midnight of the start date of the task.
- **Start Time**: Sets the start time of the movement.
- **End Date**: Sets the end date of the movement. Specify the number of days relative to midnight of the start date of the task.
- **End Time**: Sets the end time of the movement.
- **Resource Name**: Sets the target resource name.
- **Movement Type**: Select the movement type. The type that can be selected is as follows;
  - No Movement
  - Flight Trip
  - Point to Point Trip
  - Predefined Movement
  - In Place Movement
  - Delete

**Hint:**
The setting for the start position, end position, and rotation varies depending on the movement type.

- **Path**: Select a flight path when the movement type is "Flight Trip", or "Predefined Movement".
  - **Start Position, FinalPosition / Rotation**
    - Depending on the movement type, it will be the coordinate value, rotation value, distance from the starting point, and movement value:
      - Movement type = No Movement. No settings.
      - Movement type = Flight Trip
        - Start Position: Sets the start position as the distance from the start point of the path.
        - End Position / Rotation: Sets the distance from the start point of the path.
      - Movement type = Point to Point Trip
        - Start Position: Sets the coordinate value of the start position
        - End Position / Rotation: Sets the coordinate value of the end position.
      - Movement type = Predefined Movement
        - Start Position: Sets the coordinate value of the start position of the movement.
        - End position / Rotation: Sets the coordinate value of the end position of the movement.
- Movement type = In Place Movement
  Start Position: Set the coordinate value of the specified position
  End Position / Rotation: Set the rotation value

- Movement type = Delete. No settings.

Use the Add button to add one more model resource. Set the above "Movement settings" and "Movement Duration" for the added items. Use the Duplicate button to create a model resource with the same settings as the selected model resource. Use the Delete button to delete the selected model resource.

Hint:
To set the coordinate value and rotation value, use the "Coordinate Editor" and "Rotation Editor" displayed when selecting each field and clicking the button displayed right side.

Rotation Editor:
The amount of rotation in the moving object setting in the 4D simulation can be set on this screen.

To open this screen, click the field of "Final position / Rotation" in the Moving Objects screen for 4D simulation and click the button displayed in the right.

Note: This screen can be opened only when the "Moving Type" is "In Place Movement".

Yaw: Sets the amount of rotation of yaw axis in degrees.
Pitch: Sets the amount of rotation of pitch axis in degrees.
Roll: Sets the amount of rotation of roll axis in degrees.
OK button: Conforms the coordinate value and backs to the "Moving Objects" screen.
Cancel button: Cancels the change and backs to the "Moving Objects" screen.
Help button: Opens this help topic

To open the above editor, click the field of "Final position / Rotation" in the Moving Objects screen for 4D simulation and click the button displayed in the right.

Note: This screen can be opened only when the movement type is "Point to point trip" and "Start position" and "Final position" are set, and only when the movement type is "In place movement" and "Start position is set.

Using this screen, you can change the following parameters:
Local : Sets the X and Y values in local coordinates.
Global: Sets the X and Y value in global coordinates.

DD: Sets latitude and longitude value in degrees.

DMS: Latitude and longitude in degrees, minutes, and seconds.

Altitude: Sets the height.

The OK button sets the coordinate value and backs to the Moving Objects screen.
The Cancel button cancels the change and backs to the Moving Objects screen.

Camera Settings:
The camera settings to be assigned to the task can be set on this screen.

To open this screen, select a task in the 4D simulation task tree and select "Edit Camera" from the right-click menu.
In the left of the window, Camera settings list displays a list of camera settings assigned to the task.
Use the "Add" button to add a camera setting and the "Delete" button to delete the selected camera settings.

In the right side of the window, it is possible to set up the camera settings.

- Type: Select the type of camera setting from the following.
  - Main Camera: Moves the camera on the main screen.
  - Virtual Display: Displays a 2D overlay of image from the viewpoint registered on the main screen.
  - Camera View: Opens the camera view to open the camera view from the registered position.
- Camera: Select the camera position to be displayed from the camera setting included in the project.

(1) Main camera setting

This setting is displayed when "Main camera" is set for Type.

- Transition time: Enter the time to move to the set camera position in real time.
(2) Virtual display setting

![Virtual display setting](image)

This setting is displayed when "Virtual Display" is set for Type.

- **Display until**: Select the timing to the end of the virtual display from the following items.
  - Task End: Ends the display at the end of the task.
  - Schedule End: Ends the display at the end of the schedule.

- **Size X**: Set the horizontal size as a relative value of the main 3D display area in the range of 0.01% to 100%.
- **Size Y**: Set the vertical size as a relative value of the main 3D display area in the range of 0.01% to 100%.
- **Position X**: Specify the X value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.
- **Position Y**: Specify the Y value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.

(3) Camera view setting

![Camera view setting](image)

This setting is displayed when "Camera View" is set for the Type.

- **Display until**: Select the timing to the end of the camera view from the following items.
  - Task End: Ends the display at the end of the task.
  - Schedule End: Ends the display at the end of the schedule.

- **Size X**: Set the horizontal size as a relative value of the main 3D display area in the range of 0.01% to 100%.
- **Size Y**: Set the vertical size as a relative value of the main 3D display area in the range of 0.01% to 100%.
- **Position X**: Specify the X value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.
- **Position Y**: Specify the Y value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.

Click on the “OK” button to confirm the camera settings and closes this screen.
Click on the “Cancel” button to cancel the setting and closes this screen.
Click on the “Help” button to opens the help page related to the camera settings.

3.3 Starting the simulation

When all the tasks, resources and movements are set, it is possible to play the simulation to see the progression of the project.
Click on the green triangle to start the simulation.
Various simulation settings for 4D simulation can be set on this screen.

The simulation includes the animations and a follow line on the Gantt chart. You can also pause the simulation or move it forward or backward.

It is also possible to see a specific day to see the state of the simulation on that day. This allows you to not only to play the simulation of the schedule but also to investigate the state of the project on a specific day.
4. Import / Export
It is also possible to export into a *.csv file the data created in the 4D simulation plugin including the schedule and all the linked models and resources and actions.

The plugin can also read and import a schedule from a *.csv file. This allows the re-usability and manipulation of schedules previously created in other software in order to launch a simulation based on these schedules using UC-Win/Road.

For the Gantt chart it is possible to select which columns of the *.csv file represent what data so Gantt chart can be created in third party application and easily imported into UC-win/Road.

The plugin reads the tasks and displays them on the Gantt chart. It also imports automatically the 3D Models, if the path to the 3D model file is mentioned in the csv file. The import supports different file format for the 3D models such as 3DS and RM formats.

4.1 Export Schedule
To export a schedule, go to the main menu – File – Export CSV.

Select a name for the exported schedule and save. The 4D Simulation Plugin will be saved into 4 files.
*_.csv: this file will contain the schedule information.
*_.resources.csv: this file will contain the resources information.
*_.movingobjects.csv: this file will contain the moving commands information.
*_.cameras.csv: for camera positions data.

Note: * represents the name chosen for the export. All the models will be exported in the RM format.

4.2 Import Schedule
To import a schedule, go to the main Menu – File – Import CSV. Select the csv file for the schedule you want to import. The plugin will search for the csv files containing the information for resources and moving objects and camera settings in the same directory under the names:
*_.resources.csv: for the resources data.
*_.movingobjects.csv: for moving commands data.
*_.cameras.csv: for camera positions data.

When selecting a csv file, the field selector window opens:
It is possible to have different names or different order of the columns. Make sure to match appropriately the fields with corresponding column in your file.
Choose if the model position is stated in x, y, z coordinates or in Longitude/Latitude.
You can choose to load the models to the project.
The models can be in 3ds or rm format.

5. File format

The exported files are on the following format:

*.csv:
- Task ID
- Task Name
- Start Date
- Final Date
- Number of Days
- Parent
- Color
- Details
- Model GUID
- Model Position
- Model Path
- Model display from
- Model display until

*_resources.csv:
- Resource Name
- Model Name
- Model Path

*_movingobjects.csv:
- Task Name
- Start date
- End date
- Resource Name
- Movement Type
- Path
- Start Position
- Start Rotation
- End Position
- End Rotation

*_cameras.csv:
- Task Name
- Camera GUID
- Camera Name
- View Type
- Param[1]
- Param[2]
- Param[3]
- Param[4]
- Param[5]
- Param[6]

Depending on the type of camera settings, the parameters 1 to 5 refer to:

For Main camera setting:
  • Param [1]: refers to the transition time (i.e the time to move to the set camera position in real time).

For Virtual display setting and Camera view setting:
  • Param [1]: refers to the timing to end of the virtual display (Task End or Schedule End).
  • Param [2]: refers to the horizontal size as a relative value of the main 3D display area in the range of 0.01% to 100%.
  • Param [3]: refers to the vertical size as a relative value of the main 3D display area in the range of 0.01% to 100%.
  • Param [4]: refers to the X value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.
  • Param [5]: refers to the Y value of the upper left corner of the virtual display by the relative value of the main 3D display area in the range of 0% to 100%.

Note:
When importing a schedule, the delimiter between each column should be a coma. The first line is reserved for the names of the fields, the data should start from the second line. It is possible to use other names for the column. Moreover, the order of the column is not mandatory, it is possible to match your own column names with their reference in the Field selector (see 4.2 Import Schedule).
Data exchange with AutoCAD Civil 3D
The data can be imported/exported between UC-win/Road and Autodesk Civil 3D.

4.3.2.1. Exchange the data from Civil3D into UC-win/Road

Go to File - Import or Export - Civil3D Data exchange...
Select Civil3D version to convert to Civil3D 2006 to 2018.

Associated file: File path is displayed here.
[Open] button: Load the data file of the corresponding data file to Civil3D.
*It is only available when the data file is loaded in Civil3D and the corresponding file is different.
[Update] button: Update the form with the current 3D document.

Individual Road
Civil 3D: The roads which only exist in Civil3D are listed. Select roads to be exchanged and click [Import].
UC-win/Road: The roads which only exist in UC-win/Road are listed. Select road to be exchanged and click [Export].

Shared roads:
Common roads between Civil3D and UC-win/Road are listed. Click a road from Civil3D or UC-win/Road, and click [→]. Click [←] to release it. Then click [Upload] to specify it as a common road of Civil3D and UC-win/Road.

After selecting the road and importing them, go to [Next].
Setting of terrain
When [Surface] is checked, the terrain will be exchanged.

Setting of sub-assembly
Sub-assembly generated by Civil3D is listed.
Check Sub-assembly to be transferred to the constitution of the section.

Set up the properties and the textures of checked sub-assemblies.
Check the section configuration
The items are listed in tree view. Click an element to be highlighted in bold line.

Section Option
Go to Section Option. Edit a road slope on "Cutting/Banking" tab, a roadway boundary and lane gape on "Road marking" tab.
Click Next to exchange it.

2. Exchange from UC-win/Road into Civil3D

Go to Data exchange… in UC-win/Road. Select the roads to be exchanged and click [Export].
Click Next to display the selected elements.
Click an element to be highlighted in red frame, which can be moved by dragging the mouse.

The terrain and the selected road terrain are displayed in Civil3D.

**Setting of the surface display**
When the data are linked, only the surface borderline is displayed.
To display the components, select the surface border in the window, right click and open [Edit the surface style]. Then check the display ramp on [Display] tab.

It is possible to change the colors depending on the altitude on Analysis tab.
この画像は、日本語で書かれたテキストと、グラフや図表を含むスクリーンショットです。テキスト内容は画面の詳細な見出しや項目名、数字、ボタンなどの情報が含まれています。画像の右側には、地図のようなグラフが描かれています。全体的に、ソフトウェアのインターフェースが示されているようですね。
Crowd
Character settings

It is possible to register character models from Edit - Scene - Library - Model Panel. Characters are either in MD3 format or FBX format. They can visualize pedestrians, bicycles, strollers, wheel chairs, and even animals like dogs, cows, birds, and fishes. Load MD3 character models or FBX character models, which saved under C:\UCwinRoad Data\x.x\Characters. Each FBX character can be saved with textures and loaded from Menu - Load – MD3 model, FBX model.

Load MD3
Select a character and click Open to open "MD3 character editor". The character is added to the "Character List Editor" window.

Load FBX
Select a character and click Open to open "FBX scene editor". The character is added to the "Character List Editor" window.

2. Download MD3 / FBX
Download MD3 or FBX from Edit - Scene - Library - Model Panel menu.

3. Edit MD3 character

Edit MD3 character by right clicking - Edit.

It is possible to customize its movement, skin (clothes), scale (size), width and speed.

Symbolic mode: it will be displayed in dots or lines when it is located from a certain distance from the camera. The distance can be changed on the performance tab of "Visual options".

Display speed: The speed of movement. Move the scroll bar to adjust it.

Display stats: see stats of the character.
4. Edit FBX character

Edit FBX characters on the "FBX scene editor" form. Select a character and click "Edit" to open the "FBX scene editor" form.

**Scene name:** Name of character.
- **Camera buttons:** Position and angle of camera
- **Full size:** Resets to default size.
- **Background:** Background color of the view
- **Day:** Display a character in the day time.
- **Night:** Display a character at night.
- **Thumbnail:** Create a Thumbnail to specify this scene. The current view will be used as new one.
- **Play/Stop button:** Start/stop animation.
- **Animation drop-down menu:** Animation names included in the FBX scene model. It can be played as a preview.

**Control area surrounding the model**
- **Spotlight:** Cast a spotlight on it from the top. Brightness can be adjusted using the scroll bar.
- **Axis:** Show global coordinates axis.
- **Grid:** Show grids. The size can be adjusted by changing the value.
- **Information:** Show size of the FBX model and the number of polygons.

**Offset tab**
- **Scaling:** Its size can be changed in X, Y or Z direction.
- **Factor:** Offset factor can be adjusted.
- **Rotation:** Rotation angel of Yaw, Pitch, and Roll.
  *If the character is facing the opposite direction, set yaw angle to 180 degrees.
- **Local axis:** Show local axis.
- **Position:** FBX scene model can be moved in X, Y, or Z direction.
- **Auto-adjust:** Move the model back to the center

**Structure tab**
Click "+" to expand the node. Double click it to zoom in.
Right click it to show the following options.
- **Find:** Double click to zoom in the node in question.
- **Edit:** Open Fbx node editor form
- **Delete:** Delete the node. It cannot be restored.

**Options tab**
- **Ignore transparency from:** Illumination drawing factor. If the distance between the camera and the model is greater than the value calculated by multiplying the models size by this factor, OpenGL will ignore the transparency. For instance, if100x is set to a 10m model, model's transparency will be ignored when the camera is at a position further than 1100m from the model. Drawing performance will be improved.
• Animation

**Take manager**
Open "FBX Scene Take Manager". Adjust the animation, the movements.

- **Animation speed**: Animation speed of FBX scene model in km/h.
- **Symbolic mode**: Display a model in dot or line when a model is at a certain distance from the camera.
- **Color**: Color of symbolic mode.

**Animation root setting**
Set up the coordinates and nodes of frames at the beginning and end of animation separately.

---

**Sound source**

- **Validate sound source**: Place a sound source in the FBX scene model.
- **Property**: Apply a property of the sound source to the FBX scene model
- Sound can be used for the noise simulation.

5. Add Walkers

Place the character on a flight path.
*Add Walkers* button is effective only when some flight path has been created.
First create a flight path, then click a character on "Character List Editor" - "Add walker".

---
Advanced Movement: Walk or run.
Skin: Skin (clothes) of the walker.
Scale: Size of the character in %.
Width: Width of the character in %.

*MD3 character will move at a speed set on "MD3 Model Editor",
and FBX character will move at a speed set on "FBX scene Model Editor".

Movement Settings
Path: Choose a flight path.
Start Position: Start position of the route. (Range: 0.00 - 2m before Finish Position)
Finish Position: Finish position of the route. (Range: 2m ahead of Start Position - Total length of flight path)
Initial Position: Initial position of the route. When multiple characters move along the same flight path, input different values

for each character to avoid overlapping with each other.

6. Display characters

Click "Start traffic movement" button to display characters.
Pedestrian settings

Go to the ribbon, Edit - Pedestrian - Network to have crowd moving in 3D space.

Network of walking routes (pathways) can be built within 3D space in which multiple pedestrians can be arranged at one time. Each pedestrian determines their own path to walk on while they avoid bumping into each other.

1. Create/edit network

Create network and pedestrian profile in "Network List Editor."

Networks

- **New button:** Open the Network Editor to add a new network.
- **Edit button:** Open "Network editor" window, where it is possible to create nodes and pathways within the network, and arrange pedestrians on pathways.
- **Delete button:** Delete the network that is selected.
- **Merge button:** Merge the networks that are selected into one. Hold down the Ctrl key to select multiple networks.

Pedestrian

- **New button:** Add a new pedestrian profile.
- **Edit button:** Open the "Pedestrian Profile Editor" window. It is possible to select a profile from profiles already registered, and edit it.
- **Delete button:** Delete the profile that is selected.
2. New

(1) Click "Pedestrian network" in the "Visual Options" window.
(2) Go to the ribbon, Edit – Pedestrian - Network, and click "New" on Pedestrian.
(3) "Add" the character to place on the scene, adjust parameters for its proportion, min/max speed, and movement, and click OK.
   *Speed is selected at random within the range of max. and min. speed.

(4) Then, add a new network.
(5) Select the new network and click Edit.
(6) Click on "New" under "Pathway".

(7) Click a pathway on which to arrange pedestrians.
(8) After clicking on the final point on the pathway, click “End path” to end it.

(9) When clicking OK in "Add pathway" window, "Network editor" window containing added pathways and nodes will be displayed.
(10) In "Network editor" window, select the pathway and click "Edit".

- **Name**: Assign any name as needed.

[**Properties**]
- **Weight**: Assign a weight to the pathway.
- **Start/End node**: Pedestrians will appear between the start node and end node.
- **Flux direction**: Pedestrians will walk in this direction.
- **Speed**: Average walking speed of pedestrians in km/h

[**Shape**]
- **Width**: Width of the pathway on which pedestrians will walk.
- **Left/Right angle**: Inclination of the left/right side of the pathway. If the side of the pathway is inclined upwards relative to the center of the pathway, then the angle will have a positive value. If it is inclined downwards, then the angle will have a negative value.
- **Vertices**: Coordinates of each vertex (turning point). It is possible to add new vertex or delete a vertex. And, by splitting a path at a vertex that has been selected, the vertex will turn into a node and the pathway will be split into two.
- **Look at**: Camera will change its orientation. It is possible to look at the pathway in the main window.

(11) Click OK in the "Pathway Editor" window.
(12) On the “Pedestrians” tab in the “Network Editor” window, click “Add” to add pedestrian profile.

(13) Set maximum populate, rate, and initial population.

Set the followings.
- The weight of pedestrians profile
- Crowd’s maximum population
- The number of pedestrians generated per hour (Rate)
- Crowd’s initial population

*Check “Enable” to use the pedestrians profile in the network.

(14) Go to Options - Traffic Movement - Start Traffic Movement. The crowd appears.

*Position/height of the start/end node or points in between can be edited by clicking on each of them in the main window.
EXODUS Plug-in

1. Overview
It is possible to import simulation results from building EXODUS and animate the evacuation in UC-win/Road. EXODUS plug-in is required.

(1) Import file
It is possible to import the graphic post processor combined to EXODUS project, .very and .vrs files used by vrEXODUS. The .very files contain the geometry information of the environment. And the .vrs files contain the information of each character’s movements. This information is used to calculate the each character’s trajectories.

(2) Expression of 3D character
In UC-win/Road, the term “Avatar” is used for EXODUS characters. A model chosen from MD3 model that is already read into the UC-win/Road project at random is used to express an avatar. Since MD3 model is chosen at random, it is not possible to express visually the age, sex, and body type for each avatar. This information is displayed on the data list in the scene graph tab of an EXODUS screen. Presently, walk and standby is supported as the action of an avatar. Crawling and running are not supported.

(3) Import of the EXODUS data to UC-win/Road environment
The UC-win/Road project is required. Create and arrange 3D models within a project in order to express a building. Creation of the parameter set used for adjusting data is needed. It is created on EXODUS screen option tab before and after importing EXODUS data.

(4) The display of a character
Start an environmental display function.

(5) The check of animation
Some EXODUS original viewpoint modes are supported. With evacuation scene mode, it is possible to view the evacuation scene from various angles. With EXODUS camera mode and viewpoint mode, it is possible to display the movement of each character on EXODUS screen camera manager tab.

(6) EXODUS data
EXODUS data is not saved in UC-win/Road. Instead, the relative path for project file of each .vrg and .vrs file is saved in a project. If a project is opened again and EXODUS plug-in is launched, these files will open automatically. The files can be distributed and copied together to combine the data of EXODUS and UC-win/Road.
2. The work flow
(1) Create a UC-win/Road project and a building model.
(2) Import EXODUS data and define EXODUS parameters.
(3) Execute EXODUS animation, and search with EXODUS viewpoint mode or check EXODUS animation data
(4) Create and execute the script incorporated EXODUS animation

3. Operating procedure
(1) EXODUS Ribbon
It is possible to operate the animation using the ribbon menu.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open EXODUS screen.</td>
<td></td>
</tr>
<tr>
<td>Read EXODUS data.</td>
<td></td>
</tr>
<tr>
<td>Pause the animation.</td>
<td></td>
</tr>
</tbody>
</table>
| Previous Step, Play the animation, Next Step  
Note: Animation pauses by the first click. Although it is backed to the previous step by clicking again, it is still pausing. Animation will be resumed by clicking PLAY button after that. |
| Stop and rewind the animation. |
| Specify the PLAY speed of animation. |
| Move the Animation to the certaintime. |
| Repeat the animation. |

(2) EXODUS Screen tab
Define EXODUS import data on the screen tabs as below.

<table>
<thead>
<tr>
<th>Screen tab</th>
<th>Description</th>
</tr>
</thead>
</table>
| Camera manager  
Search for EXODUS scene using EXODUS original viewpoint mode. |
| Data tree view  
Display the detail information of the floors and avatars. |
| Option  
Edit the data position and the height of stairs. |
(3) Import EXODUS data
In order to import data certainly, .vrs and .vrg files must exist in the same directory as the same file name.
Create a UC-win/Road project and arrange a 3D building model beforehand.

(4) The import process
1) Click EXODUS link on ribbon menu.
2) Click Import button on EXODUS screen.
3) specify an EXODUS data directory and one of (vrg, vrs) pairs for the EXODUS project.
4) Click Open to start importing.

(5) Define EXODUS parameters
Define the parameter of position offset and environment. It is required in order to calculate correctly the trajectories of avatars and the bottom altitude of the building.
Note: Open the related UC-win/Road project before editing the parameters. If EXODUS data is already imported, it is possible to arrange a building in advance.

(6) The method of parameter settings
- Position offset
Specify the appropriate values in direction, Pos X, Pos Y and Pos Z.
Note: If the coordinate values currently used are not known, specify them visually.

- Environmental model
Specify the new name of environment as needed.
Adjust the bottom altitude of building for all stairs or specified stairs as below.
a) For all stairs: Display all stairs on the class list and specify the height.
b) For specified stairs: Choose the stairs from the class list and specify the height. Repeat it to specify the heights separately for each stairs.

The changes will be reflected immediately. Once the position of avatar is set, it is possible to generate the EXODUS animation and the script.

(7) Execute EXODUS animation
Open a UC-win/Road project, to which the EXODUS data was imported.

Click to display characters.
The environment and characters will start automatically. The camera position is above the characters.

1. Move to the animation starting position with the viewpoint mode of UC-win/Road.
2. Click EXODUS link on ribbon Simulation Link – EXODUS Link
3. Execute and control the animation by animation reproduction option on EXODUS screen.
To view the animation from a viewpoint of specified avatar, click Stop button and input the avatar number on camera manager tab. Then restart the animation.
4. Execute the animation on different conditions as needed. Select other avatar or other camera mode
5. Click Stop to finish.

(8) Display animation data
It is possible to display the animation data relevant to stairs or avatars
1) Click EXODUS link on the ribbon
2) Expand the stairs or avatars on scene graph tab.
3) Click Level # or Avatar # to display the information on the right.
**Script (auto presentation)**

When the script to which the content of presentation is set by the command beforehand is registered, a chain of flows can be presented automatically without operating it one by one.

Click in **Script / Animation** to create a new script, and click to edit it. After settings, click to run the script.

*Checking "Loop" can run the script repeatedly.

**Animation list**

It is possible to play animation by playing the scripts in order.

- [New Animation]: Add a new animation in the list. Click on the animation name in the lift to rename it.
- [Copy]: Copy the selected animation. Scripts added into the animation are also copied.
  * In the meaning where two or more the names exist, the number of the name immediately after the copied such as 2 and 3 attaches at the end of the name.
- [Delete]: Delete the selected animation.
- [Run]: Execute the selected animation in order of script registered. It ends when the last script is finished
  *The button is disabled when all the script are empty registered in the animation.
  *When the script registered in the list of animation is selected, it is executed from the script selected about the animation. It returns in the beginning when executed to the last one and it executes it.
- [Repeat]: Repeat the animation.
- [<< Add Script]: Add the selected script in the list on the right side into the selected animation.
- [Remove Script >>]: Remove the selected script in the animation list.
- [Up] / [Down]: Move the selected script up/down.
Example (Sample Data : "Nihondaira Park Way")
**Movie Options**

Higher-precision animation can be created by integrating AVI plug-in and the POV-Ray plug-in and outputting automatically the POV-Ray script in each frame of AVI.

1. Movie Manager Options

Open the Movie Manager Options form from Record/Play Ribbon - Options in Movie tab.

**Export Folder**
- **Default Folder**: "<<User Data Folder>>\Movie" is the default.
- **Specified Folder**: Specify an export folder. If the path does not exist, the path is created automatically.

[Note] When new files are exported, they overwrite the files that already exist. Move the files before exporting new files to avoid losing data.

**What to export**
- AVI files
- BMP files
- PNG files
- POV-Ray Files

**Export view**
- **Main view**: Export the main view
- **Camera view**: Select the saved camera view to be exported. The camera view is automatically displayed if the selected one is not shown.
- **Main view Aux**: Check it to record OpenGL Aux buffer.

**Settings**
- **Invert Horizontal**: Output the image reversed right and left.
- **Export Interface**: Output the image for interlaces. Select the odd number line or the even number line.
- **Clip Export Area**: Define the range of output. The image within this range is stretched to the size set in the AVI Option.

[Note]
- The options, Invert Horizontal, Export Interface and Clip Export Area, are only for AVI.

AVI, AVIBmp and POV-Ray folder of each view for AVI output are created.

Stop AVI recording from Record/Play - Movie. Then a folder titled "Date of starting the record_Record" will be generated in the folder, where the folder of the view for the output is generated.
Hint about the folder name
The folder of the main screen is named "Main_View", and the one of the camera view is named "Camera_the camera position name". If the camera position name contains the following characters, it is converted to "_ (underscore)".
¥ (Backslash), / (Slash), :(Colon), . (Period, dot), ;(Semicolon), * (Asterisk), ? (Question mark), " (Double quotation), <(Left angle, Less than), >(Right angle, Greater than), | (Pipe), CON, PRN, AUX, NUL, COM1-9, LPT1-9, CLOCK$

The number of the file name is the one of the beginning and the end. It corresponds with each image when AVI bit map is output.

2. AVI Options
Record simulations in AVI. Open the setting by clicking AVI Options on the Movie Manager Options

AVI Options

<table>
<thead>
<tr>
<th>Width</th>
<th>Height</th>
<th>Frame Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1520 pixels</td>
<td>1024 pixels</td>
<td>30 fps</td>
</tr>
</tbody>
</table>

- Use Screen Size
- Maintain Aspect Ratio
- Stretch to Fit
- Draw Border

Compression

- Codec: Logitech Video (MJPEG)
- Quality: 100%

Width: Width (pixel) of the screen.
Height: Height (pixel) of the screen.
Frame Rate: Frame rate in 1 second. As default, 30 frames/sec (30 fps).
Maintain Aspect Ratio: Maintain the ratio of width and height on AVI recording.
Stretch to Fit: Check this when the aspect ratio of the AVI screen is different from one of the display.
Draw Border: Draw borders around the AVI.

Codec: Select a compression engine.
The engines which are installed on the PC are listed. The high AVI file with high compression rate can be created in high-precision by using DivX.
Quality: Set the quality used for Microsoft Video 1.

3. AVI Output

Start AVI Recording: Click "Record/Play" Ribbon - "Start"
Stop AVI Recording: Click "Record/Play" Ribbon - "Stop"
The AVI file is saved in "UCwinRoad folder\AVI folder".

The size of the screen is stretched to fit with the height and the width of the screen to be captured.
The current screen size is displayed from "File" menu - "Output Bitmap File".
Shapefile Plug-in

Overview
Shapefile is the file format that is developed by ESRI, and is generally used in the world of GIS today. Shapefile means not one file format but the GIS data which consisted of two or more files. At least three files (shp, shx, dbf) are required for one data set, and eight more kinds of option files exist if needed. It can respond to "the peak, the poly line, and a polygon" as geometric information, for example, it can be used for geographical feature, a road, a lake, a river, etc. Furthermore, the attribute data for actually using data is also contained.

How to Import Shapefile data
Read the shapefile or read the factor of shapefile into exiting project to the base edited as a new project by using Shapefile importing screen. Open the screen from File - Import - Import Shapefile.
Road plane alignment, geographical feature altitude, and the information on a building can be directly read from shapefile.

(1) Importing procedure
1) Import as new
It is possible to import Shapefile as a new project. This option creates the base project used to be edit in future.

2) Merge
It is possible to import Shapefile into the existing project. The project which reads Shapefile data must be read into UC-win/Road then. This option is helpful to add a deeper detail and realism than the existing project.

(2) The coordinate system of Shapefiles
Specify the coordinate system relevant to Shapefile. Choose from Japan, New Zealand, and other areas.

<table>
<thead>
<tr>
<th>Coordinate System of the Shapefiles</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ ] Japan</td>
</tr>
<tr>
<td>[ ] New Zealand</td>
</tr>
<tr>
<td>[ ] Other</td>
</tr>
</tbody>
</table>

Default Terrain Height: 1.1 in
1) Japan
Choose this option, when Shapefile data is Japanese data.

-Detail setting
Choose the area number of the actual coordinate system relevant to Shapefile data, and a coordinate system. Geographical feature certainly applicable to the position in Japan is created, and the coordinate value of the project is correctly changed to suit a project.
Choose from the following items: World land survey system (Japanese land survey standard 2000), the old land survey system, No(Input the area number of a coordinate system (1-19))

Note: UC-win/Road changes a coordinate value into a world land survey system at the time of synthetic import with the existing project. Refer to conversion of Shapefile.

This cannot be used:
-When a Japan.map file does not exist in ¥Data directory.
-When the input method is in synthetic import, and the exiting project is not Japan, or when the region setting is not Japan including copy.

2) New Zealand
Choose this option, when Shapefile data is data of New Zealand.

-Detail setting
Choose the actual coordinate system relevant to Shapefile data. Geographical feature certainly applicable to the position of New Zealand is created, and it is correctly changed so that the coordinate value of Shapefile may suit a project.
Choose from the following items: Horizontal Mercator projection, New Zealand Map Grid

Note: UC-win/Road changes a coordinate value into horizontal Mercator at the time of synthetic import with the existing project. Refer to conversion of Shapefile.

This cannot be used:
-When a New Zealand.map file does not exist in ¥Data directory.
-When the input method is in synthetic import, and the exiting project is not New Zealand, or when the region setting is not New Zealand including copy.

3) Other area
Choose this option, when Shapefile is the data of countries other than Japan and New Zealand. UC-win/Road does not save the map data for foreign countries. Therefore, the coordinate conversion processing cannot be performed. If this option is chosen in a new project, flat geographical feature will be created in the position relevant to the present application option default area setting.

-Detail setting
Input the initial geographical feature altitude in the position which creates blank project.
By a default, the altitude of 1.1m above sea level is created.
(3) Shapefile for conversion

All added Shapefile(s) are displayed in a list. Click Add Files button to choose the file and define the parameters.

1) The addition of a file
Search the shape formal file (.shp) relevant to Shapefile to be read by clicking this button.

Note:
- Shapefile consists of sets of some files which save core data together. Therefore, it is filtered to display only shp format file.
- UC-win/Road cannot open a shape formal file (.shp), unless the attribute formal file (.dbf) relevant to the same directory exists.

2) File name
The name of all .shp (shape form) files added to the screen is displayed in a list.

3) Type
When a certain Shapefile is added to a list for the first time, it is displayed unknown until it defines as the parameter in reading. Once it is defined, the file type, geographical feature, a road and a building in UC-win/Road will be displayed.

(4) The parameters in reading the files
Set up the rules when UC-win/Road convert Shapefiles.

1) Object
It displays if Shapefile includes the peak, a line or a polygon. This information is useful to determine which file is changed by UC-win/Road. The table below shows the object type and what UC-win/Road will be transferred.

<table>
<thead>
<tr>
<th>Object</th>
<th>Point</th>
<th>Line</th>
<th>Polygon</th>
</tr>
</thead>
<tbody>
<tr>
<td>Terrain</td>
<td>☐</td>
<td>☐</td>
<td>☐</td>
</tr>
<tr>
<td>Roads</td>
<td>☐</td>
<td>☐</td>
<td>☐</td>
</tr>
<tr>
<td>Buildings</td>
<td>☐</td>
<td>☐</td>
<td>☐</td>
</tr>
</tbody>
</table>

Note: The options are different in each object.

Terrain
The object will be changed to the terrain.

Altitude field
Choose the file attribute field for using for terrain altitude information.
Note: it is required to choose it.

Road
The object will be changed to the road.

Road name field
Choose the file attribute field used for a road name. Note: If a blank or None is selected, the default name is used for each road. A number is added to each road during conversion, Road 0 for first one, and it will be continued for next one.

**Building**
The object is changed into 3D model building.

**Height field**
Choose the file attribute field which defines the height of each building. All buildings will be changed according to actual height.

Note: If the height value cannot be read for a certain building from the field, the column is created by using the default height (in standard conversion setup of building).

If this is blank, the default value of height will be used for all buildings. UC-win/Road defines the wall of a building using a polygon. And a square pillar is created using the actual height or default height of the building entered here.

**Building name field**
Choose the file attribute field used for the name of 3D building model created.

Note: If there is no available data for the building name, and a blank or none is chosen, the name assigned for each model by the system is used. The default name of the model + its position in coordinate is given for generated model as default model. Example: Building, x = 9031, y = 9303.

(5) The conversion method
Select one of the option below to specify how the road changing direction point is defined during conversion.

Note: The selected option is applied to all listed roads listed by Shapefile for conversion.

1) Keep as Break Line
The point changing the direction is located as a circle with a radius of 0.1m.

2) Circle
The point changing the direction is located as a circle that is automatically located. A radius is calculated depending on the deviation of the point before and behind on plane alignment changing direction and distance.

3) Spline
A spline road will be created and each point will be used as the peak of a road. Detail information how UC-win/Road manages conversion of Road Shapefile, refer to conversion of Shapefile.
A standard conversion setup for a building

The setting here is applied to all the buildings Shapefile listed for Shapefile conversion in global value.

<table>
<thead>
<tr>
<th>General Conversion Settings for Buildings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default Height</td>
</tr>
<tr>
<td>Group Buildings</td>
</tr>
<tr>
<td>Surface Details</td>
</tr>
</tbody>
</table>

**Default height**
Define the height of a building. (Range: 1-100000m)
This information is used when it corresponds to below.
-When none is set as the height field of a building, all the changed buildings becomes the same height.
-When a numerical value cannot be read in the specified height field, the building in this state becomes default height.

**Detailed expression of the wall surface**

If this check box is checked, each wall will generate the building model created as each surface used on the “3D Model Editor” screen (new building). Since this can set a separate texture as each wall of an after-conversion building, it is effective.

For example, in the following figures, a surface texture applied to all the surface of a wall of the building changed into the edit display of 3D model is listed. Unchecking the box will generate a building with a simpler, but integrated layered building model.

In the 3D Model Editor window, surface texture that will be applied around the entire building are listed.

**Grouping of a building**

If a check box is checked, the buildings are grouped in specific square area. Grouping will become effective if the value of one side of area is set up on the right-hand side.

When a buildings are grouped, a 3D model which consists of many buildings is created.
It is possible to edit the building model together by using “3D model Edit” screen (new building), so it is effective to manage a lot of building models.

**Note:** If it is judged that each building model is too many to create during a change by system, it is automatically grouped according to the information entered in creating dialogue of building grouping.
Building texture setup
The building texture screen is opened, and the default texture for all building roof or wall to be changed can be set.

Wall surfaces
The default wall surface texture applied to this building can be set up respectively in daytime and nighttime. Or if "No Texture Applied" is chosen, a default color can be changed.

Roof surfaces
The default roof texture applied to a building can be set up respectively in daytime and nighttime. Or "No Texture Applied" is chosen, a default color can be changed.

(6) Preview area
The preview of the object contained in Shapefile to be read from now on. It is helpful when determining which file is used in UC-win/Road if the object type is not clear.

- Selection of reading area

Note: When Shapefile is synthetically imported to the existing project, the geographical feature, a building, and a road of the existing project are displayed here.

<table>
<thead>
<tr>
<th>Preview color introductory notes</th>
<th>Shapefile Object</th>
<th>The object of the existing project</th>
</tr>
</thead>
<tbody>
<tr>
<td>Terrain</td>
<td>Deep gray</td>
<td>Thin gray</td>
</tr>
<tr>
<td>Road</td>
<td>Blue</td>
<td>Green</td>
</tr>
<tr>
<td>Building</td>
<td>Bright green</td>
<td>Non-display</td>
</tr>
</tbody>
</table>

The maximum project area guide
The maximum project area where the visible outline of a purplish red color can read an object is shown.

New project
When having read data as a new project, the visible outline shows the area of 20km x 20km, mainly the reading area. (see the selection of reading area). The visible outline shows the maximum area which can be used in UC-win/Road. The area exceeding this project area cannot be read. Since the size of actual project area is changed when related map data is processed in the case
of Japan and the New Zealand project (or project using these area setup), for details, refer to Shapefile conversion.

In case other project, if a selected area is located within a region of a guide, a guide can be moved. refer the movement of project area about this operation.

Note: After selection of reading area, a reduction function may be needed to see a guide.

**Existing project**

When reading data into the existing project, the actual range of the existing project is displayed inside a guide. The object outside this area is not read.

All the objects settled in the existing project area can be imported (When the area to read is chosen). Or the object of the selection area in a project boundary can be read. Refer to the selection of reading area for this operation.

Note: About the project using Japan map data, the east-and-west width and the north-south width of the area of the existing project are about 20km by the processing method of Japan map data. When choosing reading area, it is necessary to know this, and it needs to exceed the maximum project size which is 20kmx20km, and needs to make it the contents. For details, refer to Shapefile conversion more.

Note: Project area is unmovable to the existing project.

### Tool bar

The following operations can be performed by preview.

<table>
<thead>
<tr>
<th>Operation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Expand</td>
<td>Display area is expanded based on the center of display area.</td>
</tr>
<tr>
<td>Reduce</td>
<td>Display area is reduced based on the center of display area.</td>
</tr>
<tr>
<td>Move</td>
<td>Move the preview area to see the invisible part.</td>
</tr>
<tr>
<td>Whole display</td>
<td>Back to an initial display state.</td>
</tr>
<tr>
<td>Selection of reading area</td>
<td>Select the range to be read Shapefile.</td>
</tr>
<tr>
<td>Movement of reading</td>
<td>After the area to be read is selected, this button allows moving this area.</td>
</tr>
<tr>
<td>Movement of project area</td>
<td>Once the area to be read is selected (see the note below), the position of the project area can be changed whenever the place that is read by this movement button. <strong>Note:</strong> This button is only available when a new project is read in the coordinate system in other area.</td>
</tr>
</tbody>
</table>

**All files**

With this default option chosen, the preview of all listed Shapefile(s) is displayed. When it is synthetically imported, the road and the terrain of the present project are included in the existing project.

**Selected file**  This displays only the preview of the Shapefile currently selected.

**Only the present project**

This option is only available when the synthetic reading procedure is chosen. If this option is chosen, only the data relevant to the present project will indicate by preview.
(7) Import
Shapefile object is changed according to the setting.
A conversion situation is displayed before reading is completed. Check a situation message, and when solution is a required problem, refer to solution of the problem of the error message in reading. An example of the successful conversion situation message is shown on the right.

Close.
a screen will be closed and one processing will be performed in following.
- If conversion is successful, or if it succeeds partially, the procedure of creating of a new project, or synthetic reading to the existing project is completed.
- When conversion is failed, the screen will be closed without processing anything.
- If the close button is clicked, the screen will be closed without holding data.

It may be shown that the item in Shapefile may be unable to be changed into some messages displayed on the panel which displays a conversion state. The project of the result for giving the determination of which Shapefile item was not changed is shown in a preview panel.
**GIS View**

GIS View is the tool to import, display, edit and export the GIS data. It is possible to import the GIS data, and integrate the map file (FGW: Forum8 GIS Workspace). It allows GIS data to import, integrate to the map file (FGW: Forum8 GIS Workspace) and display in UC-win/Road or Google Earth.

1. **Import and Export**

<table>
<thead>
<tr>
<th>File Type</th>
<th>Args</th>
<th>Supported Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>shp File</td>
<td>ESRI shape file type</td>
<td>Import/export</td>
</tr>
<tr>
<td>dxf File</td>
<td>Auto CAD file type</td>
<td>Import/export</td>
</tr>
<tr>
<td>kml File</td>
<td>Google Earth Polygon, Polyline, Model file type</td>
<td>Export</td>
</tr>
<tr>
<td>slm, slp, sal, xml File</td>
<td>Japan Geographical Survey Institute Digital map 1/25,000,1/2,500</td>
<td>Import</td>
</tr>
<tr>
<td>Mem File</td>
<td>Japan Geographical Survey Institute 5m,10m,50m Mesh</td>
<td>Import</td>
</tr>
<tr>
<td>Text File</td>
<td>Polyline text</td>
<td>Import/export</td>
</tr>
<tr>
<td>Image File(.*bmp,.tif)</td>
<td></td>
<td>Import/export</td>
</tr>
<tr>
<td>fgw File</td>
<td>FGW : Forum8 GIS Working space file</td>
<td>Import/export</td>
</tr>
</tbody>
</table>

2. **Function List of exporting to UC-win/Road**

The function of exporting to UC-win/Road is listed as below.

<table>
<thead>
<tr>
<th>Object type</th>
<th>Property</th>
<th>Remarks</th>
<th>UC-win/Road</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point</td>
<td>None</td>
<td>The point and the group points included in the optional shape.</td>
<td>Terrain patch</td>
</tr>
<tr>
<td></td>
<td>Height</td>
<td>Define the terrain elevation</td>
<td>Terrain elevation</td>
</tr>
<tr>
<td>Line, Polyline</td>
<td>None</td>
<td>Optional line and polyline are exported as a road line</td>
<td>Road</td>
</tr>
<tr>
<td></td>
<td>Name</td>
<td>Set the road name to be exported.</td>
<td>Road name</td>
</tr>
<tr>
<td>Spline</td>
<td>None</td>
<td>Optional spline curve is finely cut, and exported as road horizontal line.</td>
<td>Road</td>
</tr>
<tr>
<td></td>
<td>Name</td>
<td>It can be set for exporting the road name</td>
<td>Road name</td>
</tr>
<tr>
<td>Polygon</td>
<td>None</td>
<td>Optional polygon is exported such as the block is pushed to the given height in the space</td>
<td>3DModel (building)</td>
</tr>
<tr>
<td></td>
<td>Height</td>
<td>Define the block height</td>
<td>Height of model</td>
</tr>
<tr>
<td></td>
<td>Name</td>
<td>Set the model name to be exported.</td>
<td>Name of model</td>
</tr>
<tr>
<td>Rastar Image</td>
<td>None</td>
<td>The image layer is exported as satellite image or street map of UC-win/Road.</td>
<td>Satellite image</td>
</tr>
</tbody>
</table>
## 3. Window

### 1. Menu, toolbar

### 2. Layer list

### 3. Map view

### 4. Navigation map

---

### Menu, Tool bar

#### 1) File

<table>
<thead>
<tr>
<th>Menu</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open map</td>
<td>Open FGW file</td>
</tr>
<tr>
<td>Close map</td>
<td>Close the current FGW file</td>
</tr>
<tr>
<td>Save</td>
<td>Save the current FGW file</td>
</tr>
</tbody>
</table>

#### 2) View

<table>
<thead>
<tr>
<th>Menu</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Show Full Extent</td>
<td>Display the map in full size within map view</td>
</tr>
<tr>
<td>Panning</td>
<td>Remove the map view by dragging</td>
</tr>
<tr>
<td>Zoom In</td>
<td>Left-click on the map view position, drags it to specify the rectangle area and zooms in.</td>
</tr>
<tr>
<td>Zoom Out</td>
<td>Zoom out around the position where the map view is left-clicked</td>
</tr>
<tr>
<td>Dynamic Zoom</td>
<td>Zoom in/Zoom outs by dragging.</td>
</tr>
<tr>
<td>Centre Zoom In</td>
<td>Zoom in around the map view.</td>
</tr>
<tr>
<td>Centre Zoom Out</td>
<td>Zoom out around the map view.</td>
</tr>
<tr>
<td>Refresh</td>
<td>Refresh the map view.</td>
</tr>
<tr>
<td>Previous View</td>
<td>Display the previous map view.</td>
</tr>
<tr>
<td>Next View</td>
<td>Display the adjacent map view, then further map view</td>
</tr>
</tbody>
</table>
### 3) Edit

<table>
<thead>
<tr>
<th>Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select object</td>
<td>Select the object.</td>
</tr>
<tr>
<td>Select object by area</td>
<td>Select the object within the rectangle area by dragging.</td>
</tr>
<tr>
<td>Select/Unselect all</td>
<td>Select/Unselect all available objects.</td>
</tr>
<tr>
<td>Selected object's information</td>
<td>If the object is selected, the information will be displayed in [Object information] window.</td>
</tr>
<tr>
<td>Cut</td>
<td>Cut the selected object.</td>
</tr>
<tr>
<td>Copy</td>
<td>Copy the selected object.</td>
</tr>
<tr>
<td>Paste</td>
<td>Paste the copied object to edit layer.</td>
</tr>
<tr>
<td>Move object</td>
<td>Select and move the object. Object is selected by first click, then it starts to be moved by second click, and placed by third click.</td>
</tr>
<tr>
<td>Edit Vertices</td>
<td>Select the object, and edits the vertex. Edit will be finished by double clicking, and the edited information is saved.</td>
</tr>
<tr>
<td>Rotate object</td>
<td>Select and rotates the object. Object is selected by first click, and then decides the rotation center position by second clicking and the rotation will be finished by third clicking.</td>
</tr>
<tr>
<td>Delete selected object(s)</td>
<td>Delete the selected object.</td>
</tr>
</tbody>
</table>

### 4) Draw

<table>
<thead>
<tr>
<th>Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyline</td>
<td>Add a polyline object to the editing layer. Adds the vertex by clicking the map view. Adds the end by double-clicking.</td>
</tr>
<tr>
<td>Spline</td>
<td>Add a spline object to the editing layer. Adds the vertex by clicking the map view. Adds the end by double-clicking.</td>
</tr>
<tr>
<td>Arc</td>
<td>Add an arc object to the editing layer. Creates the ellipse figure by first two times clicking, defines the start and end of the arb by next two times clicking.</td>
</tr>
<tr>
<td>Polygon</td>
<td>Add a polygon object to the editing layer. Adds the vertex by clicking the map view. Adds the end by double-clicking.</td>
</tr>
<tr>
<td>Rectangle</td>
<td>Add a rectangle object to the editing layer. Adds the corner of rectangle by first clicking, defines the opposing corner by next double clicking.</td>
</tr>
<tr>
<td>Ellipse</td>
<td>Add an ellipse object to the editing layer. Defines the rectangle angle which defines the ellipse size by first clicking, then defines the opposing corner by next double clicking, and draws the ellipse inside of it.</td>
</tr>
<tr>
<td>Load image</td>
<td>Add an image to the editing layer. Clicks the button, and selects the image file (*.bmp, *.tif) in displayed dialogue box. Draws the rectangle which defines the image area.</td>
</tr>
</tbody>
</table>

### 5) Snap Grid

<table>
<thead>
<tr>
<th>Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Right-Angle Snap</td>
<td>Draw the point horizontally or vertically for the previous point.</td>
</tr>
<tr>
<td>Grid Snap</td>
<td>Align the figure to the grid in case drawing and editing the object or editing the vertex.</td>
</tr>
<tr>
<td>Disable Snap</td>
<td>Release the snap function for the right angle, grid.</td>
</tr>
<tr>
<td>Snap Options</td>
<td>Set up the property of the snap grid.</td>
</tr>
</tbody>
</table>
### Snap Options

<table>
<thead>
<tr>
<th>Snap type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Snap Enable</td>
<td>Turn on/off the snap function.</td>
</tr>
<tr>
<td>Right-Angle Snap</td>
<td>Turn on/off the snapping the right angle.</td>
</tr>
<tr>
<td>Grid Snap</td>
<td>Turn on/off the snapping to the grid.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Grid Size</th>
<th>Input the size of grid cell. X is the lateral direction. Y is the vertical direction in meter.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Color</td>
<td>Input the color of the grid cell for snapping.</td>
</tr>
</tbody>
</table>

### 6) Layer Manager

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer Merger</td>
<td>Opens [Layer Merger] window.</td>
</tr>
<tr>
<td>Polygon cleaner</td>
<td>Opens [Polygon cleaner] window.</td>
</tr>
<tr>
<td>DEM cell converter</td>
<td>Opens [DEM cell converter] window.</td>
</tr>
<tr>
<td>Coordinate converter</td>
<td>Opens [coordinate converter] window.</td>
</tr>
</tbody>
</table>

### 7) Functions

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Send Data to UC-win/RRoad</td>
<td>Draws and defines the rectangle area for exporting to UC-win/RRoad. [Export to UC-win/RRoad] window will be opened.</td>
</tr>
<tr>
<td>3D View</td>
<td>Defines the rectangle area for displaying [3D view] window.</td>
</tr>
<tr>
<td>Area survey</td>
<td>If the polygon is drawn on the map view, the message information window will be displayed in right bottom, and the circumference and dimension of polygon will be displayed.</td>
</tr>
<tr>
<td>Distance Measure</td>
<td>If the polyline is drawn on the map view, the message information window will be displayed in right bottom, and the segment distance and total distance will be displayed.</td>
</tr>
</tbody>
</table>

### 8) Bookmarks

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add bookmark</td>
<td>Add the scene of current map view to the bookmark list.</td>
</tr>
<tr>
<td>Delete bookmark</td>
<td>Delete the scene of current bookmark from the bookmark list.</td>
</tr>
</tbody>
</table>
2. 2 Layer list
All layer lists included to the current GIS data map are displayed. The layer display in the map view such as adding/deleting the layer, managing the property is controlled in this list.

Layer list manager

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Label</th>
<th>Count</th>
<th>Set up the layer display.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light on</td>
<td>display</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Light off</td>
<td>not display</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Set up the layer in editing layer. Object can be created and updated within this editing layer.

**Light pencil**: Editable

**Dark pencil**: Read only

Select the availability of selection.

**Light arrow**: Selectable

**Dark arrow**: Unelectable

<table>
<thead>
<tr>
<th>Move the selected layer to next up level.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Move the selected layer to next down level.</td>
</tr>
<tr>
<td>Move the selected layer to top level of the map.</td>
</tr>
<tr>
<td>Move the selected layer to bottom of the map.</td>
</tr>
<tr>
<td>Make all layers displayed.</td>
</tr>
<tr>
<td>Make all layers non-displayed.</td>
</tr>
<tr>
<td>Make all layers editable.</td>
</tr>
<tr>
<td>Make all layers uneditable.</td>
</tr>
<tr>
<td>Update the map view.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Label</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name of the selected layer.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Type of the selected layer.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Label field of the selected layer.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Object number of the layer.</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Pop-up menu on the manager of layer list

<table>
<thead>
<tr>
<th>Layer Information...</th>
<th>Layer information</th>
<th>Display layer information window.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Layer(s) to Map..</td>
<td>Full Extent</td>
<td>Display selected layer in full window.</td>
</tr>
<tr>
<td>Create Layer..</td>
<td>Rendering</td>
<td>Display layer rendering window.</td>
</tr>
<tr>
<td>Copy Layer..</td>
<td>Table View</td>
<td>Display table view window at the bottom of main window.</td>
</tr>
<tr>
<td>Delete Layer..</td>
<td>Add layer(s) to Map</td>
<td>Add layer data file (<em>dxf,</em>.shp,.lem,.mem,.png,.tif,.png,.xml,.png,.tif,.txt). It is possible to import multiple files at a time.</td>
</tr>
<tr>
<td>Export Layer..</td>
<td>Create Layer Polygon : Create polygon type of layer.</td>
<td></td>
</tr>
<tr>
<td>Line Cleaner..</td>
<td>Polyline : Create polyline type of layer.</td>
<td></td>
</tr>
<tr>
<td>Polygon Cleaner..</td>
<td>Image : Create the image type of layer.</td>
<td></td>
</tr>
<tr>
<td>Set Map Coordinate System..</td>
<td>Delete Layer</td>
<td>Delete selected layer.</td>
</tr>
<tr>
<td>Coordinate Converter..</td>
<td>Export Layer</td>
<td>Export selected layer to the selected file.</td>
</tr>
<tr>
<td>DEM Cell Converter..</td>
<td>Layer Merger</td>
<td>Open the Layer Merger window.</td>
</tr>
<tr>
<td>Line Cleaner..</td>
<td>Layer Separator</td>
<td>Open the Layer Separator window.</td>
</tr>
<tr>
<td>Polygon Cleaner..</td>
<td>Line Cleaner</td>
<td>Open the Line Cleaner window</td>
</tr>
<tr>
<td>Set Map Coordinate System..</td>
<td>DEM Cell Converter</td>
<td>Open the Polygon Cleaner window</td>
</tr>
<tr>
<td>Coordinate Converter..</td>
<td>DEM Cell Converter</td>
<td>Set selected layer coordinate system</td>
</tr>
<tr>
<td>DEM Cell Converter..</td>
<td>Handle Cell Converter</td>
<td>Open coordinate converter window</td>
</tr>
</tbody>
</table>

3. Map view
The current map data is displayed by the view which has the different layer and the settings in the map view.

Pop-up menu
Right-clicking on Object information window.

Copy Image to Clipboard: Copy the current map view image to the clipboard.

Save to Image File: Save the current map view image in *.bmp or *.tif file. Hint: When the image is saved to a *.bmp file, a *.bpw file containing position information is saved in the same folder. Saving to a *.tif file will create a *.tfw file for position information.

Mini Information Window: Displays a little information window in the selected object.
Google Earth

Google Earth Plugin
It is possible to confirm the current map location with Google™ Earth plug-in.
Firstly, the window for selecting the current coordinate system is displayed (required to select).
The result will be displayed on browser after setting.

Google Maps
It is possible to confirm the current map location with Google™ map.
Firstly, the window for selecting the current coordinate system is displayed (required to select).
The result will be displayed on browser after setting.

4. Navigation map
The navigation map displays whole of the current data map.
Red rectangle indicates the range displayed on the current map view.
The display range of the map view can be controlled by removing the red rectangle.

Navigation map option window
Choose the layer for displaying on the navigation map from this window.
4. Layer

1. Layer information window
Set up the layer information on this window.

Name: Selected a layer name
Layer Type: Selected layer type
Field type: STRING or NUMERIC.
Count: Object number included in layer
Brush: Inside color of polygon
Transparent: Makes the inside of polygon transparent
Label: Select the layer field, it will be the label value of the layer.
Font: Set up the label font.
Visible: Set up the label visible/invisible

2. Layer rendering window
Set up the layer rendering information on this window.

Name: Current layer name
Field Name: Select the field for rendering.
Field Type: Select the field type from NUMERIC, STRING.
Type: Select the rendering method for the field.
Count: Set up the count number of the object for rendering.
Color: Select the rendering color.
Style: Select the rendering style.
3. Layer table view window
This is the table view information.

It displays the count number of name and object for selected layer.

4. Layer Separateor window
The layer is separated into some new layers based on the object type on this window.

Original Layer: Select the layer for separating.
Layer Type: Display the original layer type.
Polyline Type to: Pick out the polyline layer.
The layer is saved in setup name.
Polygon Type to: Pick up the polygon layer. The layer is saved in setup name.
Point Type to: Pick up the point layer. The layer is saved in setup name.

5. Sort function
1. Line cleaner window
The selected layer will be connected based on the condition setting.
On this window, connect the end of the individual line.

Target Layer:
Select the layer for connecting the line.
Connection Tolerance:
Set the range for connecting. The end distance of the line is smaller than this range, these points are connected.
Merge to One:
The line is merged into a line. Uncheck it not to merge the line.
2. Polygon cleaner window
On this window, connect the end of the polyline. The polyline is converted to polygon.

Target Layer:
Select the target layer from polyline to polygon.
End Points Connection Tolerance:
Set up the maximum interval between the connected two points. If the edge point distance of the polyline is smaller, this edge point will be connected and transferred to polygon.

6. Convert function
1. Coordinate Converter
On this window, set up the layer coordinate system and convert the layer coordinate system to the different one. There are some coordinate system in GIS data, so it is needed to integrate different coordinate system.

Layer List:
Display all layer lists in the current map. Select the layer for setting up the coordinate system. It is possible to select the multiple layers from the list.

Current Coordinate System:
Set up the current coordinate system of the selected layer.

New Coordinate System:
Set up the new coordinatesystem for selected layer.
Select the coordinate system related to the coordinate convert from green check command.

2. DEM cell converter window
On this window, create DEM cell which depends on the setting of the z value for target layer (or selected layer) and the cell size. The new DEM cell layer will be created as a layer of polygon type for rendering DEM cell efficiently.

Target Layer:
Select the target layer for exchanging to DEM cell layer.
Z Value Field Name:
Select the field for Z value (height value).
Cell Size:
Set up the cell size.
7. Export into UC-win/Road
On this window, connect GIS view data to UC-win/Road.

The table below indicates the relationship with UC-win/Road object.

<table>
<thead>
<tr>
<th>GIS layer type</th>
<th>UC-win/Road object</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyline type</td>
<td>Road, Terrain</td>
</tr>
<tr>
<td>Polygon type</td>
<td>Terrain, Building</td>
</tr>
<tr>
<td>Point type</td>
<td>Terrain</td>
</tr>
<tr>
<td>Image type, Captured image</td>
<td>Satellite image</td>
</tr>
</tbody>
</table>

UC-win/Road GIS view plug-in
Select [Import GIS view data] from File menu in UC-win/Road main window. [UC-win/Road connecting window] in GIS view is only available when the system is working.

If [UC-win/Road connection window] in GIS view is not opened, the empty message window is not displayed.
Open the [Export to UC-win/Road] window from GIS view user interface, the application will be mutually connected.
8. KML export window
On this window, export the selected layer to KML file.

Target Layer: Select the layer for exporting KML file.
Object Name Field: Select the field for exported object name.
Z Value Field: Select the field for height value (Z) for exported object.
Map Coordinate: Select the coordinate system of the selected layer.
Export Only Selected Object(s): Export the selected object of the target layer only. If you uncheck layer, all objects will be exported.
See the Result in Google Earth: Check the created KML file in Google Earth.
Latitude: Setup the offset of the latitude.
Longitude: Set up the offset of the longitude.

9. 3D view window
On this window, display the map view data in 3D view window.

Mouse operation
The mouse operation is available to use 3D view conveniently.
Move left: Turns around Z axis in clockwise.
Move right: Turns around Z axis in anti-clockwise.
Move up: Zooms in the center
Move down: Zooms out the center.

By holding Shift key
Rotate: 3D view by mouse cursor.

By holding Ctrl key
Move: 3D view in mouse cursor.

Fully Extent: Display fully 3Dmap.
Zoom in Towards the Center: Zoom in the center of view.
Zoom Out From the Center: Zoom out the center of view.
Vertical Exaggeration: Highlight the vertical direction by adjusting the slide bar.
Rotation Speed: Set up the rotation speed.
Map view: Switch to the map view.
Right click to display the popup menu.

- Show XYZ
- Show Sky
  Rotation
  Copy Image to Clipboard

**ShowXYZ**: Display the coordinate.

**Display Sky**: Display the sky image.

**Rotation**

**Copy Image to Clipboard**: Copy the current image to the clipboard.

**Object information window**

There are two tabs on the object information window, one displays the layer attribution, another displays the space information.

**Left**: Attribution information tab

**Right**: Space Information tab
Parking Model Plug-in

It is possible to import parking lot drawing data created by FORUM8's CAD system "UC-1 Parking Drawing System", which is based on parking regulation. The outer line of the parking lot, boundary line for each parking space, and road marking (such as an arrow indicating the driving direction) can be drawn accurately based on it. Since the imported drawing data, like buildings and cars, is treated as a model, it is possible to place them in an arbitrary position or rotate them.

**Parking Model Plug-in (ParkinglotPlugin.bpl)** is required.

- Open the form
  Go to File - Import - Load Parking Lot.

- List
  Models of parking lot are listed.
  
  Load
  Import a new parking lot data.

  Edit
  Open the Parking Lot Editor for the selected model.

  Delete
  Delete the selected model from the list.

- Parking Lot Editor
  Click Edit button or load the data of parking lot (*.pfr) to open this form.

- Parking Lot tab
  Input the name of the parking lot, and set up the texture on the ground.

  **Name**: Specify the name.
  **Height**: Set the height.
  **Texture**: Select a texture of ground from <<User Data Folder>>¥Textures¥Parkinglot
  **Repeat/Stretch**: The selected texture is enlarged to the whole (Stretch) or repeated in the specified size.
  **Width/Length/Keep the ratio**: Input the actual width and height of texture.
  Also check if keeping the ratio of Width/Height of texture.
■ **Lane tab**
Set the color and line width of passage line.

■ **Stall tab**
Set the color and line width for the parking mass for the vehicles.

■ **Stop tab**
Set the color and the height of the stop block set up in each parking mass.

■ **Arrow tab**
Set the color of road sign.
Handicapped parking space
Set the color and width of the stripes and internal parking signs for handicapped spaces.

The color of the internal parking signs and stripes are as followings.
・Stripe width: Width of the stripe lines of the boarding area.
・Symbol width: Width of the internal parking space with a symbol
OSM (OpenStreetMap) Plug-in

Open Street Map (OSM) data can be linked with UC-win/Road. OpenStreetMap (OSM) is a free and collaborative map of the whole world. Elements such as roads, tunnels, bridges and buildings can be displayed.

1. Export OSM data from website
Export map data from OSM website (extension: .osm).

2. Load OSM file from OSM Plugin
Select File – Import – Import OpenStreetMap Data... The OSM setting form is opened.

3. Customize parameters
Customize default parameters for roads such as Lanes, lane's width, Middle width, and other width as needed. Click Next.

4. Connect ways
Select roads to be connected. Selected roads are displayed in red and orange. Click Next.

5. Customize road parameters
Change the names of road. Selected road is displayed in color. Whether the road is Two-ways or one way is also displayed.

6. Output LandXML file
Click "Save to LandXML File" and specify a space and name. Without saving here, no data will be saved. Choose to save it to LandXML file or load it to UC-win/Road directly.

When it is saved to Land XML and loaded, the roads on OSM are reproduced.
OpenFlight Converter plug-in

1. Overview
The OpenFlight file format (PRESAGIS) is a file format to express 3D space. It has a database structure called scene graph and controls information such as 3D geometry, texture, position/direction, etc. for each node.
It is possible to divert 3D models created by OpenFlight format with this OpenFlight Converter plug-in.

The OpenFlight plug-in imports the "3D streetscape scene" created by OpenFlight format and exports UC-win/Road scene to OpenFlight format. The scene file of OpenFlight format exported by the other application can be imported to UC-win/Road.

2. Operation flow
1. Activate
Go to File-License Manager, activate "OpenFlight plug-in"

2. Export FLT file
Open a UC-win/Road project and go to "Export FLT file..." - "File" - "Export" - "File" - "Export" - "Export FLT file..." to open "Export FLT" form. Set up the items and click "Export" to export a UC-win/Road project data to the specified folder in FLT format.

3. Import FLT file
Go to "File" - "Import" - "Import FLT Model" to select a FLT file, and click Open to import the data into the UC-win/Road. The data is imported as a general model, so that it is possible to put the models on the VR space in the same way as the normal models.

4. Create the driving road
The imported OpenFlight model is recognized as 3D model in the UC-win/Road. Add a transparency road fitting with the road on the OpenFlight model and drive on it.

What’s LOD?
It switches the existence of models.
Check Active to use LOD (This option can be selected when Output option is each kind or each object.).
"Switch in" and "Switch out" means that the specified 3D model is displayed in the range from "Switch out" to "Switch in" and otherwise it is hidden outside (a unit is meter in case 3D model is created in meter unit). An error will occur if not "Switch in" < "Switch out". And if "Switch in" = "Switch out", the models are exported without LOD.
Export Scene Plug-in

All of models and terrain on 3D space of UC-win/Road can be file output in an arbitrary 3D model data format. Road generation function which is a feature of UC-win/Road generates a high accuracy and high quality 3D model, and it is possible to output this to outside and to reuse it.

Go to File - Export - Export 3DS... to open the setting form.

Output format
Specify the format of 3D model data file. The initial version supports only 3DS file format.

Option
Check a method of file output from the following.
1. Whole Objects: Output in one file.
2. Each Kind: Divide the file for each kind of output target and output it.
3. Each Object: Output one 3D data file for each one object of UC-win/Road.

Coordinate system
Check a coordinate of the model to be exported.
1. UC-win/Road local coordinate system
   Outputs models with the coordinate of the origin at the southwest end of the UC-win/Road project.
2. X-Y coordinate system
   Outputs models with the coordinate that is displayed as X-Y coordinate in the UC-win/Road. In the case of Japan in the project, it will be the plane rectangular coordinate system.
3. Object coordinate system
   This is available only when the "Each objects" is selected in the Output Option. It outputs models in the coordinate system of the object. So, the coordinate information for the instance is not included in the model. In addition, it is output only one file for information on objects that are uses in multiple instances.
   When this is selected, it is possible to use the "Output xml file for project information" in the Additional options.

Use offset
The input offset is added to the output coordinate system. This is not available when the object coordinate system is selected.

Filter Option
Specify whether to output the object for each kind in UC-win/Road. The checked one is a target of output.
It is possible to specify by kinds of road, railroad, intersection, road sign, road model, model, FBX scene, road marking, tree (2D), terrain, backdrop, powerline, and custom object.

Additional options
Output xml file for project information
This is available only when "Each Objects" is selected in the Output Option. XML file is output to the destination folder, where the coordinate information of models, scale information and the visible or non-visible information are included.

**Separate road side terrain**

This is available only when "Each Objects" is selected in the Output Option. The terrain grid, which including the road is divided into terrains grid in the range of less than or equal to the number of grids that set in the "Number of road side grid" and others, then saved into two 3DS files.

**Texture format**

Choose a format of texture files to be output with 3DS file, from bitmap file format (.bmp) and PNG file format (.png).

**Output Unit**

It is possible to output the files in meter, millimeter, inch, feet or yard.

"Export" button

Executes the output processing of 3D model data file on the specified condition.
FORUM8 Products (3D Model Creation)
1. Create models by UC-win/FRAME(3D)
The model data which is created in UC-win/FRAME(3D) can be loaded in UC-win/Road.

(1) Create a flame shape

[Model Generator]

Select a basic pattern, enter the arrangement shape size, and create automatically the node coordinates and elements.

The node coordinates and the elements can be cut and pasted using Excel spread sheet.

(2) Create the section shape

[Creating section shape 1] The superstructure section

Enter a geometrical shape using the section form template, and create a section. It is possible to generate various shapes by combining several forms.
(3) Define the section shape and set the color

Select an element and define the section

【After definition】

Save the 3ds file from "File" - "Export".

Hint: Depending on the model shape, define the flame and section form better.

For the arch model below, define a section focusing on A component instead of B.
Create the following section and define it.

When loading a model in UC-win/Road, it is separated into layers of components. Therefore, do not define unnecessary components. When an unnecessary layer is created, it can be integrated using the other modeling tool.

When the section shifts from the flame position, the differences in the section can be adjusted by setting the position of the flame line (optional).

Edit/inspect section properties.
For the arch model below, set a shell and section as follows.

The model after transformation can be output.

Click after frame calculation, and then click.

Change the displacement in scale setting form.

The model before transformation can be hidden by clicking Display wireframe. And the element...
Click Display solid model to export the model before transformation. Then load it to UC-win/Road.

2. Creating models in UC-1 series

At present, the compatible FORUM8 UC-1 products for 3 dimensional file output are followings:
- Pier design
- Abutment design
- Box type abutments design calculation
- RC substructure design calculation
- Pile foundation design
- Foundation design calculation
- Temporary sheathing work design
- Temporary piled jetty design
- Double-wall cofferdam design calculation
- Box culvert design
- Retaining wall design
- Slope stability analysis
- Manhole design calculation
- 3D landslip slope stability analysis (LEM)
- UC-Draw Tools
3D model creating software

1. "Create a 3D model with AutoCAD"

(1) When using a DXF file which is created by CAD

1. Click "File" - "Open", and select DXF file from "File type".

   ![Image of DXF file open dialog]

2. Click "Display" → "Viewport" → "4 Viewport", and set the display screen.
   Set each window to active, and click "Settings" → "3D View" → each view (the side, the front).

   *As the screen which loaded 2D DXF is set to a plan figure, change the viewpoint to the side, front, and 3D (the southwestern isometric-drawing etc.)

3. Move / Copy each figure to each viewpoint.
   1) Select a figure to move (copy), and click "Edit" → "Copy" of the drop-down menu.
   2) Select the position to move (copy), active (click with a mouse), and select "Edit" → "Paste" of the drop-down menu.

   Arrange to the optional position.

   *When moving (copying) the figure to each viewpoint, "copy" command of Auto-CAD cannot be used.

4. Delete the unnecessary figures and lines from the plan figure, and create the intersection point in 3D view by using "move command".

   ![Image of 3D model creation]

-453-
5. Click “Create” → “Surface” → “3D mesh” or “3D face” and define each 3D face. “Display” → “Rendering” is used for 3D display confirmation.

*Refer to the manual of Auto-CAD for the operation of “3D mesh” and “3D face”.

6. It is possible to create 3D through the use of a polyline and depth specification if the diagram is level with some surface.

(2) Create using AutoCAD 3D creating tool

1) Edit the polyline which closed the diagram
2) Specify depth and press angle by clicking “Create”→“Solid”→“Press”.

With Auto-CAD, 3D diagram command of the solidness and column etc. is supported.
In addition, pulling out only "summation", "difference" and "cross" part of 3D figures is possible.

(3) Save to 3DS file

Note
The unit of UC-win/Road is "m". When creating by "mm" with Auto-CAD, reduce it to 1/1000 before converting (save to 3DS format), or reduce to 1/1000 after reading with UC-win/Road.

Select the range to convert to 3DS and specify the file type: "3DS" through "File" →"Output", and save.
2. "Create 3D models with Autodesk VIZ"

(1) When using DXF file created by CAD

① When using DXF file created by CAD, select DXF file from "File type" after clicking "File" → "Load"

"Load DXF file" dialog is displayed, but click "OK" button as the default.

② Select "Object" from the tab panel or create using the menu from the list.
(2) Paste the texture

1. When "Rendering"→"Material Editor" is selected, the dialog box of the left figure is displayed. Many spheres can be seen and the sphere within the white frame is in an active state.

2. Refer to the folder which the texture data is saved with Windows Explorer. Drag the texture data to be pasted and drop it on the active sphere. The texture data which was dropped on the sphere is displayed.

3. Click "Allocate the material to the choice" of the material editor, and close "Material Editor" dialog box.

(3) Save to 3DS file

Specify the file type to "3DS" by clicking "File" → "Output" and save.

Uncheck "Save Max texture coordinate".

To load the model with pasted texture from UC-win/Road, it is necessary that the saved model and texture data are saved in the same folder. When saving the model with pasted texture, the maximum length of name of the texture data is 8 characters as it is essential that texture data must be in "8.3 format".

-456-
3. "Create 3D models with Shade"

(1) Create a model

1. Screen settings

Shortcut keys

- Ctrl+Q... Shading
- Space... Screen scrolling
- Alt+]... Expand
- Alt+ [...]... Reduce
- Ctrl+Click... Basic setting
- Ctrl+Shift+Click in the field... Copy
- Z+X+Click the control point... Delete
- Ctrl+Shift+Drag the point... Tangent handle

1. Use the tool box in shape creation.

**PEN Tool**

- Closed line profile
- Sphere

**Text Effectors**
SOLID Tool Setting of color

(2) Save to 3DS file

With UC-win/Road, settings for color and display/ hide functions are done for every layer. When creating a model with Shade, it is necessary to set a layer in the following procedure. (In direct conversion to 3DS, the layer is created for every created element)

1. Select every parts to classify, and export by clicking “File” → “Export” → “3ds”.
2. Import the parts which has been output in the new creation screen.
3. Select “File” → “Import” → “3ds”.
4. Select each part perform polygon conversion. Click “Modify” → “Convert.”
5. Select all and re-export by clicking “File” → “Export” → “3ds”

Click here
Select a color after the pallet is displayed.
(3) When using 3DS file which has already been created

Select “File” → “Import”, and choose a file to upload. When the display after loading is small, select “Special” → “Fit to the drawing”.
Create MD3 model
Using some applications, it is possible to get the MD3 character file.
Here, RokkakuDaiou and MilkShape3D are used.
Note: it is also possible to obtain other application to create the MD3 character file.

Software
RokkakuDaiou Super5.5
Development / Sales SHUSAKU Inc.

MilkShape3D (shareware)
http://chumbalum.swissquake.ch/ms3d/

1. Human body 3D model creation – Use RokkakuDaiou
   1) [File] → [new] → [human body creation mode]
   2) Specify images to create in [load image]
   3) Model will be created by setting up according to wizard.

4) Save the created model
   [file] → [write] → [LightWave3D] → [LWO2]

Check [create with few polygons]
2. Create skeleton (framework) using MilkShape3D

1) Read created model by RokkakuDaiou
   [File] → [Import] → [LightWave LWO]

2) Create Skeleton (framework)

   - Click Joint and setup joints of model
   - Select Joint tab from tab menu and select the last point of joint.
   - Check Show Skeleton

Perform settings of Joint in the following order.
1. Lower back → center of the back → chest → neck → crown of the head
2. Chest → shoulder → elbow → waist → finger tip
3. Creating of the group

① Divide the 3D human body model to 3 groups: "h_head", "u_torso", and "l_legs".

② Create Tag which links the 3 of the groups created.
Create Group name of the tag which links "head" and "upper" "tag_head",
Create Group name of the tag which links "upper" and "lower" "tag_torso".

③ Paste the texture.
1. Select "Material" tab.
2. Click "Load Texture" and load "File name Lwo1.jpg" (created when saving with Rokkaku Daioh)
3. Click "Assign" button.

4. Creating of the animation

1. Click "Anim" button.
2. Set the action with "Rotate" button
3. Set the key frame with [Ctrl] + [k]
5. Save to MD3 format
   ① Create qc file.
   1. Select "h_head" of "Group" tab, and click "Select" button.
   2. Select "quake III AreanaGenerate Control File" from the tool menu, and save.
      File name: "h_head" → "head.qc", "u_torso" → "upper.qc", "l_legs" → "lower.qc"

   ② Save to MD3 format
   1. Select a group to save from "Groups" tab, click "Select" button.
   2. Select "File" → "Export" → "MD3", and select "quake III Areana MD3".
      File name: "h_head" → "head.md3", "u_torso" → "upper.md3", "l_legs" → "lower.md3"

6. Creation of other related files
   ① Creating of skin file.
      It is the file to define texture of each part, and is uniquely created with an editor.
      (Correct with a memo book after copying from the sample)

   ② Creating of Animation.Cfg
      Describe the action of the animation defined in the model.
      Refer to the sample and make necessary corrections with the editor.

7. Subscribe to UC-win/Road
   ① Create an icon file by cutting off the face of the person.
      Size: 64 pixel x 64 pixel Type: .tga
      File Name: icon_default.Tga

   ② Change MD3 to PK3.
      1. Copy the necessary file to ¥models¥players¥<the optional folder>
      2. Compress the entire above mentioned folder by zip format.
      3. Change the extension of the zip format file which is compressed to ".pk3" from ".zip".
      4. Save the created .pk3 file to <UC-win/Road installed folder>¥characters.
      5. Start up Road, and load the MD3 data

The following file is always necessary for ¥models¥players¥<optional folder>
- MD3 model data: head.md3, upper.md3, lower.md3
- Defined data of the animation: animation.cfg
- skin file for each parts: head_default.skin, upper_default.skin, lower_default.skin
- icon file: icon_default.tga

-463-
Texture Creating And Editing

1. Editing with Windows Paint.

1. Go to the Windows Start Menu - Accessories - Paint, and open the image file to be edited. Confirm the image size by right clicking on it and select Properties.

2. Change size or rotate the picture.
2. "Edit the image with Photoshop"

(1) New creation

1. Specify the size from "File" - "New".

2. Draw or edit with the following tools.

[Tool Box]

- Choose: Choose an area.
- Move: Move a range.
- Free choice: Specify a range freely with free hand.
- Auto-select tool: Choose the range (the red flower etc.) which a color is uniformly put to even if it didn't trace out lines.
- Clipping tool: Remove the part of the image to create the focus and the emphasis part of the composition.
- Correcting brush tool: Correct the incomplete part and match it with the image around.
- Brush: Paint with brush.
- Stamp: Create the copy of the image.
- Eraser: Restore the part of the image.
- Fill: Fill in color.
- Shade: Shade a cleared part and an edge of image.
- Text: Enter a text.
- Pen tool: Create or edit the line and the shape of the straight line, the curve or the free-form.
- Drawing tool: Draw a rectangle, a circle, a line etc.
- Syringe: Copy the color.
- View: Move the image.
(2) Create a sign

It is possible to process in the penetration by making a background color black.

1. Use "Fill", and set the background color black.

2. Click the drawing tool to open the menu bar. Choose "Oval tool" (choose "white" for the palette). Press shift key and draw a circle.
3. Click "Creating the opaque part of the layer" from the dialog.

4. Go to "Choosing range" - "Change the choice range" - "Reduce the choice range", and create a layer from "Layer" - "New Layer".

5. Choose "Red", Click "Fill".
6. Click "Rectangle tool" from the menu bar, and draw a rectangle.

(3) Change the color of whole texture

Go to "Image" - "Tone of color revision" - "Variation".
Practical
Point cloud modeling plugin
The point cloud modeling plug-in is the plug-in that import and utilize the millions of point data counted in the field by 3D laser scanner (High accurate GPS mobile counting system) into UC-win/Road. Furthermore, it is possible to import point cloud data collected from a mobile 3D laser scanner, MMS (Mobile Mapping System), to UC-win/Road to create road alignments (horizontal curve, vertical curve, road section curve) or extract and edit road section from an arbitrary point.

- Import more than 50 million point cloud data (Note: the amount of point cloud data that is importable depends on the PC hardware capacity)
- Extract coordinates and color information from the point cloud data which are recorded in text or CSV format

*Note: Point cloud data is not saved with the UC-win/Road data, therefore, the point cloud data is not included in the UC-win/Road data file (*.RD). However always keep the UC-win/Road data file and the point cloud data file together in the same relative folder tree.

If there is the associated point cloud file when Road file is opened, load can be selected.

1. Point cloud modeling
Point modeling plug-in screen is displayed by selecting Point Cloud Ribbon-Load data.
Settings

File extension: Specify the file extension of the point cloud data file that is to be imported. Use semicolons to separate the file extensions, when importing multiple extensions.

Default Scanning Method: Specify "Normal" or "MMS" for the default form.

Coordinate System Type: Specify "Left Handed" or "Right Handed" for coordinate system type.

North – South: Specify the nth "column" in which the value of the coordinates along the North-South axis exists.

East – West: Specify the nth "column" in which the value of the coordinates along the East-West axis exists.

Height: Specify the nth "column" in which the height value exists.

Color (Red): Specify the nth "column" in which the R (Red) value of the color exists.

Color (Green): Specify the nth "column" in which the G (Green) value of the color exists.

Color (Blue): Specify the nth "column" in which the B (Blue) value of the color exists.

Separator: Specifies delimiter of data unit. Space() / Comma(,)/ Period(.) / tab(¥)

Decimal Point: Specify the decimal-point character. Space()／comma(,)／period(.)／tab(¥)]

Color Strength Parameter:
Sets the Min Value and Max Value of color strength for the read target data file of point cloud. The default is set to 0 for the min. value and 255 to the max.
It is clamped within the range set here when the value of the data file is outside the range.

Point’s Scale Factor (Zooming ratio):
The coefficient value set here is multiplied to the value in the point cloud data file. Default value is 1.
*When the coordinate value of the measured point is a value in millimeter, for instance, if 0.001 is set here, the value is convertible to the standard unit, meter, which is the standard unit in UC-win/Road.

Underflow prevention processing on loading.
The display might fall into disorder when the point is displayed in the VR space according to the coordinate value in the data file of the point cloud. This is a problem that occurs when the coordinate value is a very big value. In this case, it might be improved when checked the Underflow prevention processing on loading, and read the point cloud data. This option controls the underflow generation by processing coordinate data by relative coordinates when the point cloud data file is read and prevents the display from falling into disorder.
However, it becomes impossible to acquire the location information correctly and the default position of the point group is not set correctly. So edit the coordinates of the origin point on the main form of point cloud and arrange the data at a correct position.
● File format
This is an example of the file format for the point cloud data.
Data is divided by comma (,), and decimal point character is period (.).

In the point cloud data file, a line shows the information of a point. This plug-in extracts the below data in that csv data:
- x coordinate
- z coordinate
- y coordinate (height)
- r value (red constituent)
- g value (green constituent)
- b value (blue constituent)

The information and the order depend on the type of 3D scanner.

■ Open Files
Import the file recording the point cloud information. It's possible to select the several point cloud file and import them at once.

Setting of white cell can be changed by placing the cursor over and pressing Enter key.

Name: Display the name of the point cloud data. It can be renamed by pressing Enter.
View: Choose whether to display the point cloud model in the main view. Uncheck the box to hide the model.
Move: Lock or unlock the model using the padlock icon. When locked, the positions and the angles of the model cannot be edited in the main view.

- [Ctrl] + Mouse dragging: Move point cloud model horizontally
- [Alt] + Mouse dragging: Move point cloud model vertically
- [Ctrl] + [Alt] + Shift + Mouse dragging: Rotate point cloud model

It is possible to unlock more than one data to adjust the position of them at once.

File: Displays the full path and file name of the point cloud data.
Number of points: Indicates the number of points imported.
Type: Indicate the scanning method. Select a scanning method from the settings form opened from Settings button in advance. Normal indicates that the data was collected using a fixed equipment while MMS indicates that a mobile device was used.

Origin X, Y, Z: Display the coordinates of the point cloud data in meters. These coordinates are not based on the coordinate system within the UC-win/Road environment, but it is based on a coordinate system where the center of the point cloud is assumed as 0. The unit is m.

Angle: Indicate the angle of the point cloud model. The unit is Degree.

Delete: Delete the selected point cloud data.

● Coordinate of Point Cloud data

This plug-in adjusts the center position to move onto the center of the terrain map for x coordinate (east-west) and z coordinate (north-south). But for the y coordinate (Height), the height of the center position should be 0 m. So it is required to move the data to specified origin x,y,z by the Point Cloud Modeling Plug-in form. And the angle of data should be also adjusted in Angle.

*When loading the point cloud data, the Origin Y of the model is 0m. So if the current terrain is higher than the modeling data, the modeling data is under the terrain. Then, even though the eye position is moved by shown later, there is sometimes not the case it is possible to confirm the model. In that case, set the Origin Y by which the model should be on the terrain.

*If one cloud data is divided into the more than one data, load them at once. If loading one by one there is sometimes that those position are not matched with each other because the position is calculated by the file one by one.

■ Center Line

For point cloud data by MMS (Mobile Mapping System), there is a function to help generate road shapes and sections. It enables to generate high-precision roads in a short time from the point cloud data.

Click 「New center line」 button to show information such as a name of an analyzed center lines. White cells can be changed by meeting the cursor on the cell and pressing the Enter key

Name: The name of Center line. It is possible to rename it.

Point Cloud: Name of point cloud data.

Road:

If a road has been created for the selected center line, the road name will be displayed here. However, if no road is associated with the center line, <Undefined> will be displayed here.

Reverse:

If an analysis of the road of the point cloud data has been carried out using the New Center Line button, the start and the end point will be defined automatically. The horizontal line will need to be defined in the same direction as of the direction used in the analysis. If the direction was defined wrongly, checking the Reverse box will correct the direction of the vertical line.
Offset: Input the offset in meters, if the start position of the road is different from that of the center line. The unit is m.
Length: Display the length of the center line.
Color: Select the color of the center line when shown on the main form and in the plan view. Press the Enter key to edit the color.
Delete: Click on the X to delete the selected point cloud data.

Slide bar: Move the focus point from start point to end point on the center line. For the focus point, the shape of road section (scanning plane) is shown by line with yellow color.

(1) Definition of the Center Line
Double-click the point cloud data generating a road to select the point cloud data of the road and click to the button, New Center Line. It analyzes the characteristics of MMS point cloud data and estimates the following:
- The possible position of the road
- The road section shapes from the start of the road to the end of the road (scanning plane)

If the analysis is successful, the analyzed road information is added to the table. However, if there is noise, lacking data in the loaded data or if the data format is not MMS, it may fail to estimate the position of the road and the road section properly. In these cases, create the road manually.

First, using the slide bar shown in the red box (see below image), move the camera to the road in question.
- The left end of the bar • • • represents the beginning of the road
- The right end of the bar • • • represents the end of the road

By moving the bar, the yellow line (scanned plane) in the main form is displayed. Sometimes, the plane is not shown properly due to the presence of noise or obstructions such as cars in the scanned data. When this happens, please move the bar to the position where the yellow line can be seen. If the yellow line is still not visible, moving the camera position in the main form may make it visible.
If the plane displays the possible road section properly, move the mouse cursor onto the line in the scanned plane to set the center line. In the below example, center of the white line for one way of the road is set. By moving the mouse cursor to the line on the plane, the color of the point changes to pink.

(2) Adjust the center line
Having estimated the center line, the analysis of the point cloud data is done and the position of the dot sequence of the center line is estimated. The center line is shown in green initially, so using the slide bar in the Center line tab, check the position to adjust.

Sometimes, the center line is placed at the wrong position. If this occurs, first, click on the center line to adjust the position, so the plane (the below yellow line) is shown there. Then click where the mouse cursor is on the line of the plane to estimate the center line again and this moves the new center line there.

Double-click where the mouse cursor is on the center line to set the reference point. If the reference point is set (as shown in the below image) the previous or the next line which passes through the point is set. Please adjust the center line with the reference point added. The reference point is highlighted in red.
Right click to bring up the popup menu, where the reference points that were placed previously can be deleted, and where the selected points can be moved to the previous or the next reference point. And the reference point is shown highlighted on the main form.

(3) Defining the horizontal line
Open the Plan View form and define the horizontal line. If the center line was defined properly in the previous 4 steps, the center line will be shown in the Plan View form. Zoom in and select Add -> Start the road from the right-click popup menu and define the horizontal line tracing the center line.

(4) The definition of the vertical line
First, click the icon (shown below) and open the editor form. Then, select the center line corresponding to the current road, so that the center line (point cloud data) is associated to the current road. This can also be done in the Point Cloud modeling plug-in form.
Then, define the vertical line by tracing the center line. Adjust the line adding the turning points arbitrarily. And add bridges and tunnels area as required.

(5) The definition of the road section
When defining the road section, first, using the slide bar in the Center Line tab in the Cloud point Modeling plug-in, move the plane to define the road section and to add the new road section and open the road section editor.

Road Surface

Name: Display the name of road section. It can be changed by pressing Enter key.
Edit: Open the Section Editor.
**Center Line**: Display the name of the center line associated to the point cloud data.

**Road**: Display the name of the road that is associated to the point cloud data. If no road is associated, it is Undefined.

**Position**: Display the position the road section from the start of the road in meters.

**Delete**: Delete the selected center line.

In the Road Section Editor, the point cloud data is shown in the left top drawing area. Create the section with reference to the data.

The button above can change the position for the display of point cloud.
**Render Option**

![Render Option Image]

**Point Size:** The size of point can be changed.

Size: 5  ![Size 5 Image]  Size: 20  ![Size 20 Image]

**Parameter:** Set the parameters and choose the appropriate size and brightness of the points according to the distance of eye position. Note: These parameters are used by the OpenGL API. The changes may cause unstable results.

**Lightning:** Set whether the lighting is taken into consideration when drawing the points.

---

### 2. Measure Tool

Go to the ribbon, Point Cloud – Measure to access the Measure Tool Window. This window allows users to measure the distance between 2 point cloud data groups.

![Measure Tool Image]

Select any 「Measure n」 and choose 2 point cloud data positions from the Main Navigational Panel. The distance between 2 points are automatically measured as width, height, and length.
**VR-Cloud® Plug-in**

VR-Cloud® has been integrated on UC-win/Road Ver. 11. VR-Cloud® Plug-in which is installed as default on UC-win/Road Advanced, Driving Sim and Ultimate version. It is possible to show the project under consideration to interested party in faraway place. VR data can also be shown on smart phones and tablets with Android, so that it is possible to check the scenery of after design while walking the site actually.

1. **What is VR-Cloud® Plug-in?**

VR-Cloud® Plug-in activates UC-win/Road on server, so that it is possible to operate it remotely in VR space on Web browser. This is not just video streaming and it can provide interactive real-time VR. A variety of operation functions of UC-win/Road such as walking, traveling, driving and flying as well as comparative study of before and after design, and running of scenarios. And communication tool feature between clients are equipped as default to Advanced or greater version. It is possible to write comments in VR space, communicate on 3D bulletin board, and paste photos.

2. **Flow of VR-Cloud® server setting**

1. Cloud server construction

   It is possible to use the own server or FORUM8 servers. Please ask for details.

2. Authenticate VR-Cloud® Plug-in

   From "File" Ribbon - "License Manager" and check VR-Cloud® Plugin, VR-Cloud® Collaboration Plugin, and VR-Cloud® Script Plugin to open "Server" Ribbon and "A3S" menu. For UC-win/Road Ver. 11 Advanced or greater version, install the plugins. Click "Options" icon to open A3S Options window.

3. a3s Server setting

   Set up TCP server on "Server" setting form. Input port number, maximum number of clients and password protection. A port number must be used only for a3s server. And set up the"Audio-video". As for audio settings, it is possible to switch enable/disable audio and set bit rate value.
4. General setting of VR-Cloud® server

Set up the authorization. Check actions that can be performed by clients.

Check “Start the environment” or “Start the traffic” to automatically start the environment and/or the traffic.

On the client setting tab, select a user interface which will be shown to other users.

When VR-Cloud® Scripting plugin is loaded, it is possible to use custom scripts on the client machines. It does not require high level programming skills and can be done with a simple text editor. It is possible to set some constants used by the archived scripts and the custom scripts.

5. VR-Cloud® User content / conference setting

VR-Cloud® supports the creation, display and edition of collaborative contents, which are useful to share information at a specific point in the 3D virtual world. These contents can be fully customized by creating a new script (see above). To use the collaborative features, Install the a3S Collaboration plugin. These features are supported by only VR-Cloud® version 6.1.

User content option

By default the following contents are available.

- Discussions: Text discussion can be created by any user.
- Opinions: Express own opinions by rating a 3D position.
- Annotations: Show explanations at a specific point, which is composed of various shapes and lines
- Photos: Upload and share photos.
All collaborative contents can be commented, and users can reply to it.

VR-Cloud® user contents setting

VR-Cloud® annotation feature

■ Conference

It is possible to chat via text messages by organizing in conferences in the virtual world.

Install the VR-Cloud® Collaboration plugin. It is supported by only VR-Cloud® version 6.0.

It is possible to specify an administrator password to start a conference. Moreover the conference report can be exported to local or FTP in HTML using a default CSS. If checking FTP server, specify the server information.

VR-Cloud® conference setting

6. Enable a3s server

Click "Enable Server" in Server Ribbon to launch a3s server and start cloud service. 3D images are displayed. Click "Disable Server" to disable. Settings can be changed after disabling it.
Xpswmm Plug-in

It is possible to exchange data between flood analysis software Xpswmm and UC-win/Road. Importing the result of flood analysis, drawing the surface flood, drainage system, pipe work are available as well as simulation expression in 3D-VR based on time.

Activate the license by checking "Use" the plug-in in the License Manager from File – License Manager. Operation is not available when there is no project or no imported data.

1. Xpswmm ribbon

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import xpswmm data</td>
<td>Open the &quot;Import xpswmm data&quot; screen. When the project is created or opened, it becomes effective. It becomes invalid when there is no project.</td>
</tr>
<tr>
<td>&quot;Prev&quot; &quot;Next&quot; Button</td>
<td>Load the previous / next analytical result. After &quot;Temporary stop&quot; button is pushed, water data import settlement and the toolbar become effective.</td>
</tr>
<tr>
<td>&quot;Play&quot; button.</td>
<td>The simulation starts. Check Loop to repeat the simulation.</td>
</tr>
<tr>
<td>&quot;Pause&quot; Button</td>
<td>The simulation stops temporarily.</td>
</tr>
<tr>
<td>&quot;Stop&quot; Button</td>
<td>The simulation stops, and the state parameter is reset.</td>
</tr>
<tr>
<td>&quot;Setting at start of the simulation time&quot;</td>
<td>Specify the start time of the simulation.</td>
</tr>
<tr>
<td>&quot;Setting at simulation end time&quot;</td>
<td>Specify the finish time of the simulation.</td>
</tr>
<tr>
<td>&quot;Simulation speed setting&quot;</td>
<td>Specify the speed of the simulation.</td>
</tr>
<tr>
<td>Edit flooding option</td>
<td>Open to the option of flood water. It becomes effective with data import settlement.</td>
</tr>
<tr>
<td>&quot;Pipework drawing option&quot;</td>
<td>Open the pipework Display Options screen.</td>
</tr>
</tbody>
</table>
2. Import Xpswmm data

On this screen it converts the value in the horizontal direction of the Xpswmm terrain data into the value for the direction of east to west or north to south with the coordinate adjustment.

(Refer to "Road plan" of UC-win Road for help.)

| Surface Fluid | Specify the surface flood analysis result file. |
| Pipework | Specify the pipework file. Make the pipework file of Xpswmm by using the xpx file export function with Xpswmm software. |
| Pipework Fluid | Specify the jurisdiction style analysis result file, *.csv. |
| Direction of coordinates | Specify the direction of coordinates and specify whether to change the value of the data for the direction of east to west and north to south. |
| Offset input | Specify the place where the simulation is reproduced. It is necessary to specify the place where the simulation is reproduced when the geographical features of xpswmm are not corresponding to the rd file of UC-win Road. |
| Coordinate conversion | Set parameters as needed. |
| Open | Parameter saved file * only for the Xpswmm plug-in, Open the xps file, and set each of the parameters. |
| Save | This parameter is only for the Xpswmm plug-in. Save it in xps. |
| OK | Back to the main screen of UC-win Road, with the data import. |
| Cancel | Back to the main screen of UC-win Road without the data import. |
3. Hazard water level display option

Set up the surface of the water, the flow velocity arrow, the time character, and the surface of the water layer.

**Water Surface**
- Display On/Off: Draw the water surface or not.
- Height Offset: The water surface offset in height direction

**Mesh Display of water surface**:
- Display On/Off: Display the water surface or not.
- Color: Specify the color of line of mesh edge.
- Size: Specify the size of line of mesh edge.

**Velocity / Arrow**
- Display On/Off: Draw the arrow for the fluid velocity
- Color: Click Option to specify the color of the arrow for the velocity.
- Length Coefficient: Input the length coefficient to multiply when calculating the arrow length by the flood velocity.
- Height Offset: Input the offset value of the arrow of fluid in the height direction.
- Line Width: Input the line size for the arrow of fluid
- Head Size: Input the length of the head of the arrow.

**Time String**
- Display On/Off: Display the time strings or not.
- Color: Specify the color of the time strings.

**Position Adjust**:
- Select: Select the position where to display the time string from Left Top, Right Top, Left Bottom and Right Bottom.
- Fine Tuning: Input the drawing position of time strings set by the horizontal and vertical positions of the screen.

**Draw Option**
Contour: Display the result of the analysis by contour.
Reflection and Refraction: Display it in VR considering reflection and refraction.
Reflection and Refraction:
• Water Color: Set the color of water by mixing with the color of water affected by the reflection of sky.
• Water Color Ratio: Set the percentage of the color of water to show within the range of 0.0 and 1.0 when a tone of color is added to the water as an effect of reflection. If it is 0.0, the color of water affected by reflection will not be mixed and the entire part affected by reflection will be entirely reflected. Note that the part affected by refraction will not change. If it is 1.0, the result of visualization of reflection is not added, and the percentage of color affected by refraction will be greater.

Randomize Small Waves Direction:
Set whether the direction of small waves (ripple) is made randomly or not. Check to move the small waves in random direction based on the value of "Small Waves Random Ratio" below. Remove the check to move all waves in the direction of the flow.
• Small Waves Random Ratio: Set the percentage of small waves that is moving in random direction within the range of 0 and 1. If it is 0, all the small waves move in the direction of the flow. If it is 1, all the small waves move in random direction.
• Ratio of Noise Texture: Set the percentage of noise to add to the small waves within the range of 0 and 1. If it is 0, noise will not be added at all and the small waves are expressed by their shape. If it is 1, the shape will not be taken into consideration at all, and small waves will be expressed solely by noise. Adding some noise makes small waves look more real. The default is 0.3.
• Add Small Wave's Brightness: Set whether to add brightness and shininess of water to the high point of small waves or not.

Add Large Wave:
Set whether to add a little shape to the wave at shoreline or not, in addition to the height of the water from the data of xpswmm.
• Wave Water Depth: Set the maximum value of the water depth of the above-mentioned wave in meters. It can be seen at the position where the water depth is less than the value set here.
• Large Wave's Height: Set the wave's height with m.
• Wave Frequency Speed: Set wave frequency speed. Value is within the range of 0 and 10 and it is expressed as the number of times to multiply the default speed by.
• Splash Water Depth: Set the maximum water depth in meters where the expression of the splashing water is added. The water splashing can be seen at the position where the water depth is less than the value set here.
• Splash Whiteness: Sets the color of splashing water within the range of 0 and 10. If it is 0, the expression of splashing water is not added. The bigger value, the whiter the color of splashing water.

Water Level Layer:
Divides the result of the water's flood surface analysis into colored layers by water surface level specifies the color.

"Open": Open a parameter file for the xpswmm plugin (*.xps) and set parameters in this form.
"Save": Save the parameters to a parameter file for the xpswmm plugin (*.xps).

*The rate of reflection on the water surface and the light's refraction is calculated using the Fresnel's equation and shows a situation that has a great amount of reflection due to the water being shallow, conversely, the 3D water is shown to be much darker in color if the angle of the visual line is deeper.

4. Linkage with Tsunami Analysis Software
UC-win/Road can visualize tsunami analysis result using xpswmm plug-in.
To simulate tsunami analysis result, click the “Surface Fluid” button in the Import xpswmm data screen opened from the menu, File – Import – Import xpswmm data, and select the file containing the analysis result. Click OK button to load xpswmm analysis results.
Before starting the Tsunami simulation, set the start time and the finish time of the simulation in the "Start time" and "Finish time" drop down menu on the ribbon. Set it that was set when exporting the result of analysis in xpswmm.

Then, set the time interval of the simulation using the "Water surface display speed" drop down on the toolbar. Here, set the time interval of the analyzed data in seconds. In the case of tsunami simulation, set the actual time interval of the analyzed data in order to do simulation in real time.

Lastly, click the "Environment" button on the ribbon, Home – Simulation in UC-win/Road and "Play" button in xpswmm plug-in to start the simulation based on the result of analysis by xpswmm.

The result of the simulation is expressed on the UC-win/Road as seen below.

Smooth shoreline  Waves on water

*The Visualization of the Level of Damage to Buildings (Under Development)*

The drag acting on buildings by the tsunami is calculated and a function that visualizes the degree of building damage is implemented.

The degree to which each building is damaged is calculated from the ‘suction power’ parameter of energy (in kJ) set for each drag of the tsunami and the target building. This is shown within the software through the changing of its color depending on the tsunami's power, below. Moreover, though the damages caused on buildings are commonly considered in terms of the water
height, this plug-in calculates the damages on buildings using the information obtained from xpswmm (the depth, speed, waterpower calculated from the building shape and resistance to the waterpower), thus, making the result more accurate.

・Example of contour color based on the levels of damages on buildings

<table>
<thead>
<tr>
<th>Minimum (kJ)</th>
<th>Maximum (kJ)</th>
<th>Color</th>
<th>Sample color</th>
<th>Level of energy and breaking</th>
<th>Height of wave (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.50</td>
<td>Blue</td>
<td>Blue</td>
<td>No-swimming warning issued</td>
<td>0.00～0.25m</td>
</tr>
<tr>
<td>0.50</td>
<td>1.25</td>
<td>Green</td>
<td>Green</td>
<td>Adults unable to stand</td>
<td>0.25～0.50m</td>
</tr>
<tr>
<td>1.25</td>
<td>2.50</td>
<td>Yellow</td>
<td>Yellow</td>
<td>Wooden structures partly damaged</td>
<td>0.50～0.70m</td>
</tr>
<tr>
<td>2.50</td>
<td>5.00</td>
<td>Orange</td>
<td>Orange</td>
<td>Wooden structures partly damaged</td>
<td>0.70～1.00m</td>
</tr>
<tr>
<td>5.00</td>
<td>10.00</td>
<td>Dark orange</td>
<td>Dark orange</td>
<td>Wooden structures partly damaged</td>
<td>1.00～1.50m</td>
</tr>
<tr>
<td>10.00</td>
<td>20.00</td>
<td>Red</td>
<td>Red</td>
<td>Danger of complete collapse</td>
<td>1.50～2.00m</td>
</tr>
<tr>
<td>20.00</td>
<td>30.00</td>
<td>Dark red</td>
<td>Dark red</td>
<td>Wooden structures partly damaged</td>
<td>2.00～2.50m</td>
</tr>
<tr>
<td>30.00</td>
<td>100.00</td>
<td>Purple</td>
<td>Purple</td>
<td>Wooden structures partly damaged</td>
<td>2.50～5.00m</td>
</tr>
<tr>
<td>100.00</td>
<td>-</td>
<td>Black</td>
<td>Black</td>
<td>Wooden structures partly damaged</td>
<td>5.00m～</td>
</tr>
</tbody>
</table>

**The Pipework Display Option**

This is the display setting of the pipe work, the manhole, and the drainage network.

**Water Velocity Arrow - Size Adjust**: Set the parameters for the shape of the arrow.

"Open": For xpswmm. Open the xps file.

Height Offset: The entire drainage network can be offset. As a result, it becomes possible to improve viewing of the ground level with an aerial shot to check by comparing with the land use situation.
Drainage system and tube flow

Flood flow velocity vector
**Tsunami Plug-in**
This plugin simulate tsunami using programs developed by research institutes or commercially available software. The result can be imported into UC-win/Road and visualized. There are 3 features.

1. FORUM8 unique open format makes it easy to transform one format into the one recognized by UC-win/Road, allowing the visualization of a variety of different tsunami analysis data.
2. Importing of the terrain mesh used for the analysis.
3. Other than the fundamental elements such as water depth contour map, flow speed, wave force strength, it is possible to optionally enable water surface reflection and water ripples for a realistic presentation.

**1. Load Tsunami data**
The supported format is as follows.

- Tsunami Plug-in standard format
- Other tsunami data format 1 (combination of DEP file and SUP file)
- Other tsunami data format 2 (combination of DEP file and Z file)

**(1) Reading standard tsunami data format**
Go to Analysis Ribbon - Tsunami - Editor to open the dialogue below. Right click on the list in the dialogue, and select Load Tsunami... to open Load Tsunami form. Select Tsunami plugin standard format as Data format.

![Load Tsunami window](image)
- Tsunami Name: Input a name. Default is 'Tsunami 1'.
- Open definition file: Select tsunami definition file.
- Ground Level file: Ground height above sea level. Click on "・・・" to specify a file.
- Grid Interval: Mesh interval of the terrain data. The default value is 5 m (5.000).
- Time Interval: Time elapsed between each step. The default value is 5 seconds (5.000).
- Position X, Y, Z: Origin of the data.
- Water Level File: Water height. Click on "Add Files" and specify multiple files that contain the analysis results over the water level.

Reading too many files will take more time to process. It may be better to skip or thin out data. Thinning out data is easily achieved by specifying the "wildcard" to read only a certain range of data that fits the criteria. For example, typing "Zmax?????00. dat" into the File name box and press Enter will search for only the data that ends with 00. The files selected are displayed in the box in the bottom half of Load Tsunami window.
Click Import to load the data. Once loading is completed, go to Tools - Tsunami to open the Tsunami Plugin window and check that the Tsunami is properly loaded. It will appear with the specified name (Tsunami 1 by default).

(2) Reading other tsunami data format (DEP, SUP files)

Go to File - Load Tsunami again. Select "Other tsunami data format 1 (DEP File, SUP File)".

Tsunami Name: Enter a name. By default it is "Tsunami 1".
DEP file: Specify terrain file. Click on the "・・・" to open file directory.
SUP file: Specify water level data file.
StreetMap File: Currently not available.
Grid Interval: Grid interval of terrain mesh data. The default value is 5.000 m.
Mesh Cols
Mesh Rows
Data digits: Data digits that define one datum written in the SUP file. For example, the following data set contains another datum for every single 10 digit interval.

Time Interval: Time elapsed between each step. The default value is 5 seconds
Click Import to load the data. Once loading is completed, go to Tools - Tsunami to open the Tsunami Plugin window and check that the Tsunami is properly loaded. It will appear with the specified name (Tsunami 1 by default).

*It is not required to enter “Mesh Cols.” and “Mesh Rows”.

(3) Reading other tsunami data format (DEP, Z files)

![Tsunami Plugin window](image)

Tsunami Name: Enter a name. By default it is “Tsunami 1”.
Ground Level File: Specify a round level file.
Water Level File: Specify the analysis result water level file. Click on Add Files to select the file(s).

*The more data loaded, the longer it takes to process. Refer to (1) Reading standard tsunami data format on how to perform wildcard selection.

Click Import to load the data. Once loading is completed, check that the Tsunami is properly loaded. It will appear with the specified name (Tsunami 1 by default)

2. Tsunami setting
(1) Aligning the tsunami analysis data to the terrain.

Align the data with the actual terrain.

Right click on the tsunami data in the Tsunami Plugin window. Select “Look at <Tsunami name>” from the pop-up menu to center the camera.

![Tsunami Plugin window with pop-up menu](image)

*Normally, Origin Y is 0 m. When aligning, it is easier to do it from a bird’s eye view.

The blue part of the following picture is the tsunami water surface. However, for easier reference, the following purposely set
Origin Y to 50m to ensure the water is not hidden underground. When making terrain patches, ensure that Origin Y is 0 before proceeding.

(2) Making terrain patch for the tsunami plugin
If the tsunami data contains terrain data, the data can also be read to display a more detailed terrain. After position alignment, right click on the tsunami from the Tsunami Plugin window, and select "Make patch for <tsunami name>".

(3) Tsunami Rendering Option
Set visual effects parameters.
3. **Tsunami Visualization Animation**

The loaded tsunami animation can be played with the “Tsunami Player”, which is can be opened from Tsunami Plugin Window or “Tools” - Tsunami Player.

![Tsunami Player Interface]

Speed Factor: Playback speed factor. Range between 0.10 ~ 100.00 possible.

4. **Saving tsunami plugin data**

All entries within the Tsunami Plugin Window are saved inside the .rd data. However, the analysis data remains an external source of reference, and its relative directories will be saved inside the .rd file.
Noise Simulation

Noise simulation option can simulate a general soundscape by arranging the sound source and the sound receiving plane on 3D VR space of UC-win/Road. Considering the influences such as the ground level, structures and buildings etc., and the level of sound pressure on each sound receiving point on sound receiving plane is analyzed. This option is composed of pre-processor (Input), main-processor (Analysis) and post-processor (Displaying the result). The sound source and the sound receiving source can be arranged in the pre-processor, and it is possible to get the result for analytical accuracy which is according to various purposes by abundant analysis option in main-processor. In addition, the sound pressure level can be displayed by contour figure, contour line, grid and spherical configuration etc. *This option needs "NoisePlugin (Noise Simulation Option)".

2. Sound source settings

■ Moving sound source Form (Plugin)
Go to Analysis Ribbon - Noise propagation - Sound source - Moving Sound.
Select a model as a moving sound source. Multiple sound sources are generated by arranging it separated in time and space to simulate a moving sound source. It is recommended to use Automatic placement.

![Moving sound source Form](image)

**Actual Simulation Time:** Actual simulation time set in the [Simulation Parameters Form (Plug-in)].

**Settings:** Open the [Criterion value for simulation Plug-in] to edit simulation actual time.

**Arrange spacing:** Time interval to place the sound sources. The range: 0.0 - Simulation length.

**Character of sound source:** Characteristics applied to the sound source.

**Sound Source selection:** Sound source model.

**Start arrangement:** Start placing the sound sources.

**Stop arrangement:** Stop placing the sound sources

■ Character of sound source
Set the characteristic value of sound source from Analysis - Noise propagation - Options - Sound property. Up to 5 base waves for 1 character can be set.
[Add profile] button: Add a characteristic set.

[Characteristics set name]: Name of the characteristics. The range: Up to 10 byte characters

[Frequency]: Frequency (pitch). The range: 1 - 100000Hz

[Amplitude]: Audio power level. The range: 0 - 1000dB

[Add a basic wave] button: Add a basic wave.

[Delete a set] button: Delete the characteristics set. At least one case must remain.

[Copy] button: Copy the selected set.

[Delete a basic wave] button: Delete the selected basic wave.

Hint about amplitude:
Audio power level is the total sound power that the source radiates.
Generally, it seems 40 - 60dB (decibel) is the best value, and that the sound can be dissonant at higher values.

*Note
After deleting a characteristics set, check the value of the set in the sound source models in the Sound source tab in the Model Tool. If a set is deleted, the models using it will use the initial value.
Click a 3D model or a FBX scene model to enable sound.

3. Setting of the sound receiving plane

There are 2 ways:
- Arrange individually for the sound source arrangement
- Arrange the several faces at a time

■ Setting individually
Select “sound receiving plane” from Edit - Library - ▼-Add details - Sound receiving plane, then click on the main screen. A sound receiving plane will be placed at this position.
Several faces at a time.

Set up the size of the sound receiving plane from Analysis Ribbon - Noise propagation - Receiver - Sound Receiver

[Sound receiving plane type]: Type of the sound receiving plane
[Plane size (Width)]: Width of the plane. The range: 0.1 - 999.99
[Plane size (Height)]: Height of the plane. The range: 0.1 - 999.99
[Division width]: Division size Dx, Dy to calculate the influences at the each sound receiving point (intersection point). Smaller values make the result more precisely, however it takes much time to calculate. The range: 0.01 - 999.99
[Setting conditions - Type]: Type of arrangement. Select from "Along the road", "Along the flight path" or "Arbitrary point". *Note: Refer to Hint below for "Manual arrangement".
[Start position]: Distance of the first plane from the start of the road/flight path. The range: -1000.00 - 1000.00
[Offset (Horizontal)]: Horizontal offset from the edge of the road. Positive values are directed towards the center of the road. The range: -1000.00 - 1000.00
[Offset (Vertical)]: Vertical offset. Positive values are ascending. The range: -500.00 - 500.00
[Number of faces]: Number of sound receiving planes. The range: 1 - 999
[Interval of faces]: Interval between each plane. The range: 0.01 - 999.99

Click OK to confirm.

Hint for Operations for arbitrary point.
1. In the main screen, click at a point to place the first plane.
2. Then click on the direction to arrange the planes.
3. The planes are arranged in the direction. Click OK button to confirm.
Repeat 1 and 2 as much as necessary.

3. Setting of analysis condition

**Criterion value for simulation**


**General tab**

**[Simulation type]**: Select Stationary sound source (available on all time) or Moving sound source (available on specified time).
For moving sound sources simulation, a part of the total simulation time is assigned to a source. And it needs much more computation time than a fixed source simulation.

**[Actual time for simulation]**: Total time length of the simulations.
The total length of the simulation should be a little longer than the sound source moving time to ensure the sound can reach the farthest sound receiving plane. The range: 0.00 - 3600.00 sec

**[Interval of simulation time]**: Shorter intervals will cause results mode precisely but take more time.
*Important note*
As if a sound wave is reflected more than once in a single time interval, all reflections after the first one will be ignored, it is important to avoid too long time intervals. 0.01 or smaller would be better. The range: 0.001 - 9.999

**[Division number of a side of icosahedra]**
By default the sound routes are directed towards the vertices of icosahedra. To obtain better resolution, the faces are divided in sub-triangles, and new sound routes point to the centers of those triangles. As the resolution increases, it takes more time. 4 to 8 triangles are better. The range: 1 - 99

**[Adjustment factor of limitation length]**
When the distance from a sound receiving point to a sound route is smaller than the limit distance, it receive the sound of the route but when it’s longer the point doesn’t receive any sound. This factor allows the use of a longer limit distance than the basic one. When a sound route is in the valid distance from the point, the sound pressure level of all those routes will be combined. (The range of input: 0.1 - 9.9) For a too small value, full routes could be ignored, but for too high value it would make distant
unlinked routes have an effect. 1.0 - 1.5 is better.

Details tab

[Sound reflection (Energy ratio)]: Reflection ratio for each surface type. The range: 0.01 - 1.00

[Average density of the object]: Average model density. The range: 0.01 - 9.99

[Run simulation] button: Click to run.

4. Simulation results display options

Display option form is opened from Analysis Ribbon - Noise propagation - Options - Drawing options.

[Display type]: Display type for results. "Contour line (polygonal)", "Contour (fill)", "Contour line (curve)", "Grid" or "Sphere"

[Mesh type]: Mesh type. "None", "Type1", "Type2", or "Type3"

Sound pressure level

[Show legend]: Show color legend.
[Follow a minimum, maximum level of all faces]: Use the simulation result's minimum and maximum sound pressure.

[Set directly]: Set manually the minimum and maximums.

[Minimum level]: Set the minimum sound pressure level.

[Maximum level]: Set the maximum sound pressure level. The range: 0 - simulation result's maximum

[Interval of contour line]: Set the interval between the contour lines. Input range: 0 - simulation result's maximum

[Transparency]: Transparent at 0.0, opaque at 1.0. The range: 0.0 - 1.0

5. Activation and save of the result

Set a destination folder of results from Ribbon, Analysis – Noise propagation – Run.

Right click the existing folder, and select "New" to create a new folder. Click [OK] to execute simulation.

*Note: All file (*.txt) might be deleted when selecting a folder in which contains result files (*.txt). Change the destination folder or save the existing results into the other folder in advance.

6. Importing the result

Import the result from Analysis Ribbon - Noise propagation - Load simulation.

*Specify the data (*.rd) of UC-win/Road which accords with the current data.
**Fluid Analysis Plug-in**

The Fluid Analysis Link Plug-in is a plug-in that takes the CFD analytical result analyzed with general-purpose fluid analysis tool 'OpenFOAM' and displays the streamline.

1. **Flow of analysis**

The work flow starts from the definition of the analysis condition to visualizing the result. However, in case of VTK file, it is exception.

1. Create model
   
   Create the terrain, roads and arrange the structure such as buildings and various models on the terrain.

2. Create STL file
   
   Export VR space of UC-win/Road in POV-Ray format, import it into 3ds Max and export it in the STL (Standard Triangulation Language) format.

3. Analysis with OpenFOAM
   
   After importing the file with OpenFOAM, create meshes, define analysis conditions and execute the analysis.

4. Create VTK file
   
   After importing the analytical result using the ParaView bundled in the OpenFORM, generate the file of each time step in VTK (Visualization Tool Kit) format.

5. Visualization on UC-win/Road
   
   Import the VTK file into the UC-win/Road with the Fluid Analysis Link plug-in and visualize it.
2. How to use fluid analysis plug-in

Install VTKPlugin.bpl. Go to Analysis Ribbon - Fluid - Load simulation to open the setting form.

- Tool Button

- Import VTK or VPF file.
  When importing VPF file, specify VTK Plugin Data Files as "Category of file".
  * .VTK: it is the analysis result of [OpenFOAM]. To display all the steps, select all files subject in the window.
  * .VPF: it is the file(*.VPF) saved by this plug-in. This file contains the information on coordinates.
- After loading the VTK file and setting the coordinates, the extension changes to *.VPF.
- Delete all current data.
- Return to the first step.
- Play steps in reverse.
- Play steps.
- Pause.
- After stopping it, the steps will be played from the first step when Play button is clicked again.
- Jump to the last step.
- Play steps in any time frame by moving the scroll bar.
- Open Help.
- Repeat the playback.
**Geometry tab**

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Color</th>
<th>Display</th>
<th>Other</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X</td>
<td>1.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y</td>
<td>1.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td>-1.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Offset</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>-8.19</td>
<td>$11.66$</td>
<td></td>
</tr>
<tr>
<td>Angle</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>9.00 deg</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Scale**
  Specify the scale within the range of the maximum display. When it is 1.0, fluid line is drawn using the original coordinate value. When the negative value is set, it is drawn in reverse. X is for the east-west direction, Y is for the vertical direction and Z is for the north-south direction.

- **Offset**
  Set the offset from the origin point of the coordinate system of UC-win/Road to the center within the maximum range where the fluid line can be seen. The origin point is the northwest corner of the terrain.
  Note: It is necessary to adjust the geometry by changing the offsets because usually the coordinate system of the VTK file (the coordinate system used in the analysis on the 'OpenFORM') is different from one of UC-win/Road. Calculate and set the difference between them or move the fluid line data to a present position with 'Adjust' button.

- **Adjust button**
  Calculate the center position of the fluid line and then move the view point to this position. Moreover, the direction of height is adjusted to locate the bottom of the fluid line data in the ground level.

- **Angle**
  Specify the angle from the upper side. The fluid line data rotates on its center position. Clockwise is positive.

**Color tab**

- **Data to set color**: Select a result to display.
- **1D data settings**: Set the 5 colors of maximum/minimum value. Check Use 3 colors setting to display with gradation...
- **3D data settings**: Select Magnitude (resultant force of value of two or more axes) or Pick up component (display by focusing on either of axis)

**Display tab**

- **frame step**: Specify time interval in each step.
· **Display legends**: Display Name of display object, Maximum value, Minimum value and Gradation on main screen.

· **Display Frame Count**: Display the 'current step number / total step number' at the bottom.

· **Antialiasing**: Reduce the jaggies (diagonal line looks the blurry like stairs).

· **Line width**: Specify the line width. It is generally between 1.0 and 2.0.

---

**Other**

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Color</th>
<th>Display</th>
<th>Other</th>
</tr>
</thead>
</table>

Axis of VTK file: Set the axis when the VTK file is read.
Munsell Color Space Export Plug-in

The Munsell Color Space Export Plug-in is a plug-in to visualize the scenery displayed on the main window by the Munsell color system, and exports the scenery to the Munsell color file (the extension: mcs). This file can then be used in the design or research based on Munsell color.

Execution Method

Select File - Export - Save to MunsellColor File...

* In case of cutting out unneeded things for munsell value like street tree, road sign etc. for output uncheck the element to cut out in "Display" tab of Home Ribbon - Visual Options, and then it is possible to execute in the state of displaying only building. It is possible to execute in a state of hiding the street tree and road sign like the following example.

Structure of output file

Munsell color file is an open format with an extension, .mcs, and has following format.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Header</td>
<td></td>
</tr>
<tr>
<td>Data 1</td>
<td></td>
</tr>
<tr>
<td>Data 1</td>
<td></td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
<tr>
<td>Data N</td>
<td></td>
</tr>
</tbody>
</table>
- Header of Munsell color file

- Header part: Header part is 32byte

<table>
<thead>
<tr>
<th>Byte count</th>
<th>Data Content</th>
<th>Data Type</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>3byte</td>
<td>‘MCS’</td>
<td>—</td>
<td>MCS (4D 43 53) is put at top of the header, 3byte.</td>
</tr>
<tr>
<td>1byte</td>
<td>—</td>
<td>—</td>
<td>Not used (it must be 0)</td>
</tr>
<tr>
<td>4byte</td>
<td>The size of the header</td>
<td>Unsigned 32 bit integer</td>
<td>The size of the header unit.</td>
</tr>
<tr>
<td>4byte</td>
<td>Major version of the format</td>
<td>Unsigned 32 bit integer</td>
<td>The major version will be changed when the file format is changed so that it is influenced between formats. It is 1 now.</td>
</tr>
<tr>
<td>4byte</td>
<td>Minor version of the format</td>
<td>Unsigned 32 bit integer</td>
<td>The minor version will be changed when the file format is changed without influences in the same major version. It is 0 now.</td>
</tr>
<tr>
<td>4byte</td>
<td>The size of the file</td>
<td>Unsigned 32 bit integer</td>
<td>The size of the exported file.</td>
</tr>
<tr>
<td>4byte</td>
<td>Image width</td>
<td>Unsigned 32 bit integer</td>
<td>The image width of the mansell color exported.</td>
</tr>
<tr>
<td>4byte</td>
<td>Image height</td>
<td>Unsigned 32 bit integer</td>
<td>The image height of the mansell color exported.</td>
</tr>
<tr>
<td>4byte</td>
<td>The offset to the Mansell color data</td>
<td>Unsigned 32 bit integer</td>
<td>The top address of mansell color data.</td>
</tr>
</tbody>
</table>

- Structure of data part of Munsell color file

- Data part : 16byte for 1 unit.

One unit of data is 16 byte. The top left corner of the saved scenery will be the starting point.

Therefore, the top left pixel will be saved first and then the pixel sitting directly to the right and below that pixel will be saved, and so on. The pixel at the bottom right corner will the saved last. During this process, the pixels are converted to Munsell Color. Note that achromatic color in Munsell color file is represented as the following: Color Code "N", Color dividing value "-1.0", Chroma value "0".

<table>
<thead>
<tr>
<th>Byte count</th>
<th>Data Content</th>
<th>Data Type</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1byte</td>
<td>Hue code</td>
<td>R:00, YR:01, Y:02, GY:03, G:04, BG:05, B:06, PB:07, P:08, RP:09, N:0A</td>
<td>The value that identifies the hue code (N shows the achromatic color). For instance, when the hue is 5R, the code for R, 00, is here. All codes for hue are shown in the data type.</td>
</tr>
<tr>
<td>3byte</td>
<td>—</td>
<td>—</td>
<td>Not used (it must be 0)</td>
</tr>
<tr>
<td>4byte</td>
<td>Dividing value of hue</td>
<td>The single precision real value, 4byte</td>
<td>The value of dividing value of hue. For instance, when hue is 5R, 5, but expressed in the single precision real value, is here. It is -1.0 for the achromatic color.</td>
</tr>
<tr>
<td>4byte</td>
<td>Value</td>
<td>The single precision real value, 4byte</td>
<td>The value of the brightness, but expressed in the single precision real value.</td>
</tr>
<tr>
<td>4byte</td>
<td>Chroma</td>
<td>The single precision real</td>
<td>The value of the chroma, but expressed in the single precision real value.</td>
</tr>
</tbody>
</table>
Example of Munsell color file opened by binary editor.

<table>
<thead>
<tr>
<th>Value</th>
<th>4byte Real Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value1</td>
<td>Real Value1</td>
</tr>
<tr>
<td>Value2</td>
<td>Real Value2</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>ValueN</td>
<td>Real ValueN</td>
</tr>
</tbody>
</table>
SfM (Structure from Motion) Plug-in

1. What is SfM (Structure from Motion) plugin?

SfM Plug-in creates a 3D point crowd model of the objects in the space from plural photos. It also estimates the positions of cameras and displays the models on UC-win/Road. The point cloud and the camera positions are obtained by applying the SfM method for the loaded photo images.

*SfM (Structure from Motion): The technology that estimates 3D location of the feature point in the picture and the camera position and orientation through analyzing plural photos.

2. Main functions of SfM plugin

SfM plugin has the following functions.

● Create point cloud from photo
   Export a space based on image extracted from photos and videos which taken with a digital camera as point cloud.

● Point cloud display adjustment
   Adjust to display appropriately the exported point cloud on UC-win/Road. By adjustment, VR model can be rotated and displayed over the point cloud data.

● Create camera calibration files
   Create a revision file to eliminate the distortion of image by taking photos of a chess board with digital camera and analyzing focus distance and lens distortion properties of the camera

● Create VisualWords files
   Create and export VisualWords file needed for creating point cloud.

3. Operation flow

1. Take photos
   Take photos of space with a digital camera. The photos are at least 2 that are moving little by little. And if EXIF information is not added on the photos, take photos of chess board pattern because SfM analysis needs the characteristics of digital camera.

   *Exif (Exchangeable image file format) : An image file format that can be add the information about the shooting conditions to the digital camera photo data. Metadata such as shooting date, model name, resolution, exposure time, diaphragm value, focal length, ISO speed, and color space are saved with the image.

2. Create camera calibration files
   Create camera calibration file from photos of chess board pattern.

3. Create Visual Words files
   Create Visual Words file for SfM analysis. It takes a long time to create Visual Words file. So, it is recommended to use the sample file prepared in Plug-in.

4. SfM Analysis
   Load the photos and the camera calibration file and Visual Words file. After settings, the analysis starts.

5. Visualization on UC-win/Road
   Starting SfM analysis, Point cloud data and camera position are displayed on UC-win/Road.
4. Analysis methods

Select Real-time analysis or Batch analysis as an analysis method by SfM engine.

Real-time analysis mode monitors folder of photos, analyzes the added photos immediately and exports point cloud, but it consumes much memory.

Batch analysis mode analyzes only target photos which are set before analysis and outputs point cloud. Advantages are a little memory consumption and to be able to analyze more photos than Real-time analysis. However, the analysis result cannot be obtained if the discontinuous photos are entered because this analysis processes only continuous photos.

- **Batch Analysis**

Select photos to be analyzed and set analysis conditions. After starting the analysis, view positions of each selected photos are calculated and displayed on the 3D space. As the analysis progresses, the number of point clouds increases and gradually the structure is clarified. Check the point clouds in the space when all image analyses have finished. If the point clouds are few, select “give priority to the number of the point clouds” in the analysis condition and perform the analysis again. In the case that the analysis stops in the middle, perform it again after altering the starting image or removing the one where the analysis stops.

- **Feature point detection algorithm**

Select SIFT or SURF as feature point detection algorithm.

- **SIFT (Scale Invariant Feature Transform)**
  This algorithm detects feature points and describes the amount of feature points, used in fields such as image recognition. Transaction speed is slower than SURF, but the recognition precision is high.

- **SURF (Speed Up Robust Features)**
  This algorithm is a modification of SIFT and its transaction speed is faster than SIFT. But the recognition precision is lower than SIFT.
5. Tips on using SfM plug-in

- Shoot photos so that the objects in ones already shot will be included. If not, no point cloud will be created because the positions in the space are unrecognized. (For the batch analysis, the objects in the last 4 photos must be included.)

- Brightness should be constant because the recognition of the objects is difficult if the brightness changes greatly than the last analyzed photo. For example, indoor shooting should be done in the room lighting with curtains closed to block the outside light. In the case of the outside shooting, cloudy weather is desirable because brightness changes between sun and shade. If with both of these photos, analyze them separately and connect them in VR space by using the point cloud adjustment function.

- The first 2 pictures for the analysis affect the number of the point clouds output after that. If the point clouds are not output properly, it may be improved by changing the order of the photos to input to SfM plug-in.

- If the analysis stops immediately, set the "give priority to the number of the point clouds" in the priority content of the feature detection algorithm. This has lower point cloud precision but has higher detecting ratio of the camera position.

The analysis result by SfM plug-in: point crowd, camera position and direction (A white globe and arrows)
Assessment Plug-in

1. Overview
It is possible to assess 4 environmental items.

- **Green coverage rate calculation**
Evaluate the green coverage rate of the scene captured from different angles.

- **Sunshine calculation**
Evaluate the sunshine for certain time in details.

- **Reflected light check**
Check visually the reflected lights on solar panels.

- **L-Tree**
Generate L-trees, which are 3D trees generated by L-System.

2. Operation flow

Start UC-win/Road and enable Assessment plugin from File - License Manager.
The ribbon menu is displayed on the Analysis tab.

(1) Green coverage rate
Click Green coverage rate to open "Green coverage rate calc" form. Load a (*.rd) file in advance.

Color range setting
Set the color range to identify green area.
It is calculated using HSV format. The pixels whose H value is within the specified range and whose S and V values are larger than the selected value are considered as green, and the ratio to the whole capture is calculated in percentage.

Color range
H: Hue 0 to 360, S: Saturation 0 to 100%, V: Value Lightness 0 to 100%

"Capture" button: Take a screenshot. The first scene is captured when "Green coverage rate calc" screen is opened.
"Calc" button: Calculate the green coverage rate of the captured scene. Green parts are displayed as white.
"Close" button: Close the form.

(2) Sunshine calculation
Click "Sunshine calc" to open the Sunshine Calc form. Load a (*.rd) file in advance.

Date: Set the date. The winter solstice day is set as default.
Time: Set the time to start the calculation. Sunshine is evaluated every hour after the starting time.
Selected model: After clicking "Sunshine Calc" button, click on a model whose sunshine is evaluated.
•  Time: The table displays the execution time of Sunshine Evaluation.
•  Judgment: The result is displayed as O or X. When the model goes into the shade, the result is X. It is measured at the center of the upper surface of the model.

Example

(3) Solar panel reflection light check
Click "Reflected light check". Load a (*.rd) file in advance.

Position setting: Set the position of solar panel.
Height from the Ground: Set the height of the solar panel from the earth.
Solar panel slope: Set the slope of the solar panel.
Solar panel orientation: Set the orientation of the solar panel.
Draw settings: Check Draw to display the incident and reflected light. The length of the beams can be change using the slide bar.
Length of one side of solar panel: Specify the length of the edge in the result.
Draw: Check Draw of the time to calculate the reflection.

Examples
(4)L-Tree
L-Tree is a 3D tree model created by L-System, which is a system to simulate objects naturally by changing only the size and repeating it.

L-System (G) is calculated as $G = \{V, S, \omega, P\}$.

- **V**: Variables which are replaced in order based on the replacement rule $(=P)$. As the calculation proceeds, objects grow up by the strings of V.
- **S**: Numerical constants
- **$\omega$**: Strings composed of V.
- **P**: Replacement rule to change V. Each element is written with the combination of strings.

1. **Create a new L-Tree**
Click the ribbon L-Tree, and click Add button to open the L-System tree settings. Or go to Model Panel, New – L-tree.

2. **Settings on the Default tab**
Set up the length, thickness, rotation angle, type of the trunk and growth rules.

3. **Settings for the texture and material of trunk**
Click Texture edit button to open Trunk texture edit form.
4. Settings for the texture and material of leaves

Go to the Leaves tab on L-Tree Edit form and click Texture edit button.

Leaves are added to the points of each unbranched branch.

5. Settings for generations

Set up the generations, which means the growth stage by the generation number. As the generations increase, the straight parts of each branch are replaced based on the growth rule.

Note: This process will take time to complete depending on the tree shape.

6. Arrange

Arrange the L-Tree.

- L-Tree command

<table>
<thead>
<tr>
<th>Command</th>
<th>On command editor</th>
<th>Description</th>
<th>Image</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>Straight</td>
<td>The straight part of branch</td>
<td></td>
</tr>
<tr>
<td>Y</td>
<td>+Yaw rotation</td>
<td>Yaw rotation +direction</td>
<td></td>
</tr>
<tr>
<td>-----</td>
<td>---------------</td>
<td>-------------------------</td>
<td></td>
</tr>
<tr>
<td>y</td>
<td>-Yaw rotation</td>
<td>Yaw rotation –direction</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td>+Pitch rotation</td>
<td>Pitch rotation +direction</td>
<td></td>
</tr>
<tr>
<td>p</td>
<td>-Pitch rotation</td>
<td>Pitch rotation +direction</td>
<td></td>
</tr>
<tr>
<td>R</td>
<td>+Roll rotation</td>
<td>Roll rotation –direction</td>
<td></td>
</tr>
<tr>
<td>r</td>
<td>-Roll rotation</td>
<td>Roll rotation –direction</td>
<td></td>
</tr>
<tr>
<td>[</td>
<td>Branched</td>
<td>The starting point of branched part</td>
<td></td>
</tr>
<tr>
<td>]</td>
<td>-</td>
<td>The end point of branched part</td>
<td></td>
</tr>
</tbody>
</table>

**Command example list**

**Growth rule: F**
F is a straight part of trunk or branch. It is displayed with a blue line.

**Growth rule: FYF**
It is displayed from a base of stalk toward a point.
Each R, P and Y mean Roll rotate, Pitch rotate and Yaw rotate of the curved parts to positive directions', p and y means the curve to a negative direction,

**Growth rule: F [YF] F**
A branched part is enclosed with [].

**Growth rule: F[YF [YF]]F**
The growth rule of the branched part on [ ] are enclosed with [ ].

---

**Command replacement as L-System generation increases**
In this rule of L-System command replacement, as the generations increase, the straight parts of each branch are replaced based on the growth rule.

<table>
<thead>
<tr>
<th>Default</th>
<th>Growth rule</th>
</tr>
</thead>
</table>

-516-
Command: F [YF] F (Yaw rotation=90°) It is not replaced.
The branches enclosed in red are replaced in the next generation.

1
This is the first generation
Two branches are replaced base on the shape of the growth rule.
The branches enclosed in red are replaced in the next generation.

2
This is the second generation
Command: F[YF[yF][yF][yF][yF]][yF][yF][yF][yF][yF][yF][yF][yF]
Four branches are replaced base on the shape of the growth rule. The branches enclosed in red are replaced in the next generation.
UAV Plug-in

1. Safety Guidelines
This system is intended to be used under the following general conditions:

- Please fly in an open area, with nothing that could potentially obstruct the flight (buildings, trees, power lines …) or get caught in the propellers
- Weather conditions: no rain, no wind (the maximum allowed wind speed is 4 m/s for optimal flight experience)
- Please keep a safe distance from sources of possible electromagnetic interference (radio transmission towers, high-voltage wires, cell towers…)
- Please always keep direct sight of the UAV while flying
- Please keep a distance of at least 30 meters from any persons in the vicinity
- Flying over a crowd is usually not allowed
- Please fly only during the day, between sunrise and sunset, to avoid the UAV being caught accidentally in power lines or electric towers

Also, please check the local flight regulations, usually available at the city hall or prefectural office. These regulations will allow to confirm the locations of restricted areas and under which conditions it is possible to use UAV.

Please be aware that flying a UAV comes with risks that could have heavy consequences. Some UAV distributors offer insurance subscription services that might cover possible damages caused by the UAV. It is recommended that the user consults with the UAV distributor regarding these insurance options.

Forum 8 will not be held responsible for damages resulting from any misuse of the whole system, or any malfunction of the software.

2. Overview
The purpose of the UAV plugin for UC-win/Road is to allow the user to take aerial pictures/videos in a very simple way. It is designed for a wide range of users, from users with few flying experience who wish to take pictures or videos without the stress of controlling the UAV with a remote controller, to experienced flyers who wish to automate the whole process in order to improve efficiency.

Within the Virtual Reality environment, the user creates a 3D path as a series of points (waypoints) connected by lines that will serve as guiding line for the UAV to follow. Upon any of the waypoints, the user can add actions to perform when the waypoint is reached (take picture, start recording a video, stop the video recording, and change the camera orientation…). The combination of the 3D path and actions defines a flight mission that specifies what the UAV should do from take-off to landing. Once the flight mission is defined, the user can start the mission execution simply by pressing a button in the graphical user interface (GUI). The mission will be uploaded automatically to the UAV, which will then take off, fly along the predefined 3D path, perform the required actions and land at the last waypoint. After landing, the user can download the pictures and video taken during the flight simply by pressing a button in the GUI, so the pictures and videos will be available on the PC.

The UAV plugin was developed with the DJI Phantom 3 Professional UAV as it supports autonomous flight functionalities and offers remarkable flight stability, but ultimately, the UAV plugin is designed to extend the support to other UAV platforms offering similar autonomous flight capabilities.

3. Requirements
The most basic control of a UAV is usually made manually through a remote controller. Nowadays, a wide range of UAV’s can execute a flight mission planned beforehand on a separate computer and uploaded to the UAV to be executed autonomously. More recent models of UAV’s allow the connection of a mobile device (mobile phone, tablet) to the remote controller to design a flight mission while on site, upload it to the UAV and monitor its motion during the whole flight through the screen of the mobile device.

Our approach is to use Virtual Reality functionalities of UC-win/Road to simplify the creation of the flight mission and the monitoring of the flight mission execution. Within a 3D virtual environment, it is possible to create a flight mission in a simple way by selecting
3D positions, upload the information to the UAV and launch the execution of the flight mission. While the UAV is flying, its progression is continuously monitored in real time in 3D to confirm that everything is going according to the flight plan.
In order to do so, the mobile device is used as an interface between the actual UAV and the PC, which allows the user to control and monitor the UAV’s flight from a PC through UC-win/Road.
This was achieved by developing an Android interface application named F8 UAV Controller, which is based on the android version of the DJI SDK that communicates with UC-win/Road through Wi-Fi.
The full system is shown in Figure 1: DJI Phantom 3 Professional, remote controller, android tablet (android interface), laptop (UC-win/Road and UAV plugin) and a Wi-Fi router for the communication between UC-win/Road and the android interface.

Note:
Now the application cannot be installed from Google Play. The APK package of the Android application is offered with UAV Plugin from Forum8.

Note:
It is recommended to install the DJI application according to UAV version.
* DJI Go: For UAVs released before Phantom 4. (For example, Phantom 3 series, Inspire 1 ...)
* DJI Go 4: For UAVs released after Phantom 4 (For example, Mavic Pro, Phantom 4, Phantom 4 Pro, Inspire 2 ...)
This DJI Go/Go 4 is necessary to activate and set up UAV.

Figure 1: Required devices

3.1. UAV
The UAV plugin supports a wide range of models from DJI but only the following ones could be confirmed: Phantom 3 Advanced, Phantom 3 Professional, and Phantom 4 Pro.
The following models should also be supported but we didn't have the opportunity to confirm them: Phantom 4, Mavic Pro, Inspire 1, Inspire 2, Matrice 200/210 series, Matrice 600, and Spark.
The models used in our development are the Phantom 3 Advanced, Phantom 3 Professional (shown in Figure 1) and Phantom 4 Pro.
All these UAV’s are equipped with (at least) one camera mounted on a 3D gimbal, which purpose is to stabilize the camera’s orientation while the UAV is flying.

3.1.1. Coordinates systems
The UAV plugin uses two types of coordinates systems (Figures 2, 3):
- The UAV’s local coordinates system. In this system, the motion of the UAV is given independently of its current location and orientation (for instance, move forward/backward)
- The Absolute coordinates system describes the motion of the UAV’s in absolute way, with regards to its latitude, longitude, altitude and heading

![Diagram of UAV's body coordinates system](image1)

**Figure 2: UAV’s body coordinates system**

![Diagram of Absolute coordinates system](image2)

**Figure 3: Absolute coordinates system**

About gimbal, camera, remote controller, Android tablet, laptop PC, Wifi router and Internet connection, refer to Help document.

### 4. UC-win/Road and UAV Plugin

Start UC-win/Road. From the plugin manager, make sure the UAV Plugin is authenticated (USB protect key required).

![UC-win/Road interface](image3)

**Hint**: It is recommended to use the UAV Plugin in full screen mode by either pressing the F10 Key or by a right click anywhere in the 3D environment then select "Full Screen". For the rest of the document, we will be in full screen mode.
In that view, we can see the UAV Plugin window, on the left side of the screen, the side bar on the right side of the screen, and finally in the top side of the screen, the camera latitude and longitude, and the location pointed by the mouse cursor.

4.3. Sidebar buttons
The sidebar buttons include "Mission", "Locate UAV", "Pause/Resume mission", "Download media", "UAV Camera" and "Take off/Land" function.

5. Flight Mission
The autonomous flight capabilities of the UAV are accessible through the definition of a flight mission. A flight mission has two components:
- a flight route created from as a series of 3D positions/orientations (waypoints)
- actions that can be performed when the UAV reaches a waypoint

Once a flight mission is created within UC-win/Road (mission planning), it is uploaded to the UAV through the Android interface and executed autonomously. The UAV will take off autonomously, fly to each waypoint successively and land. Whenever a waypoint contains an action (or a series of actions), the UAV will stop at the waypoint, execute the action(s) and fly to the next waypoint.

After the flight mission is completed, the user can download the media files captured during the mission. The main steps of a typical use of the system are shown in Figure 17.
4.4 Main Window
It has 4 tabs, Connections, Network Data, Camera Control, and Settings.

4.4.1 Connection
Use this to connect UAV Plugin with the UAV via a remote device (Host). It is possible to detect devices to be connected automatically or manually. For details, refer to Help.

Figure 17. Typical use of the system
4.4.2 Network Data tab
On this tab, it is possible to confirm the information about UDP frame data, UAV information, UAV position and UAV orientation.

4.4.3 Camera Control Tab
It is possible to control a camera mounted on UAV.
4.4.4 Settings tab
It is possible to set up the options.

5.1. Mission Editor: Creation of a Waypoint Mission
5.1.1. Mission Edition Mode
The mission planning consists of the full definition of task the UAV will perform. It starts by creating the path that the UAV will follow. In the current version of the UAV Plugin, once the user opens the mission editor, the camera will automatically move to a top-view position and the mission edition mode will be enabled (Figure 18). From there, the flight route is created in the virtual environment by drawing lines with the mouse.
As the user creates new waypoints, they will appear in the waypoints list, and by selecting a waypoint in the list, its properties will be displayed in the "Waypoint Properties" area, along with the list of actions defined for that waypoint ("Actions" area). Note: For the DJI Phantom 3, a flight mission should contain between 1 and 99 waypoints, while each waypoint contains between 0 and 6 actions.

5.2 Mission Execution

If the user is satisfied with the generated preview, the execution of the mapping mission can be launched by pressing the "Execute" button. Before the mission is executed, a confirmation message will be displayed. Once the execution is started, the UAV will:
- takeoff automatically
- fly vertically until reaching the altitude of the first waypoint
- fly directly to the first waypoint
- take a picture at each waypoint
- land automatically
5.3. Media Files Download and Post Processing

Once the UAV has landed, the F8 UAV Controller application will automatically download a preview version of the pictures recorded during the flight and send them back to the connected clients.

The pictures are stored on the android device and on the PC under the mission's corresponding folder, identified by a unique timestamp.

An example of the file path that picture will be saved to be:

- On the PC:
  %userdatafolder%¥UAV_DATA¥yyyyMMdd@hhmmss¥MEDIA_FILES

  With yyyyMMdd@hhmmss being the current date and time defined as year, month, day of the month, hour, minute, second.
  (For instance, 20171108@154632 stands for the November 8th 2017, 15 hour 46 minutes and 32 seconds)

- On the android device:
  /mainstorage/Forum8/UAV_DATA[PLUGIN_VER_x.x]/yyyyMMdd@hhmmss/MEDIA_FILES/

  With x.x being the UAV plugin version (Currently: 3.0)

When the media files are requested by pressing the "Download all available media files" button, and they are not part of a specific flight mission, then they will be downloaded to the default file path as shown below:

%userdatafolder%¥UAV_DATA¥MEDIA_FILES.

Note: In the case of an important number of pictures or videos to download, the wireless download might take a long time to complete, so it is recommended to connect the UAV to the PC with a USB cable and download the media files by using windows explorer, as it will be seen as a digital camera.

6. Starting the UAV and the Android Device

Android Application F8 UAV Controller is used to connect PC with UAV.

It supports the following models:

- Phantom 3 Advanced, Phantom 3 Professional, Phantom 4, Inspire 1, Phantom 4 Pro, Phantom 4 RTK, Mavic Pro, Mavic 2 series (Pro, Zoom, Enterprise), Mavic Air, Inspire 2, Matrice 200/210, Matrice 600, DJI Spark

Note:

It is necessary to use DJI Go application to activate an UAV for the first time. This process requires Internet connection, so it is recommended to activate and adjust calibration in advance.

Quick Connection Steps

- Power OFF the UAV, remote controller and android device.
- Connect the android device to the Wi-Fi network
- Connect the micro USB port of the android device to the USB port of the remote controller by using a micro USB cable (Figure 7)
- Check the Remote Controller battery level (press the ON button of RC shortly once). If the battery indicator shows less than 3 solid white dots over 4, then charge the RC first.

Note: Avoid using the remote controller while charging it, as it will overheat and possibly get damaged.

- Open the F8 UAV Controller Application in the Android device

  Note: in the previous version of the application, turning the RC would automatically start the application. Now the application needs to be explicitly launched before turning the RC ON

Note: F8 UAV Controller V2 is started automatically by operating the remote controller. In case of F8 UAV Controller V4, Start the
application before turning on the remote controller.

F8 UAV Controller

Note:
The device must have Internet access to start the application for the first time. Once Authentication is completed, it is used offline.

After starting the application, the screen changes to the following.

After initialization is completed, click NEXT. Do not turn on the remote controller yet.
After initialization

Then turn on the remote controller.

Press a power button once, and press it for 2 seconds again. The Android device vibrates and displays the dialog to select an application.

Note: If the following dialog is not displayed, repeat turning on and off the remote controller. If it does not still work, close the application and try again from the first process.
Choose F8 UAV Controller, and click Just Once. When Always is selected, it is not possible to access to USB by other application. Change the setting to Just Once on the Android device settings.

After starting F8 UAV Controller application, the remote controller will be detected. The screen will change to the following.

- Turn on UAV

Push ON button once to confirm the battery level. If the number of green bar is less than 3, charge it at first.

Push the power button short, and then hold down it for a bit and release it to turn on UAV.

The application detects and lists the UAV being connected and all components (remote controller, camera, gimbal, battery...). The screen changes depending on the connected devices.
Click Start.

Now the system is ready to be used outside or inside (with Simulator mode).
Before using it outside, wait for the UAV preheating and GPS lock to finish. The LED light becomes orange, then blinks n green and orange, and finally it becomes green.
If the light does not become green, confirm the number of satellites on the left-side status bar. At least 7 satellites are required to stabilize

7. Android Interface
7.1. Overview
The F8 UAV Controller android application makes the link between UC-win/Road and the UAV. Its purpose is to relay the commands from UC-win/Road to the UAV for execution and retrieve real time navigation data from the UAV and send them back to UC-win/Road.
The android device is connected to the USB port of the remote controller through a USB A cable. The user should always make sure that the android device is fully charged before starting a flight mission.
At this stage, we assume that the UAV has its firmware updated (main controller firmware and remote controller firmware), and that its compass has been calibrated.
If not, refer to DJI website to check the firmware update procedure and the compass calibration procedure corresponding to the
Once the UAV is powered on and the GPS location could be acquired, the android interface should be displayed as in Figure 40.

The android interface shows the UAV’s camera view so the user can check (almost) in real time what is actually captured when the camera takes a photo or records a video.

**Left Status Bar**

The left status bar shows the status and information of the main components of the UAV system currently in use: connection to the UAV, model of UAV, remote controller connection, compass status, status of the UAV’s battery, GPS status, and status of the UAV’s SD card.

Note: In order to accommodate a wide range of android devices display, the status bar can be scrolled vertically by pressing it and swiping down or up.

**Real Time Video Stream**

The application screen shows the view from the UAV’s front camera (Figure 40), so the user can check in real time what the camera is currently capturing. The video streaming is received wirelessly through the remote controller and displayed on the android interface. The streaming usually takes few seconds to stabilize. However, on occasions interferences can occur, for instance when the remote controller is located near a strong magnetic field, causing distortion in the image or even loss of signal. In that case, please move away from possible source of interferences and restart the system.

Although the quality of the video feed doesn’t influence the operation of the UAV, we recommend to have a stable video stream and free from interferences before executing a flight mission.

**Right Buttons Bar**

Located on the right side of the interface, the button bar allows to set the system options and control the UAV with main commands (Figure 41).

All the functions on the right side bar (Figure 41) are accessible as soon as the connection with the UAV is established.
Top Status Bar
On the top side of the interface, status indicators as well as warning messages are displayed (Figure 42):

7.2. Android Interface Settings
The settings window is shown in Figure 43. It is accessed by pressing the “Settings” button on the main view of the application (Figure 42).
Settings

Local IP Address: 192.168.21.15
Android Application Version: 3.0

- Allow waypoints recording by using the remote controller
- Automatically start video recording at takeoff and stop at landing
- Automatically start flight data recording when motors are turned ON
- Allow camera to look up (+30 deg max)
- Display front distance sensors data (if available)

Waiting time between commands

- FORMAT UAV'S SD CARD
- SHOW UAV'S INFO
- IMPORT WAYPOINTS
- ENTER SIMULATOR MODE
- OK
- CANCEL

Figure 43: F8 UAV Controller settings
Oculus Rift Plugin

1. Operation environment
Make sure both requirements of Oculus Rift and UC-win/Road are satisfied.
Oculus Rift requirements are detailed on Oculus website.

Recommended system requirements
OS: Windows10
CPU: Intel i5-4590 / AMD Ryzen 5 1500X or later
Memory: 8GB+ RAM
Graphics: NVIDIA GTX 1060 / AMD Radeon RX 480 or higher
(Others: HDMI1.3 video output, USB3.0 port ×3, USB2.0 port ×1)

Necessary system requirements
OS: Windows10
CPU: Intel i3-6100 / AMD Ryzen 3 1200, FX4350 or later
Memory: 8GB+ RAM
Graphics: NVIDIA GTX 1050Ti / AMD Radeon RX 470 or later
(Others: HDMI1.3 video output, USB3.0 port ×3, USB2.0 port ×1)

UC-win/Road requirements are detailed on FORUM8 website.
OS: Windows8.1/10 (64bitOS)
CPU: Intel Core i7 Quad-core or more. CPU 3.2GHz or more.
Memory: 8GB or more
Graphics: NVIDIA GeForce GTX 950 or higher. 4GB or more of Video memory.

Oculus Rift Software
Install Oculus Rift software. Oculus Rift Runtime installer can be downloaded from http://oculus.com/setup/.

2. Operation flow
This version is 1.34.0. It may differ depending on version.

1. Install Oculus Rift software. Connect Oculus following the installer's instructions.
   After setup is completed, run Oculus Rift software, mount Oculus Rift on the head. The screen is displayed on Oculus Rift.
2. Click Settings on the left-side menu and allow Unknown Sources on Oculus Rift software setting.
Click General and allow Unknown Sources.

The window shows a warning message. Check it and click Allow.

3. Run UC-win/Road and enable Oculus Rift plugin from Menu File - License Manager.

4. To start displaying scene, click Start on Device tab of Oculus Group.

5. To pause displaying, click Pause on Device tab of Oculus Group.

3. Oculus Ribbon menu
When Oculus is enabled and a project is opened, the following ribbon menus are added to Device tab.

- Run/Stop: Start/Pause displaying scene on Oculus.
- Reset view: The coordinate system of the Oculus can be initialized so the user can chose the direction of the front view. However, roll and pitch are chosen according to the direction of gravity.

In navigation mode, the coordinate system behave as follows:
• Free mode, walking mode: Yaw angle is the one of the camera, roll and pitch angle are set to 0.
• Driving mode, flight mode: the coordinate system of the vehicle is used.

**Setting:** Display Oculus setting form
**Help:** Open the help menu.

4. Limitation on Oculus Rift

The current version of Oculus Rift has the following restrictions.

- Solar flare and whiteout
  Solar flare and whiteout are not drawn correctly.

- Windshield (Rain wand wipers)
  Windshield wipers and rain on the windshield are not drawn in 3D.

- Jump and rotation with right click
  Jump and rotation with right click functions do not behave correctly.
HTC Vive Plug-in

What is HTC Vive plug-in
HTC Vive plug-in allows to use functions introduced by linking HTC Valve and UC-win/Road with OpenVR SDK. With this plug-in, it is possible to display UC-win/Road scenes on HMD (Head-mounted display) and link its movement with the camera of UC-win/Road using Steam VR (TR) tracking device.

Safety precautions
Follow the next recommendations for a safe experience.
- HMD like HTC Vive may cause 3D sickness. When it occurs, rest the eyes and take a rest in a cool ventilated place.
- Ensure all cables are correctly wired and avoid tripping over the cables.
- Avoid placing obstacles in the play area.

System requirement
To correctly use HTC Vive with UC-win/Road, make sure both systems requirements are satisfied.

HTC Vive requirement are detailed on Vive's website.

<table>
<thead>
<tr>
<th>Type</th>
<th>VIVE</th>
<th>VIVE Pro</th>
</tr>
</thead>
<tbody>
<tr>
<td>OS</td>
<td>Windows 8.1, Windows 10</td>
<td></td>
</tr>
<tr>
<td>CPU</td>
<td>Intel Core i5-4590 or later, or AMD FX 8350 or later</td>
<td></td>
</tr>
<tr>
<td>Memory</td>
<td>4GB or more</td>
<td></td>
</tr>
<tr>
<td>Graphics</td>
<td>NVIDIA GeForce GTX 1060 or later or AMD Radeion RX480 or later</td>
<td></td>
</tr>
<tr>
<td>Video output</td>
<td>1 x HDMI 1.4 or higher or DisplayPort 1.2 or higher</td>
<td>Displayport 1.2 or higher</td>
</tr>
<tr>
<td>USB</td>
<td>1 x USB 2.0 port or more</td>
<td>USB 3.0 or higher</td>
</tr>
</tbody>
</table>

UC-win/Road requirements are detailed on FORUM8 website.

<table>
<thead>
<tr>
<th>OS</th>
<th>Windows 8 /10 (recommended 64bitOS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU</td>
<td>Intel Core i7 Quad-core or more. 3.2GHz or more.</td>
</tr>
<tr>
<td>Memory</td>
<td>8GB or more.</td>
</tr>
<tr>
<td>Graphics</td>
<td>NVIDIA GeForce GTX 1070 or higher. 8GB or more of Video memory.</td>
</tr>
</tbody>
</table>

To use HTC Vive, Steam and Steam VR have to be installed. Follow the procedure below to install and configure. Steam VR can be started from the plugin.

1. Installation of Steam
Download the installer from Steam's official site and install according to the installer's instructions.
A Steam account is required. Therefore, create an account and login.

2. Installation of Steam VR
Start Steam and select "Library - Tools". Right-click on "SteamVR" in the list to install.
3. Installation of HTC Vive base station
See Vive’s official website for further details.

4. Room Setup
Connect to HMD and start Steam VR, and lunch Room Setup. Select Room Setup from the menu on the top left. In Room Setting, configure default settings such as origin point.

And configure the screen setting.

5. Cooperation with UC-win/Road
The following procedure details how to start Vive plug-in.

1. Activate HTC Vive Plugin
Start UC-win/Road. From the menu, select File - License Manager and enable HTC Vive Plugin.

2. HTC Vive ribbon menu
Once enabled, the “HTC Vive” group appears in the “Devices” tab of the Ribbon menu.
Run / Pause: Start connecting with SteamVR.

Run / Stop: Start/stop cooperation with HTC Vive. This action can be performed only once a project file (*.rd) is loaded.

Reset view: The coordinate system of the HMD can be initialized so it is possible to choose the direction of the front view. However, roll and pitch are chosen according to the direction of gravity. Hint: it is available during connection with StreamVR.

In navigation mode, the coordinate system behave as follows:
• Free mode, walking mode: Yaw angle is the one of the camera, roll and pitch angle are set to 0.
• Driving mode, flight mode: the coordinate system of the vehicle is used.

Setting: Display HTC Vive setting form.

6. HTC Vive Setting Form - HTC Vive plug-in
This form describes how to configure FOV information for HTC Vive, displays and sets the device linked with SteamVR and UC-win/Road and how to display the menu, when using the HTC Vive plug-in.

7. Camera operation function
Camera operation is performed by the controller device. When multiple controller devices are connected, it is possible to do the same operation on all controllers.
Hint: Only when HTC Vive is linked, the camera operation function is effective.

Touch pad Up/Down Move to up or down.
Touch pad Left/Right Left to move to the parent menu, right to move to the child menu.
Trigger Confirm the selection. Start operation.

8. Menu function
When HTC Vive is connected, the menus are displayed. The menu screen is displayed when pressing the application menu button on the controller device. It is also possible to hide it by pressing the same button.

9. Link with Scenario
It is possible to connect HTC Vive with the scenario of UC-win/Road.

Scenario Event Function
Use the scenario event function in UC-win/Road to set up HTC Vive. Go to HTC Vive tab on the extension functions tab.

Enable HMD: Start displaying UC-win/Road scenes on HMD.
Disable HMD: Stop displaying UC-win/Road scenes on HMD.
Reset view: Reset the view.
Controller Vibration: Check it to enable controller vibration.
  - All controllers
    Enable controller vibration for all controllers.
  - Assigned controller
    Specify a custom number of controller to be enabled vibration.
  - Vibration strength
    Set up the vibration strength from 100 to 4000
  - Duration
    Set up the duration time of vibration.

Scenario Trigger Function
It is possible to use scenario trigger function with HTC Vive.
After loading HTC Vive Plugin, the conditions VIVE device collision and VIVE device input are added.

Collusion trigger condition
The collision of a device model and a model in the scene is a trigger. The collision judge is the same as the one of UC-win/Road.
Input Trigger Condition

The operation state of the button of the device is a trigger.

In this case, the controller with custom ID 0 is targeted. And the first trigger condition is that the input of track pad is from -0.5 to 0.5, the second one is that the track pad is touched and the grip button is pushed.

All devices

Check All devices to apply the collision judge to all connected HTC Vive devices. Otherwise, select types to be applied the collision judge to.

- Type
  Select (a) type(s) of devices from Controller, Tracker, or Sensor.

- All
  Check All to apply the collision judge to all devices of the type. Otherwise, specify any device models.

- ID
  Set up the custom ID for the device.

- Condition
  True means the collision becomes the trigger.

  - Type: Select (a) type(s) of devices from Controller, Tracker, or Sensor.
  - All: Check All to apply the collision judge to all devices of the type. Otherwise, specify any device models.
  - ID: Set up the custom ID for the device.
  - Condition: Condition is displayed in tree. The node displays the condition groups divided by "or" or "and". Select a group and click Add to add a new condition. Use the right conbobox to switch "or" and "and". Click Delete to delete the selected group or condition. The root node cannot be deleted. Click Add or Edit button to open the setting form to set up a condition.
Category
Select Button or Shaft.

- Shaft: Choose a target from Track pad X, Track pad Y or Trigger. And edit the formula.
- Button: Choose a target button from Menu, Grip, Track pad, or Trigger. And choose a trigger from pushed, touched or released. Touched can be selected for Track pad or Trigger.

*Current limitation on HTC Vive plug-in
The current version of HTC Vive plug-in has the following restrictions.

- Solar flare and whiteout are not drawn correctly.
- Windshield wipers and rain on the windshield are not drawn in 3D.
- HUD display (scenario message, image, speedometer etc.) is not drawn. But in this plugin, only the digital speed meter is supported.
- Jump and rotation with right click functions do not behave correctly.
Custom Shader Sample Plug-in

The modified drawing processing has enabled the customization of rendering process.
- "Custom Shader Sample Renderer" has been added as an example of Shader customization.
- Coloring based on data of object type, drawing normal, depth, object speed, and acceleration, etc.
- Visualization of various information, switching of display methods, generation of segmentation teacher data for deep learning.

Left: Segmentation by object selection, Right: Speed information

How to use

After installing Custom Shader Sample Plug-in, the Custom Shader sample renderer is added to renderer.
It can be used on the main screen or viewpoint.

Restrictions

1) Semitransparent polygon does not support the segmentation by object selection. It is drawn in the default color (white).
2) Custom shader Sample does not support the reflection of water on road surface.
3) The shader used in some plugins, such as XPSWMM and tsunami plugin, cannot be changed to Custom Shader Sample now.

From Renderer settings of main screen

It is possible to change the renderer as followings.
1. Right click on the main screen, and click Renderer settings.
2. Choose Custom Shader Sample Renderer as renderer.
3. Set up the setting items. Refer to Renderer settings for details.
4. Click OK to confirm the settings. The main screen changes the screen rendered by Custom Shader Sample Renderer.
5. Choose Default Renderer as renderer to go back to the normal screen.
From View Editor
Choose Custom Shader Sample Renderer as renderer. Set up the setting items on the right. Refer to Renderer setting for details.

Click OK to confirm the settings.
Apply this view settings from ribbon menu or right click menu to display the screen coloring based on the data.
The parameters of coloring can be edited from the ribbon menu View – Custom Shader Sample – Settings.

Renderer settings

How to open
• Right-click on the main view and go to Renderer settings.
• Select a Current position or Move camera to, and click Edit.
Descriptions

Shader path: Click Browse to select a shader folder.
The default is <Install folder>¥Shaders¥Plugins¥CustomShaderSample.
Click Apply to confirm the settings. Click Reset and then Apply to reset the settings.

Compile shader for every frame
Check it to compile the shader for every frame. If the shader has been edited, check it to reflect the effects.

Type: Choose a coloring type.
- Object type: Coloring based on object type.
- Vector: RGB Coloring based on the absolute values of X, Y, Z of the normal vector.
- Depth: Coloring based on depth using the inverse number of depth buffer.
- Speed: Coloring based on speed information of vehicles and characters.
- Acceleration: Coloring based on vehicle acceleration.
- Distance: Coloring based on distance from camera position.
- Height from ground: Coloring based on height from ground.
- Custom: this is an index to customize a shader.

Max. Min.
Set up the max and min values of colors. Red is for max, blue is for min. And the color between red and blue are drawn in gradation.

Note: When Object type or Vector is selected as Type, it is not possible to set up max. and min. values.

Custom Shader Sample Property
Set up the properties about Custom Shader Sample Renderer. There are 2 types: Global parameter and Object property.
Hint:
Now the first property of Object property is used. Other parameters can be used to customize the shader file.
To edit the colors used in coloring by object selection, choose Color by Object type as Property, set up the values of the default value, model instance type, model group and road parts at the range of -1 – 360.
How to open
Go to the ribbon menu View – Custom Shader Sample – Settings.

Descriptions
Global Parameters
Set up 16 parameters sent to the shader. Now they are not in use for Custom Shader Sample. It is possible to use them for the customized shader.

Object properties
Set up 4 properties by object type. One of them is used for coloring by object selection. 1-3 properties are not in use now. It is possible to use them for the customized shader.
Rendering of 360-degree image
It is possible to render 360 degree video with the general Equirectangular projection on UC-win/Road. By saving as a still image or recording a video, it is possible to create images and videos which are used for explanation with the camera positions changeable in any direction of 360 degrees, and simple VR experience videos that support stereoscopic view.

(1) Apply 360 degree image
Set up the renderer settings to change UC-win/Road renderer to 360 degree image. Refer to How to apply 360 degree image.

Restrictions
1. The reflection of puddle is not supported.
2. When shader is used in plugins such as XPSWMM or Tsunami plugin, Stereo 360 degree image is not supported.
3. Stereo may not reflect on objects near the camera position. It can be resolved by dividing the polygon.

(2) Output 360 degree video
360 degree video is output in mp4 format. Refer to Output of 360 degree video.

How to apply 360 degree image
Change the renderer settings to apply 360 degree image.

(1) From Renderer setting on the main screen
1. Right click on the main view, and go to Renderer setting.
2. Choose 360 degree image as a renderer.
3. Set up the setting items. Refer to Renderer setting.
4. Click OK to confirm the settings. The screen changes to 360 degree image.
5. Choose Default renderer to switch to the original screen.

(2) From View Editor
Choose 360 degree renderer as a renderer on View editor to display 360 degree image on the main screen or camera view. Then set up the setting items. Click OK to confirm the settings.

Note:
On the view editor, the renderer settings are not changed as the default from UC-win/Road ver.14.
It means the view, which has not any renderer settings, is applied with rendering of 360 degree image on the main screen or camera view rendered with 360 degree image. Choose Default renderer as a renderer on the view editor not to apply 360 degree image rendering.
Output of 360 degree video

360 degree video is output in mp4 file format. Here the open source license x264vfw is used.

Note: it is not necessary to use it for those who have software to convert AVI to MP4

How to download and install x264vfw

2. Start installation. Install x264 vfw (x64) for UC-win/Road 64bit or x264vfw (x86) for UC-win/Road 32bit.

How to download Spatial Media Metadata Injector

Download Spatial Media Metadata Injector v2.1. https://github.com/google/spatial-media/releases/tag/v2.1

AVI settings on UC-win/Road

Go to the ribbon Record/Play – Movie – Options to open AVI Options.
Set up the aspect ratio 1:2 for 360 degree video or 1:1 for stereo. And select X264 : x264vfw – H.264/MPEG-4 AVC codec as Codec.
Click Configure to open the setting form of x264vfw. Check Zero Latency on Basic. Select File mode as output and specify the destination of mp4 file. Check on VirtualDub Hack. And then click OK to confirm the settings.

**Recording 360 degree video**

1. Click on the main screen to open the rendering setting form.
2. Select 360 degree image renderer as Renderer. It is recommended to use the default settings, but uncheck Stereo if Monocular 360 degree video will be created.
3. Click OK to confirm the settings. The main screen will change to 360 degree image.
4. Go to the menu Record/Play – Movie – Start to start recording.
5. To record the script, play it now.
6. Go to Record/Play – Movie – Stop to stop recording.
7. Confirm the mp4 file is output to the destination folder.

**Metadata Injector**

1. Run Spatial Media Metadata Injector. Click Open to open the 360 video.

2. After loading, the message ‘Current 360 video: ~.mp4’ is appeared. When it is output as stereo, check stereoscopic 3D (top/bottom layout).

3. Click Inject metadata.

4. After injecting, the message ‘Successfully saved file to ~.mp4’ is appeared.

5. It is possible to display it on the viewer. For example, the direction of video can be changed freely with Windows 10 Movie&TV application.
**Renderer setting**

Choose 360 degree image as renderer.

**Stereo**

- **Stereo (Top-bottom)**
  Check it to create 2 images for left eye and right eye. Uncheck it to create a single image from the camera position.

  **Interval between eyes**
  Set up the interval between eyes in mm.

**Cube map texture**

- **Texture size**
  Set up the size of cube map texture. The bigger it is, the more beautifully the output image, but the longer it takes time.

  **Multi Sampling anti-aliasing**
  Enable or disable multi sampling anti-aliasing for cube map texture. Check it to suppress the jaggies, which may take longer.

  **Sampling number**
  Set up the number of sampling for multi sampling anti-aliasing. The bigger it is, the longer it takes time.

**360 degree image**

- **Reduction filter**
  Choose a filtering method to reduce the texture of cube map. The quality of image is better with Linear than with Nearest. And other methods takes longer time.

- **Expansion filter**
  Choose a filtering method to expand the texture of cube map.

- **LOD bias**
  Use it to adjust the image. Input a value in minus to improve the resolution of mipmap texture.

**HUD Settings**

- **Use HUD**
  Check it to display the image with HUD, which is the overlapped display, such as scenario message or the display of dashboard.

- **HUD Type**
  Choose a HUD type to display the overlap from Plane, Four quarters or Column.

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plane (Front)</td>
<td>HUD display is projected on the square-shaped plane in front. It is possible to edit the distance to the plane, offsets, size of the plane.</td>
</tr>
</tbody>
</table>
**Plane (Front and back)**
HUD display is projected on the square-shaped planes in front and back. It is possible to edit the distance to the planes, offsets, size of the planes.

**Four Planes**
HUD display is projected on the four planes. It is possible to edit the distance to the planes, offsets and the size of the plane.

**Column**
HUD display is projected on the column. It is possible to edit the distance to the surface, offset, and height of the column. The vertical length of the column is calculated with the distance to the column×π/2.

- **Texture size**: Set up the texture size.
- **Distance**: Set up the distance to the surface set as HUD type.
- **Surface Size**: Set up the surface size set as HUD type. Depending on the type, it is possible to edit the width, height.
- **Surface offset**: Set up the offset of surface set as HUD type. Depending on the type, it is possible to edit the vertical and horizontal directions.

**Reset**: Reset the settings.
UC—win/Road Ver.14 Operation Guidance

October 2020

Published by FORUM8 Co., Ltd.
TEL +81-3-6894-1888

Should you have any questions about this product or document, please contact our support center (information below). Answer will be sent to you from 9 to 12 AM / from 1 to 5 PM (from Mon. to Fri.; in Japan time).

Contact

Homepage www.forum8.co.jp
Support ic@forum8.co.jp
fax 0985-55-3027

Use this system after fully confirming that it is suitable for your business. FORUM8 has no responsibility for any damage including lost profits, claim by third parties, or other incidental or consequential damages arising out of, or relating to your use of this system.

All brand names and product names are registered trademarks or trademarks of their respective companies.

Copyright(C) FORUM8 Co., Ltd.