PTV VISSIM

the mind of movement

PTV VISSIM 11

USER MANUAL

Copyright and legal agreements

Copyright

Legal agreements

The information contained in this documentation is subject to change without notice and should not be construed as a commitment on the part of PTV AG.

Without the prior written permission of PTV AG, this documentation may neither be reproduced, stored in a retrieval system, nor transmitted in any form or by any means, electronically, mechanically, photocopying, recording, or otherwise, except for the buyer's personal use.

Warranty restriction

The content accuracy is not warranted. Any information regarding mistakes in this manual is greatly appreciated.

Imprint

PTV AG

Haid-und-Neu-Str. 15

76131 Karlsruhe

Germany

Tel. +49 721 9651-300

info@vision.ptvgroup.com

www.ptvgroup.com

vision-traffic.ptvgroup.com

Last amended: 24.09.2018 EN

11.00

Contents

Important changes compared to previous versions

With the following changes and new features, the behavior of Vissim is very different to that of previous versions.

You can find a complete list of the new features and changes to the current version in your Vissim installation in the directory *..\Doc\<language ID>* in the file *ReleaseNotes_VISSIM_<language ID>.pdf*.

Versions before Vissim 11

- Previous versions of Vissim are available in both 32-bit and 64-bit editions.

  Vissim 11 is available as a 64-bit edition only.

  - This means that to use Vissim 11, you will need to use a 64-bit operating system.
  - If you are applying a 32-bit control procedure in Vissim 11, contact the provider of the respective control procedure and ask for a 64-bit version of the same. The DLL files of control procedures that are provided with PTV GROUP are available in a 64-bit version.

- Vehicle simulation: The vehicle attributes **Headway**, **Leading target type** and **Leading target number** may show different values than in earlier versions if vehicles are controlled via COM or the driving simulator interface.

Versions before Vissim 10

- In versions prior to Vissim 10, the *Discontinued models* directory is installed in the installation directory of Vissim, under *..\Exe\3DModels\Vehicles* and *..\Exe\3DModels\Pedestrians*.

  From Vissim 10, the *Discontinued models* directory is no longer installed. To use 3D models of this directory in Vissim 10, save the 3D models of the version prior to Vissim 10. Then after installing Vissim 10, copy them into the directory where the **.inpx* file is saved.

Versions prior to Vissim 9.00-03

- In previous versions of Vissim 9.00-03, a route location on a ramp or stairway has no direction defined for its use by pedestrians. From Vissim 9.00-03, a route location defines a direction for several cases (see "Modeling the course of pedestrian routes using intermediate points" on page 957).

Versions before Vissim 9

- In versions prior to Vissim 9, the origin-destination matrix for dynamic assignment is saved to **.fma* file. From Vissim 9 on, the origin-destination matrix is saved to a matrix in Vissim, it can be shown in the **Matrices** list and edited in the matrix editor.

- To access the Help in versions prior to Vissim 9, from the **Help** menu, choose > **PTV Vissim Help**. From Vissim 9, you can show the Help page (including attribute descriptions) for some windows. To do so, in the respective window, press the F1 button or click the **?** symbol.

Versions prior to Vissim 8.00-14 and Vissim 9.00-03

- In previous versions of Vissim, selecting the path pre-selection options **Reject paths with too high total cost** and **Limit number of paths** meant that paths were deleted from the path collection/path file. From Vissim 8.00-14 and Vissim 9.00-03, selecting these options only means that the corresponding paths will not be used in the respective time interval.

Versions before Vissim 8

- In previous versions of Viswalk, for pedestrians, you could select **Never walk back**. This attribute is no longer available. If the attribute is still activated in older entry data, the attribute is deactivated when imported.
- In previous versions, licenses could not be managed within Vissim. This is now possible from Vissim 8 (see "Program start and start screen" on page 54).
- The simulation results of Vissim 7 and Vissim 8 may differ, as e.g. the departure times from vehicle inputs, parking lots and of PT lines were made uniform and for some special cases, an improved driving behavior was integrated.

Versions before Vissim 7

- In previous versions, the point was used as decimal separator. From Vissim 7, the decimal separator in lists depends on the settings in the control panel of your operating system (see "Selecting decimal separator via the control panel" on page 148).
- In previous versions, the color of the vehicle status could be toggled during a simulation run by pressing CTRL+V. From Vissim 7, this is possible with the key combination CTRL+E (see "Dynamically assigning a color to vehicles during the simulation" on page 175).

Quick start: creating a network and starting simulation

Quick start shows you the most important steps that allow you to define base data, create a network, make the necessary settings for simulation, and start simulation.

1. Opening Vissim and saving a new network file
2. Defining simulation parameters (see "Defining simulation parameters" on page 840)
3. Defining desired speed distribution (see "Using desired speed distributions" on page 237)
4. Defining vehicle types (see "Using vehicle types" on page 267)
5. Defining vehicle compositions (see "Modeling vehicle compositions" on page 452)
6. Loading the project area map as a background image (see "Inserting a background image" on page 394)
7. Positioning, scaling, and saving the background image (see "Positioning background image" on page 402). Scaling as precisely as possible (see "Scaling the background image" on page 403).
8. Drawing links and connectors for lanes and crosswalks (see "Modeling links for vehicles and pedestrians" on page 406), (see "Modeling connectors" on page 420)
9. Entering vehicle inputs at the end points of the network (see "Modeling vehicle inputs for private transportation" on page 454). If you are using pedestrian simulation: defining pedestrian flows at crosswalks (see "Modeling pedestrian inputs" on page 936).
10. Entering routing decisions and the corresponding routes (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459). If you are using pedestrian simulation, you can also specify the following for pedestrians (see "Static pedestrian routes, partial pedestrian routes and pedestrian routing decisions" on page 940).
11. Defining changes to the desired speed (see "Using reduced speed areas to modify desired speed" on page 435), (see "Using desired speed to modify desired speed decisions" on page 440)
12. Editing conflict areas at non-signalized intersections (see "Using conflict areas" on page 560). You may enter priority rules for special cases (see "Modeling priority rules" on page 541).
13. Defining stop signs at non-signalized intersections (see " Modeling stop signs and toll counters" on page 571)
14. Defining SC with signal groups, entering or selecting times for fixed time controllers, e.g. VAP or RBC (see "Modeling signal controllers" on page 577)
15. Inserting signal heads (see "Modeling signal groups and signal heads" on page 578)
16. Creating detectors at intersections with traffic-actuated signal control (see "Using detectors" on page 593)

17. Inserting stop signs for right turning vehicles at red light (see " Using stop signs for right turning vehicles even if red" on page 575)
18. Entering priority rules for left turning vehicles in conflict at red light and crosswalks (see "Modeling priority rules" on page 541).
19. Defining dwell time distributions (see "Using time distributions" on page 246). Inserting PT stops in the network (see "Modeling PT stops" on page 511)
20. Defining PT lines (see "Modeling PT lines" on page 518)
21. Activating evaluations, e.g. travel times, delays, queue counter, measurements (see "Performing evaluations" on page 1001)
22. Performing simulations (see "Selecting the number of simulation runs and starting simulation" on page 845)

Typography and conventions

To make it easier for you to identify individual GUI elements in the manual, we have used the following typography throughout the document.

Element	Description
Program elements	Elements of the graphical user interface are bold-formatted: ▶ Names of windows and tabbed pages ▶ Entries in menus and selection lists ▶ Names of options, window sections, buttons, input fields and icons
`Input data`, `output data`, `Code examples`	Data that is entered, output or used as a code example is formatted in a different font.
KEYS	Keys you need to press are printed in capital letters, e.g. CTRL + C.
Path and file name data	Directory paths and file names are printed in italics, e.g. *C:\Program Files\PTV Vision\PTV Vissim <Version number>\Doc*.

Prompts for actions and results of actions

- If just a single step is required to solve a task, the paraphrase is indicated by an arrow.

1. In case of multiple steps to be done, these are numbered consecutively.

   If the prompt for an action is followed by a visible intermediate result this result is listed in italic format.

Also the final result of an action appears in italic format.

Warnings, notes and tips for using the program

Warning: Warnings might indicate data loss.

Note: Notes provide either information on possible consequences caused by an action or background information on the program logic.

Tip: Tips contain alternative methods for operating the program.

Using the mouse buttons

By default, **click** means left mouse click, e.g.:

1. Click the **Open** button.

If you need to use the right mouse button, you are explicitly asked to do so, e.g.:

- Right-click in the list.

Tip: In Network editors, by default a right-click opens the shortcut menu. However, you can choose to have a network object inserted instead. The right-click was used to insert network objects in versions prior to Vissim 6 (see "Right-click behavior and action after creating an object" on page 152).

Names of network object attributes

The attributes of network objects that are displayed by default in the windows of the program interface or in the attribute lists are described in tables. The first column lists the attribute name as used in the program interface, e.g. **Vehicle record**. If the short or long name of the attribute is different, these names are listed in the other columns together with a description of the attribute, e.g. **Vehicle record active** (**VehRecAct**). In the attribute lists provided of the user interface, you can show additional or hide existing attributes (see "Selecting attributes and subattributes for columns of a list" on page 112).

By default, a list of all tables, attributes, enumeration types and relations of Vissim is located in the Vissim installation directory, under *..\Doc\Eng*, in the *attribute.xlsx* file.

1 Introduction

PTV Vissim is the leading microscopic simulation program for modeling multimodal transport operations and belongs to the Vision Traffic Suite software.

Realistic and accurate in every detail, Vissim creates the best conditions for you to test different traffic scenarios before their realization.

Vissim is now being used worldwide by the public sector, consulting firms and universities.

In addition to the simulation of vehicles by default, you can also use Vissim to perform simulations of pedestrians based on the Wiedemann model (see "Version-specific functions of pedestrian simulation" on page 861).

1.1 Simulation of pedestrians with PTV Viswalk

PTV Viswalk is the leading software for pedestrian simulation. Based on the Social Force Model by Prof. Dr. Dirk Helbing, it reproduces the human walking behavior realistically and reliably. This software solution with powerful features is used when it is necessary to simulate and analyze pedestrian flows, be it outdoors or indoors.

Viswalk is designed for all those who wish to take into account the needs of pedestrians in their projects or studies, for example for traffic planners and traffic consultants, architects and owners of publicly accessible properties, event managers and fire safety officers.

Using PTV Viswalk alone, however, you cannot simulate vehicle flows. To simulate vehicle and pedestrian flows, you need Vissim and the add-on module PTV Viswalk. You can then choose whether to use the modeling approach of Helbing or Wiedemann.

1.2 PTV Vissim use cases

Vissim is a microscopic, time step oriented, and behavior-based simulation tool for modeling urban and rural traffic as well as pedestrian flows.

Besides private transportation (PrT), you may also model rail- and road-based public transportation (PuT).

The traffic flow is simulated under various constraints of lane distribution, vehicle composition, signal control, and the recording of PrT and PT vehicles.

Vissim allows you to test and analyze the interaction between systems, such as adaptive signal controls and route recommendation in networks.

Simulate the interaction between pedestrian streams and local public and private transport, or plan the evacuation of buildings and entire stadiums.

Vissim may be deployed to answer various issues. The following use cases represent a few possible areas of application:

Comparison of junction geometry

- Model various junction geometries
- Simulate the traffic for multiple node variations
- Account for the interdependency of different modes of transport (motorized, rail, cyclists, pedestrians)
- Analyze numerous planning variants regarding level of service, delays or queue length
- Graphical depiction of traffic flows

Traffic development planning

- Model and analyze the impact of urban development plans
- Have the software support you in setting up and coordinating construction sites
- Benefit from the simulation of pedestrians inside and outside buildings
- Simulate parking search, the size of parking lots, and their impact on parking behavior

Capacity analysis

- Realistically model traffic flows at complex intersection systems
- Account for and graphically depict the impact of throngs of arriving traffic, interlacing traffic flows between intersections, and irregular intergreen times

Traffic control systems

- Investigate and visualize traffic on a microscopic level
- Analyze simulations regarding numerous traffic parameters (for example speed, queue length, travel time, delays)
- Examine the impact of traffic-actuated control and variable message signs
- Develop actions to speed up the traffic flow

Signal systems operations and re-timing studies

- Simulate travel demand scenarios for signalized intersections
- Analyze traffic-actuated control with efficient data input, even for complex algorithms
- Create and simulate construction and signal plans for traffic calming before starting implementation
- Vissim provides numerous test functions that allow you to check the impact of signal controls

Public transit simulation

- Model all details for bus, tram, subway, light rail transit, and commuter rail operations
- Analyze transit specific operational improvements, by using built-in industry standard signal priority
- Simulate and compare several approaches, showing different courses for special public transport lanes and different stop locations (during preliminary draft phase)
- Test and optimize switchable, traffic-actuated signal controls with public transport priority (during implementation planning)

1.3 Traffic flow model and light signal control

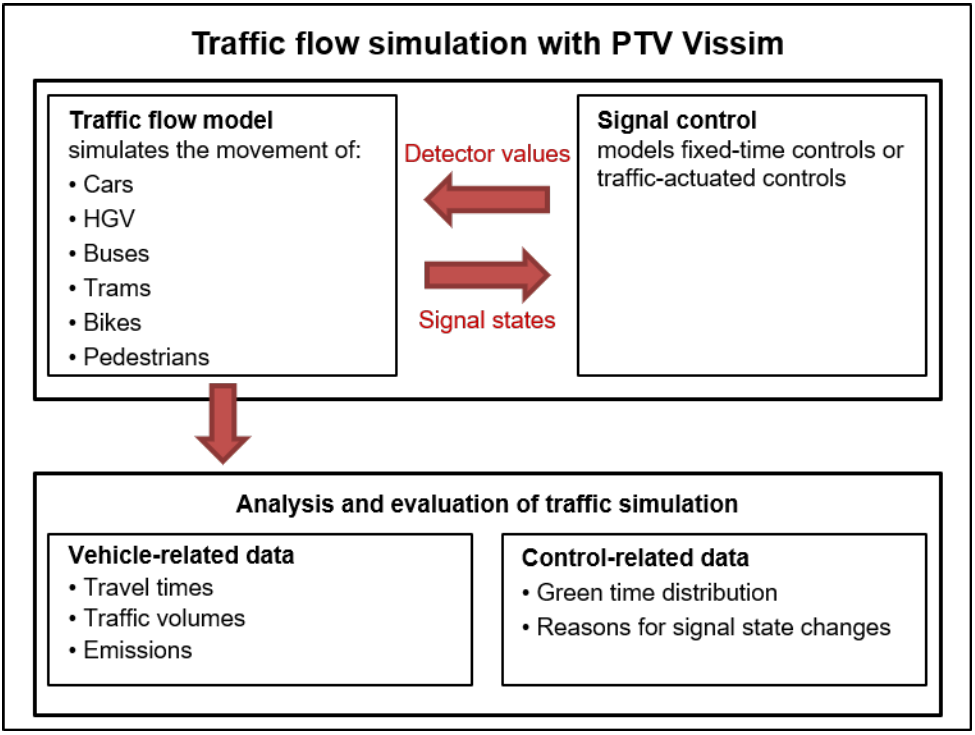
Vissim is based on a **traffic flow model** and the **light signal control**. These exchange detector readings and signaling status.

You can run the traffic flow simulation of vehicles or pedestrians as animation in Vissim. You can clearly display many important vehicular parameters in windows or you can output them in files or databases, for example, travel time distributions and delay distributions differentiated by user groups.

The **traffic flow model** is based on a car-following model (for the modeling of driving in a stream on a single lane) and on a lane changing model.

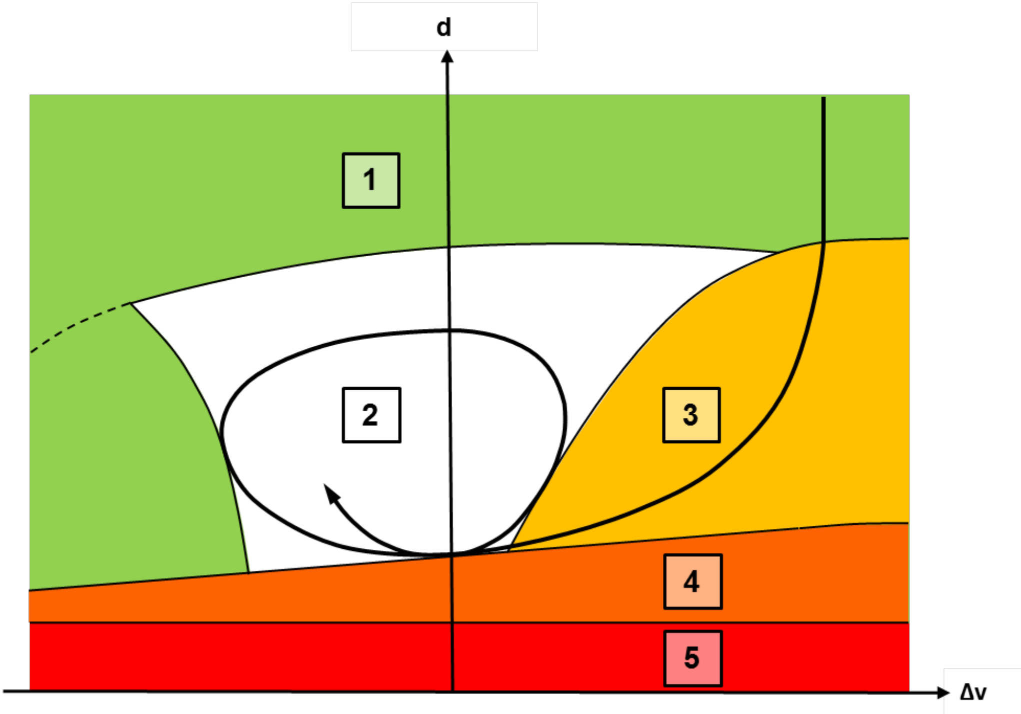
External programs for **light signal control** model the traffic-dependent control logic units. The control logic units query detector readings in time steps of one to 1/10 second. You can define the time steps for that reason and they depend on the signal control type. Using detector readings, e.g. occupancy and time gap data, the control logic units determine the signaling status of all signals for the next time step and deliver them back to the traffic flow simulation. Vissim can use multiple and also diverse external signal control programs in one simulation, for example, VAP, VSPLUS.

Communication between traffic flow model and traffic signal control:



1.3.1 Operating principles of the car following model

Vehicles are moving in the network using a traffic flow model. The quality of the traffic flow model is essential for the quality of the simulation. In contrast to simpler models in which a largely constant speed and a deterministic car following logic are provided, Vissim uses the psycho-physical perception model developed by Wiedemann (1974) (see "Driving states in the traffic flow model according to Wiedemann" on page 285). The basic concept of this model is that the driver of a faster moving vehicle starts to decelerate as he reaches his individual perception threshold to a slower moving vehicle. Since he cannot exactly determine the speed of that vehicle, his speed will fall below that vehicle's speed until he starts to slightly accelerate again after reaching another perception threshold. There is a slight and steady acceleration and deceleration. The different driver behavior is taken into consideration with distribution functions of the speed and distance behavior.



Car following model (according to: Wiedemann 1974)

Legend

Axes: **d**: Distance, **Δv**: Change in speed	**3**: Approaching state
1: "Free flow" state	**4**: Braking state
2: Following state	**5**: Collision state

The car following model has been calibrated through multiple measurements at the Institute of transport studies of the Karlsruhe Institute of Technology (since 2009 KIT – Karlsruhe Institute of Technology), Germany. Recent measurements ensure that changes in driving behavior and technical capabilities of the vehicles are accounted for.

Vissim calculates the acceleration of a vehicle during free traffic flow, below the desired speed, based on the following:

- If the desired safety distance is set to 100 percent, the vehicle drives at the same speed as its preceding vehicle.
- If the desired safety distance is set to between 100 and 110 percent, the speed is interpolated between the vehicle's desired speed and the speed of its preceding vehicle.
- If the desired safety distance is set to greater than or equal to 110 percent, the vehicle accelerates at its desired speed.

For multi-lane roadways a driver in the Vissim model takes into account not only the vehicles ahead (default: 4 vehicles), but also the vehicles in the two adjacent lanes. In addition, a signal control for about 100 meters before reaching the stop line leads to increased attention of the driver.

Vissim simulates the traffic flow by moving **driver-vehicle-units** through a network. Every driver with his specific behavior characteristics is assigned to a specific vehicle. As a consequence, the driving behavior corresponds to the technical capabilities of his vehicle. Attributes characterizing each driver-vehicle unit can be subdivided into following three categories:

- Technical specification of the vehicle, for example:
  - Vehicle length
  - Maximum speed
  - Accelerating power

and:

  - Actual vehicle position in the network
  - Actual speed and acceleration
- Behavior of driver-vehicle units, for example:
  - Psycho-physical perception thresholds of the driver, e.g. ability to estimate, perception of security, willingness to take risk
  - Driver memory
  - Acceleration based on current speed and driver’s desired speed
- Interdependence of driver-vehicle units, for example:
  - Reference to vehicles in front and trailing vehicles on own and adjacent lanes
  - Reference to currently used network segment and next node
  - Reference to next traffic signal

1.4 How to install and start PTV Vissim

Vissim is provided electronically or on a DVD with demo files, examples files, its documentation, and other license dependent files.

1.4.1 Information on installation and deinstallation

For information on program installation and deinstallation, refer to *Vissim <Version> - Installation Manual.pdf*.

- The installation guide is on the DVD, in the *..\OnCD\Documentation\Eng* folder.
- After the installation, you will find the installation guide in your Vissim installation, under *..\Doc\Eng*.

1.4.2 Content of the PTV Vision program group

After you have installed Vissim, several icons are displayed in the PTV Vision program group, depending on your license and the options chosen during the installation process. To call the program, from the **Start** menu, choose > **All Programs** > **PTV Vision <year>** > **PTV Vissim <version>** (navigation of the Start menu depends on your operating system).

Element	Description
Vissim <Version>	Opens Vissim: • with an empty network, if no default network is saved. • Opens the program with this network, if a default network is saved (see "Overview of menus" on page 121)
V3DM	Opens the add-on module V3DM (Vissim 3D modeler). Using V3DM, you can convert 3D models of the file formats DWF (Design Web Format), 3DS (Autodesk), and SKP (SketchUp) into the Vissim 3D format V3D. This applies to static 3D models and vehicle models (see "Using static 3D models" on page 674).
VISVAP	Opens the add on module VISVAP. VisVAP is tool that allows you to easily create the program logic of a VAP signal control as a flow chart.

1.4.3 Specifying the behavior of the right mouse button when starting the program for the first time

When you start Vissim for the first time, the window **Right Click Behavior Definition** opens. To select the function that is carried out, right-click in the Network editor.

1. Select the desired entry.

Element	Description
Context menu	The context menu of the Network editor opens (see "Network editor context menu" on page 80).
Creating a new object	In the Network editor, insert a new network object of type selected in the Network objects toolbar. The context menu of the Network editor does not open. This is the behavior of Vissim versions released prior to version 6.

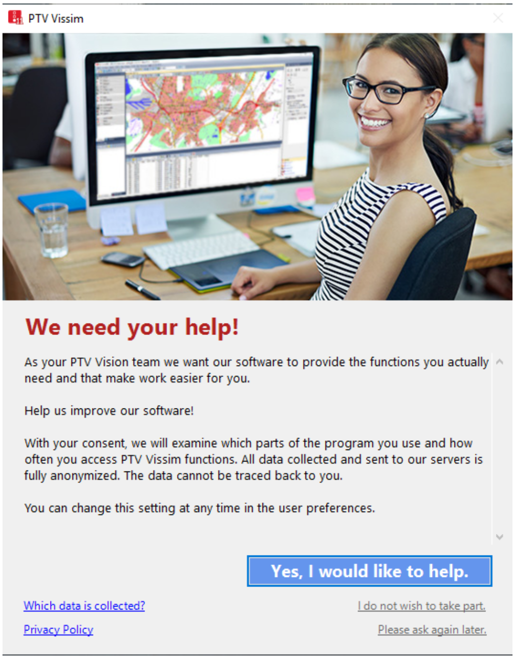
2. Confirm with **OK**.

Tips:

- If you hold down the CTRL key and right-click in the Network editor you are modeling in, you can switch to another function and execute it.
- You can change the right-click behavior again (see "Right-click behavior and action after creating an object" on page 152).

1.4.4 Agreeing to share diagnostics and usage data

With the third start of Vissim, the window **We need your help!** opens.



- Confirm with **Yes, I would like to help**.

You can deactivate the recording of diagnostics and usage data (see "Allowing the collection of usage data" on page 156).

1.5 Technical information and requirements

Current information on the technical requirements for the current Vissim version can be found on our website PTV Vissim FAQs. The following information, e.g., is essential:

- Entry (#3): What are the hardware and software requirements for Vissim?
- Entry (#6): Which graphics card has the best performance with Vissim 3D?

1.5.1 Criteria for simulation speed

The speed of a Vissim simulation primarily depends on the following criteria:

- Number of vehicles and/or pedestrians used simultaneously in the network
- Number of signal controls
- Type of signal controls
- Number of processor cores deployed
- Computer performance. On a faster computer, a simulation will run faster with the same network file.

Tip: With the 64-bit editions of Vissim and Viswalk, you can use a larger main memory of more than 3 GB RAM.

1.5.2 Main memory recommended

Vissim edition	Main memory recommended
64-bit	min. 4 GB RAM

The memory requirements may be considerably higher for the following use cases:

- Networks with many dynamically assigned paths
- Large applications with many pedestrian areas

1.5.3 Graphics card requirements

- If under **User Preferences** > **GUI** > **Graphics**, the graphics driver **OpenGL 3.0** is selected, make sure that your video card at least supports OpenGL 3.0.
- Ensure that the latest driver update has been installed for your graphics card. This way, you will avoid visualization problems when using the 3D mode. For many graphics adapters, you can download update drivers from the Internet.

Note: Should any problems arise with the 3D display in Vissim, first install the latest driver update for your graphics card before you contact PTV Vision Support.

1.5.4 Interfaces

Not all program interfaces are by default part of your Vissim license. They might be available as add-on modules (see "Add-on modules programming interfaces (API)" on page 1189). Add-on modules and their documentation are saved in the folder *..\Vissim<Version>\API*.

1.5.5 Number of characters of filename and path

- With Windows, the number of characters of a path, including drive letter and filename, is limited to 260.
- With Windows, the number of characters of a path is limited to 248.
- In Scenario Management, the length of the path to the project directory is limited to 214 characters.

▸ Make sure that your project files are not saved to any paths longer than that. This for instance includes the following paths:
  - Name of **.inpx* network file and path to the directory where the network file is saved.
  - Directories and subdirectories you create during a project Files saved to these directories
  - Directories and subdirectories that Vissim creates automatically or to which Vissim saves data, e.g. for scenario management or evaluations.

1.6 Overview of add-on modules

Vissim is available with the following add-on modules:

1.6.1 General modules

- **Dynamic assignment**

  The module Dynamic Assignment is used to distribute the vehicles automatically among the available paths. Users only need to specify an origin-destination matrix and the parking lots assigned to the corresponding zones. They are not required to enter static routes manually (see "Using the dynamic assignment add-on module" on page 692).

- **EnViVer Pro**

  EnViVer Pro is a tool used to calculate emissions based on vehicle record data.

- **EnViVer Enterprise**

  In addition to the functions of EnViVer Pro, EnViVer Enterprise allows for the modeling of additional vehicle classes, individual time periods as well as automatic processing of several input files.

- **3D packages: 3DS Max Export and V3DM**

  Export 3DS Max: Network data and vehicle positions are exported from Vissim as text files and can be imported into Autodesk's 3ds Max software. The 3D packet contains a script in the Autodesk macro language, as well as numerous vehicle model information.

  Using **V3DM** (Vissim 3D modeler), allows users to convert 3D models of the file formats DWF, 3DS (both by Autodesk), and SKP (SketchUp) into Vissim 3D format V3D. This works for both static models and vehicles.

  Example MAX files for vehicle models are located in the *..\API\3dsMaxExport\MAXModels* folder of your Vissim installation.

- **Synchro Import**

  Using this module, users can generate Vissim models from Synchro. The network geometry, volumes, turns, vehicle compositions, and signalization are imported. The module supports adaptive import. This means changes made in Vissim are not lost when more current versions of the Synchro model are imported.

- **Mesoscopic simulation**

  The module allows for significantly faster simulation of larger networks compared to microscopic simulation. The driving behavior is still based on individual vehicles and a temporal resolution of, e.g., a tenth of a second. The difference is that the individual vehicles are not looked at with each time step, but only when an event occurs. Such an event could, for instance, be a vehicle reaching the end of a route and traversing a node, or an SC switching to green.

  The main benefits of mesoscopic simulation are increased simulation speed and less time required for creating and calibrating the network.

If an area of the network still needs to be microscopically simulated, including all the details, hybrid simulation can be used. It allows you to select one or multiple sections for microscopic simulation, while the rest of the network undergoes mesoscopic simulation.

(see "Using add-on module for mesoscopic simulation" on page 801).

- **PTV Viswalk**

  Viswalk Is used for professional pedestrian simulation, either as a stand-alone solution or in combination with Vissim. The dynamic model is based on the Social Forces Model developed in 1995, inter alia, by Prof. Dirk Helbing. It allows pedestrians to walk independently to their destination, without a network model predefining their trajectories.

  A simple pedestrian simulation, based on directed routes (instead of areas), is included in Vissim. It is based on the car following model of Professor Wiedemann, as is the vehicle simulation. It does not require the Viswalk module (see "Pedestrian simulation" on page 860).

- **BIM Import**

  The **BIM Import** module converts BIM files (Building Information Model) of the data format IFC (Industry Foundation Class) into INPX files. These converted INPX files are meant for use in pedestrian simulation with Viswalk.

  Nearly every CAD software supports IFC export and thus provides an interface between the CAD software and Viswalk. The Importer converts slabs into areas, walls into obstacles and can import stairways, whilst keeping the level structure. Slabs with curves or holes are automatically optimized for use in Viswalk during data import (see "Importing Building Information Model files" on page 884).

1.6.2 Signal controllers: Complete procedures

- **Econolite ASC/3** (see "Add-on module Econolite ASC/3" on page 614)

  This module enables user to simulate signal controllers that run on ASC/3 North American controller devices by Econolite. It provides a dedicated user interface for its control parameters.

- **RBC (Ring Barrier Controller)** (see "Using the Ring Barrier Controller RBC add-on module" on page 619)

  This module enables PTV Vissim to simulate signal controllers that are controlled according to the North American standard procedure "ring barrier controller". It provides a dedicated user interface for the RBC parameters.

- **Signal control procedure vehicle-actuated programming (VAP)** (see "Add-on module Traffic-dependent VAP Programming" on page 626)

  VAP enables Vissim to simulate programmable vehicle-actuated signal controls (SC). This is possible for both stage or signal group based signal controls. During Vissim simulation runs or in the test mode, VAP interprets the control logic commands and generates the signal control commands for the signal control that become effective in the network. Vice versa, detector parameters are retrieved from the Vissim network and processed in the logic. The VAP program logic is described in a text file (*.vap*) with a simple programming language. It can be also be exported from VisVAP. The signal data file (**.pua*) can either be comfortably exported from Vissig or generated manually in a text editor. The range of application of VAP stretches from controls for individual nodes over PT priorities to complex control systems for entire corridors or subnetworks. Additionally, applications in the ITS range, e.g. variable message signs (VMS) or temporary side lane clearances are readily possible.

- **VisVAP**

  Flow chart editor for VAP: VisVAP (short for Visual VAP) is an easy to use tool for defining the program logic of VAP signal controllers as a flow chart. All VAP commands are listed in a function library. The export function allows users to generate VAP files, which saves additional changes to the VAP file. Moreover, VisVAP provides a debug functionality that during a running simulation in Vissim allows users to go through the control logic step by step using the control logic. It also shows the current values of all parameters used in the logic. To start VisVap, from the Start menu, choose > PTV Vision program group.

- **Vissig** (see "Opening and using the SC Editor" on page 631)

  Vissig complements the stage-based fixed time control (which is included in any basic Vissim version and in the Visum module "Junction editor and control") by additionally providing stage-based fixed time signal control. Vissig contains a graphical editor for defining stages and interstages. Interstages can also be automatically generated by Vissig. Besides providing the usual functionality, the signal program editor allows users to easily extend or shorten stages and interstages. Additionally, Vissig offers an interface for the export of signal data compliant with VAP in the PUA format so that a traffic-dependent signal control with VAP can be easily generated on the basis of the generated stages and interstages. All signal plan information can be exported to Microsoft Excel and easily added to reports.

- **Balance/Epics** (see "Using Balance-Central signal controllers" on page 612), (see "Using Epics/Balance-Local signal controllers" on page 613)

  PTV Balance is a comprehensive and proven adaptive transport network control software which is fully integrated into Vissim. Used in conjunction with the local adaptive node control PTV Epics or on its own, it calculates new signal plans for all nodes in the simulation network every 5 minutes based on the current detector data of the simulation.

  The module **balance/epics** allows you to simulate PTV Balance using Vissim, just like in the real application. The road network and transport demand data for PTV Balance are supplied by *.anm/.anmroutes*files and require PTV Visum. The signalization related parameters of PTV Balanceare supplied with an extended version of Vissig. PTV Balancecomes with a web-based and user-friendly program interface. It allows for a direct comparison of the calculated traffic parameters with the vehicles simulated in Vissim. Moreover, this supply does not contain any formatting and can also be used in real network control projects with PTV Balance.

  PTV Epics is a local, adaptive signal control, with a special focus on public transportation. It can be used instead of a fixed time or VAP signal control. You can simulate it using Vissim. Every second, the mathematical optimization function in PTV Epics uses current detector data to calculate the best signal plan for the next 100 seconds. It then transfers this signal plan to Vissim. All parameters required by PTV Epics are supplied by an extended version of Vissig. All modes of transport (private, public, pedestrian) are treated similarly, but can be weighted differently. This makes it particularly easy to implement acceleration in public transport with PTV Epics.

1.6.3 Signal control: Interfaces

- **External signal control SC**

  This module allows users to simulate signal controller procedures, which are available as a separate executable application (**.exe*) or program library (**.dll*). These can be either standard procedures supplied by PTV GROUP or other providers, or procedures developed internally with the API module.

- **LISA+ OMTC**

  This add-on is used to simulate signal controls specified with the LISA+ procedure by Schlothauer. The actual control DLL and the GUI for the controller parameters can be obtained from Schlothauer.

- **SCATS interface**

  This module is used to simulate signal controllers specified according to the Australian SCATS procedure. The actual control DLL and the GUI for entry of the control parameters (SCATS.DLL, SCATS_GUI.DLL, WinTraff, ScatSim) must be obtained from **Roads and Maritime Services** of New South Wales, Australia.

- **SCOOT interface**

  The SCOOT interface is used to simulate signal controllers that are specified according to the British SCOOT procedure. The actual control DLL and the GUI for the control parameters (SCOOT.DLL, SCOOT_GUI.DLL, PCScoot) can be obtained from Siemens UK.

1.6.4 Programming interfaces

- **API Package** Application Programmer's Interface:

  SignalControl, SignalGUI, DriverModel, and EmissionModel.DLL files. The API package enables users to integrate their own or external applications in order to influence a PTV Vissim simulation (see "Add-on modules programming interfaces (API)" on page 1189).

1.7 Using a demo version

- From version 11, Vissim is available as a 64-bit edition only.
- When you open a demo version, **Demo version** is displayed in the title bar.
- You can use an installed demo version for 30 days after it has first been opened.
- You can initially open the demo version during a maximum period of 180 days from its build.
- Demo version users do not receive any technical support.

- The Help and the manual can be opened via the **Help** menu. You can find answers to frequently asked questions about Vissim on our web pages PTV Vissim FAQs.
- A demo version does not allow you to print or save any data.
- The COM interface is not provided.
- Simulation runs are limited to 1800 simulation seconds.
- You can work with a demo version for a maximum of two hours.
- In the matrix editor, you cannot copy data.
- The command **Save as Default Network** is not available.
- If the demo version is overwritten by a later version of the service pack, the 30-day trial period starts anew.

1.8 Using PTV Vissim Viewer

Using the Vissim Viewer, you can, for instance, forward project data to your customers. The Vissim Viewer is a limited Vissim version, which needs to be additionally installed.

1.8.1 Limitations of the Vissim Viewer

- Network files cannot be saved.
- Evaluation files cannot be generated.
- Simulation runs are possible only for the first 1,800 s. This period cannot be extended in order to show longer simulation runs. If it is necessary to show the visualization of vehicles and/or pedestrians beyond the first 1,800 s, animation files *.ani* can be used. For animation files there is no time limit.
- The COM interface is not provided.
- There are no demo examples available.

1.8.2 Vissim Viewer installation and update

You can install or update the Vissim Viewer for 64-bit versions of Vissim. To do so, download the program from our website.

1. Open the web page PTV Vissim & PTV Viswalk Service Pack Download Area.
2. For your 64-bit version, click **<product name> (64 Bit) Service Packs**.

   *In the **Available Downloads** table, the entries are displayed sorted by:*

   - **Vissim Viewer Update** for an update of your installation
   - **Vissim Viewer Setup** for a new installation
   - descending by version number

   You can download updates in the EXE or ZIP file formats. Setups are downloaded in the ZIP file format.
3. In the row of the desired Vissim viewer, click the **Download** button.

4. Save the downloaded file to your hard disk.
5. If you have downloaded a ZIP file, extract it.
6. Start the setup respectively update program.

   The Vissim Viewer is added to the Start menu.

The *Archive.zip* file downloaded also includes a *viewer_readme.txt*. This file describes how to create a CD for your clients that contains this limited Vissim version and your project data.

1.9 Using the PTV Vissim Simulation Engine

The Vissim Simulation Engine is a limited version of Vissim. It allows users to perform simulations and record the results. It does not have a graphical user interface. The Network editor and lists are not available. The Vissim Simulation Engine needs a dongle to use license base data, Vissim modules and signal controllers.

In a typical use case, you would model your network using a full version of Vissim and deploy multiple computers installed with the Vissim Simulation Engine to calculate simulations of the network that are configured differently. You can access the Vissim Simulation Engine via the COM interface.

1.10 Using files with examples

Example data are provided in the folders *Examples Demo* and *Examples Training*.

Depending on the Microsoft Windows operating system used, they are saved to different directories, for example:

..\Users\Public\Public Documents\PTV Vision\PTV Vissim <version number>

Tip: This directory contains the *Examples Overview.pdf* file with brief descriptions available in English.

1.10.1 Opening the *Examples Demo* folder

- From the **Help** menu, choose > **Examples** > **Open Demo Directory**.

1.10.2 Opening the *Examples Training* folder

- From the **Help** menu, choose > **Examples** > **Open Training Directory**.

1.11 Opening the Working directory

The **.inpx* network file of the network currently opened is saved to the working directory.

- From the **File** menu, choose > **Open Working Directory**.

1.11.1 Opening the working directory from the Windows Explorer

To show the Vissim Working directory in the Windows Explorer, in the Windows Explorer, enter the string of a variable.

For this purpose, the variable Vissim<Version>_EXAMPLES is automatically created during the installation process.

1. In the Windows Explorer, into the address bar, enter: `%VISSIM110_EXAMPLES%` or `%VISWALK110_EXAMPLES%`
2. Confirm with ENTER.

1.12 Documents

We provide a comprehensive manual to help you quickly become familiar with Vissim and Viswalk:

- The Help and manual describe functions and procedures. The step-by-step instructions guide you through tasks. The Help and manual are identical in content (see "Showing the PTV Vissim Help" on page 44), (see " Showing the user manual" on page 44).
- Additional documentation is available for further information (see "Additional documentation" on page 45)

1.12.1 Showing the user manual

If during the installation of Vissim the installation of the user manual is not deactivated, it is saved as a PDF file to the *..\Doc\<language ID>* folder of your Vissim installation directory.

1.12.2 Showing the PTV Vissim Help

If during the installation process of Vissim the installation of the Help was not deactivated, you can access the Help in Vissim.

Opening the PTV Vissim Help and showing the start page

- From the **Help** menu, choose > **PTV Vissim Help**.

The Help start page opens.

Showing the COM interface reference documentation

*Reference documentation for the COM interface **Vissim - COM** is by default available as a separate Help in the ..\exe directory of your Vissim installation.*

Showing Help of the attributes in currently opened window

- Press F1.

1.12.3 Additional documentation

In your Vissim installation directory, in the folder *..\Doc\Eng*, you can find the following documentation. These are partly also available in other languages.

- *Vissim <Version> - Installation Manual.pdf*: Installation guide
- *Vissim 10 - what's new.pdf*: Description of the most important changes in Vissim 9 compared to Vissim 10
- *ReleaseNotes_Vissim_<language ID>.pdf*: Release notes

  They contain information on changes, bug fixes, and new features available in the Vissim service packs. You can access the information by keyword search. New service packs are provided for download on a regular basis on the PTV GROUP website.
- *Vissim <Version> - Manual.pdf*: user manual, describing functions, network objects and attributes.
- *Vissim <Version> - COM Intro.pdf*: Introduction into the Vissim COM interface in English

  Using the Component Object Model (COM) interface and scripts, you can access data and functions in Vissim. Via the COM interface, you can find the *Vissim <Version> - COM.chm* file, by default, in the *..\PTV Vissim <Version>\Exe* folder.
- *Overview_CodeMeter.pdf*: Information on how to use the CodeMeter Runtime Kit and set up CodeMeter as a dongle server
- *CodeMeter_Support.txt*: Information on support for WIBU CodeMeter dongle
- *CodeMeterAdministratorManual6.20_en.pdf*: CodeMeter Administrator manual version 6.20 - April 2016
- *VAP_<version>_<language ID>.pdf*: User manual for VAP add-on module, which allows you to simulate a freely programmable, stage or signal group oriented, traffic-actuated signal control
- *vap_chan_<version>_<language ID>.txt*: upgrades and bug fixes in VAP/SIM
- *VisVAP <version> <language ID>.pdf*: User manual VisVAP (tool that allows you to display the program logic as a flow chart)
- *LISA+_OMTC.pdf*: only in directory *..\Doc\Deu*: Information on VIAP LISA+ / Vissim interface
- *LicenseAgreementGeneral.rtf*: Vissim license agreement
- *attribute.xlsx*:
  - List of **tables** for base data types and network object types
  - List of **attributes**, including ID, short name, long name, singular, plural in the languages Vissim is available in.
  - List of **EnumStrings**: enumeration types with predefined values

- List of **relations**: base data types and network object types that have relations to other base data types and network object types, as well as the base data types and/or network object types assigned.

The directory *..\Doc\Eng* also includes the following documentation:

- *Manual_RBC.pdf*: Ring Barrier Controller Manual
- *Manual_Synchro_Import.pdf*: Synchro 7 Import Manual
- *release_notes_RBC.txt*

1.13 Service and support

With Vissim you receive extensive technical documentation and can call on the services of PTV GROUP. PTV GROUP provides technical support in the following cases:

- Program errors in the current program version
- Questions about the use of Vissim

Notes: The prerequisite for technical support is participation in a basic course for Vision Traffic Suite.

As our software is continually being improved, we regret being limited when it comes to providing support for older program versions.

Please understand that the PTV Vision Support cannot replace a training course. PTV Vision Support can neither impart specialist engineering knowledge which goes beyond the functionality of the product, for example concerning demand modeling, signal control or project-related problems. Should you require any help on these subjects, we will be happy to offer you a project-specific training course.

1.13.1 Using the manual, Help and FAQ list

Before you make an inquiry to the technical support, please read the information about the topic in the manual, the Help or the FAQs.

1.13.1.1 FAQs

Here you can find the answers to frequently asked questions about Vissim on our web pages:

PTV Vissim FAQs

Note: For access to the FAQ list you need access to the Internet.

- From the **Help** menu, choose > **FAQ (Online)**.

In your browser, the Vissim FAQ list appears.

1.13.2 Services by the PTV GROUP

1.13.2.1 Product information on the Internet

On the PTV GROUP web page you can find further product information, AVI files with examples of various simulations as well as service packs, which you can download:

Homepage of PTV Group: PTV Vissim

1.13.2.2 Product training sessions

PTV GROUP offers training sessions for entry-level and experienced users.

We will be glad to carry out training sessions in your own company. You can also participate in training sessions which we hold on our own premises.

You can find the latest information on our web page:

PTV Vissim Training Courses

1.13.2.3 Maintenance Agreement

A Maintenance Agreement ensures that the current version of Vissim or Viswalk is always available.

Advantages of a Maintenance Agreement:

- Provision of the latest version of Vissim or Viswalk, as soon as it is available
- Service packs for the current version for download from our web pages
- Support by PTV Vision Support

Please address any inquiries about Maintenance Agreements to:

customerservice@vision.ptvgroup.com.

1.13.2.4 Downloading service packs

If you have a Maintenance Agreement you can download the service packs for the current version of Vissim or Viswalk from the Internet.

1. Open the web page PTV Vissim & PTV Viswalk Service Pack Download Area.
2. Click on the desired version.

   Depending on the product you are using, you can download files in the EXE or ZIP file format.
3. In the **Available Downloads** table, click the row with the desired product. Then click the **Download <file format>** button.
4. Download the file.

1.13.2.5 Being automatically informed about new service packs

You can be notified if new service packs are available for downloading.

1. Open the web page PTV Vissim & PTV Viswalk Service Pack Download Area.
2. Click on **sign-up/sign-off**.
3. Fill in the form.
4. Click the **Submit** button.

1.13.2.6 PTV Vision Support

PTV GROUP offers technical support for Viswalk and Vissim (see "Posting a support request" on page 48).

1.13.2.7 PTV Vissim Webinars

Learn about the concepts of simulation in PTV Vissim step by step in our free webinars. For current information on our webinars, check our Webinars site.

1.13.3 Posting a support request

You can contact PTV Vision Support with a contact form in the following cases:

- Program errors in the current Vissim version. As our software is continually being improved, we regret being limited when it comes to providing support for older program versions.
- If you have a Maintenance Agreement, you can contact us with inquiries about your project and for modeling.

Before you contact us with questions and problems:

- From the **Help** menu, choose Vissim Help or use the manual to try to solve the problem. These often contain the information which you are looking for.
- Read the tips and tricks on the Internet: PTV Vissim Tips & Tricks.
- Read the FAQ list on the Internet. These often contain the information which you are looking for. The FAQ list also contains valuable information about modeling.

In the interest of an efficient processing of your inquiry to PTV Vision Support we request that you use the corresponding hotline contact form on the Internet:

- Technical Support PTV Vissim
- Technical Support PTV Viswalk

▸ From the **Help** menu, choose > **Technical Support**.

The browser opens and displays the Vissim contact form.

Note: The following information is necessary for the smooth processing of your inquiry:

- A description of the problem
- The steps which were performed immediately before the problem occurred
- If necessary, screenshots of the program states
- All files which are necessary for the reproduction of the error

When you open the form in Vissim via the **Help** menu, the following data is automatically copied into the form:

- Vissim version and service pack number, e.g. 10.00-07, as listed in the title line of Vissim
- Vissim edition, 32 Bit or 64 Bit
- The operating system and service pack number
- The PTV customer number
- The dongle number

Thank you for your cooperation!

1.13.4 Requests to the Traffic customer service

Our Traffic customer service will respond to general requests concerning Vissim.

For inquiries about license fees, please contact:

- info@vision.ptvgroup.com

Please address inquiries about Maintenance Agreements to PTV Traffic Customerservice:

- customerservice@vision.ptvgroup.com

1.13.5 Showing program and license information

You can show information on the dongle, network size, installed add-ons, version number, build number and installation directory of your Vissim installation as well as manage licenses.

▸ From the **Help** menu, choose > **License**.

*The **License** window opens.*

The following license information is displayed:

- **Maximum** section
  - **Signal Controllers (SC)**: Maximum number of SC
  - **Network Size**: Maximum network extent in km x km
  - **Link behavior types**: Maximum number of link behavior types
  - **Period**: Maximum simulation duration in seconds
  - **Pedestrians**: Maximum number of pedestrians
- **Modules** section
  - List of modules and add-on modules Check marks show the licensed modules and add-on modules installed.
- **Signal Controllers** section

- List of signal control procedures supported
- **Version** section
  - Customer-specific data of the installed version

Program path: Path of installation directory

Manage licenses: Opens the **License Management** window (see "Managing licenses" on page 50).

Note: If your Vissim license does not include the Viswalk add-on module, you may still perform a pedestrian simulation with up to 30 pedestrians.

1.13.6 Managing licenses

Note: The scope of functions of the different Vissim versions such as Uni , Demo or Vissim Viewer may differ from the scope of functions of the standard version.

You can identify the licenses available in your network and specify which licenses you want to use when you start the program. In the same way, you can manage the licenses for individual modules.

1. From the **Help** menu, choose > **License**.

   *The **License** window opens.*

2. Click the **Manage licenses** button.

   *The **License Management - PTV Vissim** window opens. PTV Vissim automatically searches for licenses.*

Tip: You can also open the **License Management - PTV Vissim** window from the start menu for programs under PTV Vision **<Year>** > PTV Vissim License Manager.

3. Make the desired changes:

Element	Description

Settings	You can show or hide the section.
	- **Check out automatically, if possible**:
	  - Select this option to have the path to the licenses in the registry automatically loaded and started when you start PTV Vissim. The **License Management - PTV Vissim** window does not open. The option is selected by default.
	  - If you do not select this option, the **License Management - PTV Vissim** window will always open when you start PTV Vissim and you will need to select a license. PTV Vissim starts automatically, independently of the option, if exactly one PTV Vissim license exists on all searched CodeMeter dongles.
	- **Delete saved license list**: Deletes all license information saved to the following directory (example): C:\User\<User name>\AppData\Roaming\PTV Vision\PTV Vissim 10

	If you open the **License Management - PTV Vissim** window, initially, the found licenses are displayed in gray because the CodeMeter servers have not yet been searched at that time. You can get the current status of all available licenses if you click **Update all displayed licenses** below the list of found licenses. The next time you open PTV Vissim, you can specify new settings. If you want to save a new *licenses.data* file, close the **License Management - PTV Vissim** window with **OK**. You can find information on central, user-independent license management below this table.
CodeMeter server envir-onment	Used license servers with status on which a CodeMeter server is installed. The list is based on the **CodeMeter server search list**.
	**Update**: Search for computers in your network on which a CodeMeter server has been installed. The list is updated.
Licenses	Vissim licenses and/or Viswalk licenses that in your network have been saved to CodeMeter servers. - **Use license**: Select the license with which you want to start PTV Vis-sim. You may select multiple licenses. The sequence in the list determ-ines the sequence in which the licenses are reserved. If a license is being used by another user, it cannot be selected. - **Product**: Shows products for which the license is valid. Use **+** and **-** to show and hide the list of modules. You can book out individual mod-ules. - **License name**: License text - **Number**: For a network license, the number of licenses is shown. For single-user licenses, the text **N/A** is displayed.

Element	Description
	- **Expiration date**: If applicable, the expiration date of the license is shown, otherwise may be used for an unlimited period. - **Network size**: Network size of the license is displayed - **Languages**: Languages supported by the licensed version - **Server**: License server - **Box**: Shows serial number of the CodeMeter stick
	Move the row of selected licenses one row up
	Move the row of selected licenses one row down
	Moves the row of selected licenses to the top of the list
	Moves the row of selected licenses to the bottom of the list
Remove from list	Delete selected licenses from the list
Find licenses	Start a network search for CodeMeter servers.
Update all displayed licenses	Search for licenses on computers that are shown in the **CodeMeter server environment** list. The **Found licenses** list is updated.
Activate new or changed license	The hyperlink opens the PTV website on which you can enter the ticket code for the activation process. For further information on how to proceed, refer to the installation guide in your Vissim installation folder under *..\Doc\Eng\Vissim <Version> - Installation Manual.pdf*, in the chapter "License provisioning". Note: New or changed license information is transferred to the dongle during the activation process. Each time a new license is issued or licenses are changed, a contact person in your company receives an e-mail with a ticket code similar to A43UT-PMXRC-43D76-KF3AH-Y5GDQ (example). Using this ticket, you can start the activation process, access new license information from the PTV license server and transfer it to the dongle.
Start	Open Vissim with the license selected

Note: If your Vissim license does not include the Viswalk add-on module, you may still perform a pedestrian simulation with up to 30 pedestrians.

Centralized license management

In certain working environments, an administrator has to preset the license selection throughout the system. In this case, a *licenses.data* file is stored in folder *..\ProgramData\PTV Vision\PTV Vissim <version number>*. The settings in this centrally stored file also apply if your user-specific directory *C:\Users\<user name>\AppData\Roaming\PTV Vision\PTV Vissim <version-number>* contains a *licenses.data* file with other settings.

The icon on the button to delete the list of licenses indicates the centralized license management.

You can only edit or delete this central file if you have administrator rights. If you delete this file, license settings will subsequently be saved separately for each user of the computer. The icon will no longer be displayed on the button if your license management is user-specific.

Please also refer to section **System-wide license selection** in the installation guide of Vissim.

1.13.7 Information about the PTV GROUP and contact data

You can find information about the PTV GROUP and contact data in Vissim and Viswalk.

- Select the menu **Help** > **About PTV Vissim**.
  - Version number
  - The Internet page of PTV GROUP
  - Copyright details

2 Principles of operation of the program

You model your network in Vissim in a network editor. The network editor shows the precise position of the network objects. By default, you edit the data for the network objects in lists. For editing, you can use, for example, mouse functions, context menus and key combinations.

Note: A good knowledge of Windows is assumed for the operation of the program.

Tip: For your first steps in Vissim you can use simple example data, which were installed with Vissim. By default, the example data are stored under:

Users\Public\Public documents\PTV Vision\PTV Vissim <Version>\Examples Demo

You can gain a first insight and practical experience with Vissim by following the tutorial "First Steps": *C:\Users\Public\PUblic Documents\PTV Vision\PTV Vissim 11\Tutorial First Steps*

2.1 Program start and start screen

The simplest way to start Vissim is by double-clicking on the Vissim icon on your desktop:

Tips: Other options to start Vissim:

- via the Start menu
- in Microsoft Windows 8, via the Start screen
- In Microsoft Windows 7 or 8 add Vissim to the taskbar
- via the command prompt (see "Starting PTV Vissim via the command prompt" on page 56)
- double-click a network file (**.inpx*) in the Windows Explorer
- as Microsoft Windows server from session 0. Vissim can then be executed on a Microsoft Windows HPC server without a user session.

If after the initial start more than one license is found, after you start the program the window ***License management - PTV Vissim*** *opens.*

1. Select the license of your choice (see "Managing licenses" on page 50).

Vissim opens. The Start sceen shows information on the program version:

- Number of Vissim version
- Installation folder

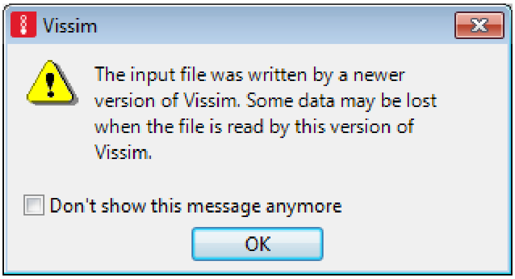
Tip: You can call up further program information in Vissim:

- From the **Help** menu, choose > **About**.
- From the **Help** menu, choose > **License**.

If Vissim writes important messages into the **Messages** window when importing a network file, a corresponding message is displayed. You can open the **Messages** window immediately or later on.

- Open immediately: In the window informing you of the messages, click the **Open** button.
- Open later: In the window informing you of the messages, click the **Open** button. At a later time of your choice, from the **View** menu, choose > **Messages**.

When you open a layout or network file that contains attributes of a more recent Vissim version than the one you are using, you will not be able to read these attributes. The following message is displayed:



2. Confirm with **OK**.

   A window opens. Attributes that cannot be read are listed.

3. Confirm with **OK**.

   If the defaults.inpx file in the ..\AppData\Roaming\PTV Vision\PTV Vissim <Version number> directory is not readable, a corresponding message is displayed in the ***Messages*** *window.*

   If the ..\AppData\Roaming\PTV Vision\PTV Vissim <Version number> directory does not contain a defaults.inpx file, the defaults.inpx file in the Exe installation directory of Vissim is used.

   When you save the Vissim network as default network (see "Overview of menus" on page 121), the *defaults.inpx* file is saved to the *..\AppData\Roaming\PTV Vision\PTV Vissim <Version number>* directory.

Vissim opens and shows the start page in a tab displayed in the foreground (see "Using the Start page" on page 57).

4. If you want to go from the start page to the network editor, click the **Network Editor**tab below the start page.

Up to four instances of Vissim can be opened at the same time.

Note: Your user preferences are saved in the Windows registry and in the **.layx* layout file when Vissim is ended. The settings are used automatically when the program is restarted.

2.2 Starting PTV Vissim via the command prompt

You can also start Vissim via the command prompt. Here, you can use parameters to control various start options.

1. Press the Windows key and the *r* key at the same time.

   *The **Run** window opens.*

2. Click the **Browse** button.

   *The **Browse** window opens.*

3. Browse to the Exe directory of your Vissim installation. This is by default:

   - 64-bit: *C:\Program Files\PTV Vision\PTV Vissim 11\Exe*

4. Double-click the *VISSIM 11.exe* file.

   *The **Browse** window closes. The path is entered in the **Start** window, in the **Open** box.*

5. Next to the *..\VISSIM 11.exe* file, into the **Open** box, enter a space.

6. After the space, enter the desired command line parameter:

Parameter	**Description**
-automation	Provides Vissim as a COM server in the automation mode for COM scripts that are started subsequently.
<input file>	Loads the specified network file **.inpx* or **.inp*. If the network file has not been saved to the Exe directory, enter the path in front of the <input file>. If a path contains spaces, add the character " at the beginning and end of the path, for example *"C:\Program Files\PTV Vision\PTV Vissim 9\Exe\Vissim90.exe" Busmall.inpx*
-b <layout file>	Loads the specified layout file **.layx*.
-flush	Immediately saves any debug messages to the file *%Tmp%\VISSIM\vissim_msgs.txt*, instead of waiting until a certain number of messages has been reached. Use this parameter to ensure that the *vissim_msgs.txt* file contains all messages, e.g. when looking for a program error and you want to send all messages in a Hotline package to support. In this case, use the `-flush` parameter together with the `-o` parameter: `...-o -flush` `-flush` has an impact on the performance of Vissim. Therefore do not use this parameter by default. Only use it when you want to make sure that all messages are included in the *vissim_msgs.txt* file.
-force3d	Deactivates the testing of the supported OpenGL version.
-o	Saves debug messages in the *%tmp%\VISSIM\vissim_msgs.txt* file.

Parameter	Description
-q <input file>	Enables the Quick mode during simulation. If the network file has not been saved to the Exe directory,next to the `<Input file>`, enter the path. `-q` must be followed by a space.
-regserver	Registers Vissim as a COM server. If the registration has not been successful, a message opens.
-s <input file>	Batch operation: Vissim starts the simulation and closes after the end of the simulation. If the network file has not been saved to the Exe directory,next to the `<Input file>`, enter the path. `-s` must be followed by a space.
-unregserver	Deregisters Vissim as a COM server.
-version	Opens the Start screen and shows information on the program version

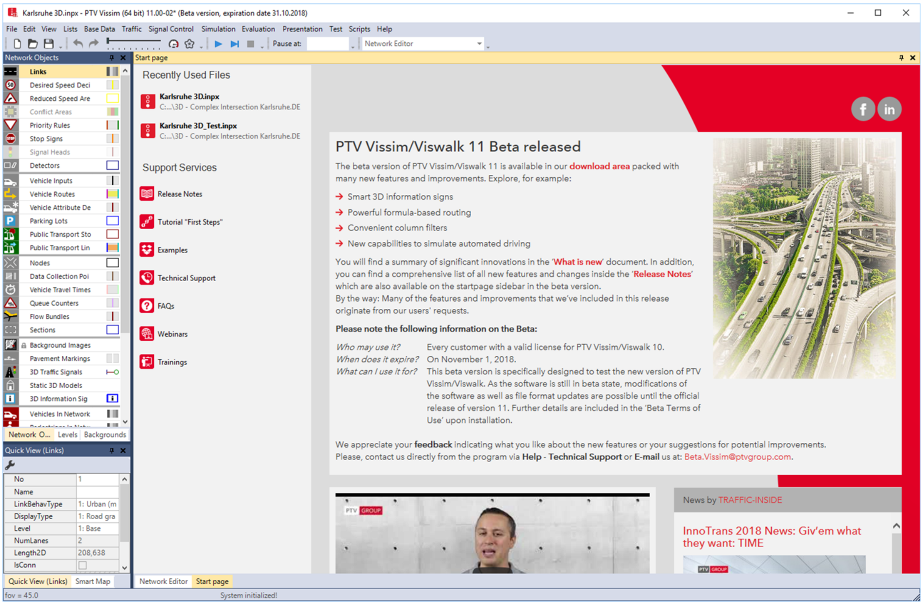
7. Confirm with **OK**.

2.3 Using the Start page

After you start the program, the start screen opens and the Vissim user interface with the **Start page** tab is displayed.

The Start page allows you to have quick access to the following elements:

- **Recently used files**
- **Support Services**: Opens Help file, documents or web pages that provide support for users
- In the section on the right: Latest product news, interesting PTV news, information on new service packs. This section may contain information that varies by region. Some of it is available in different languages. Select the desired region (see "Selecting the country for regional information on the start page" on page 150).



An Internet connection is required to make use of the full functionality of the Start page.

When you open a ***.inpx** network file, the tab with the Start page is closed. The program elements are then displayed based on the settings saved to the ***.layx** file.

When you choose **File** > **New** or click the **New** button , the tab and Start page are moved to the background and the Network editor is displayed in foreground.

Opening the Start page tab

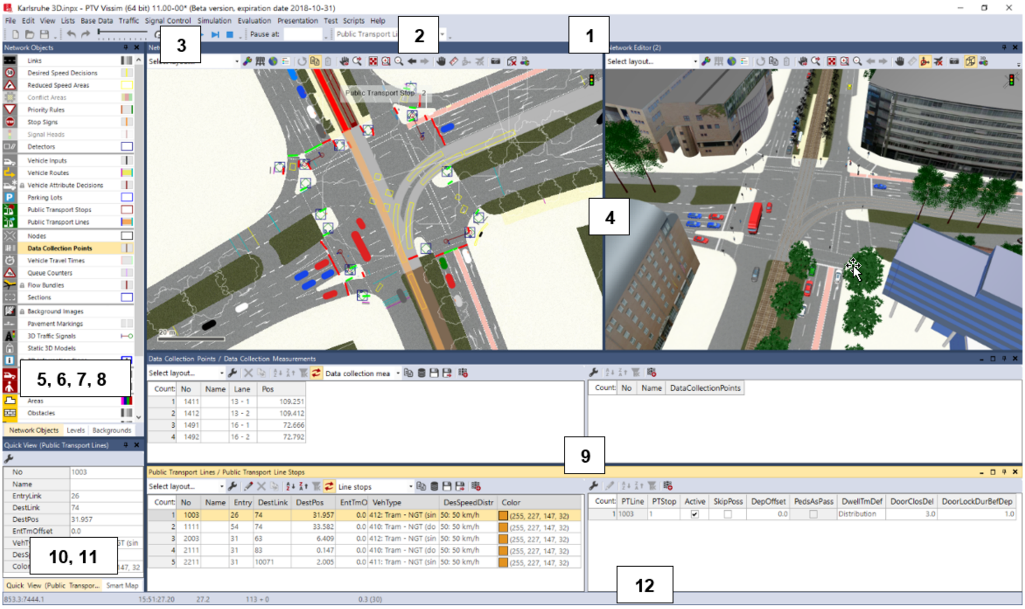
- From the **View** menu, choose > **Start page**.

2.4 Becoming familiar with the user interface

After you start the program, the start screen opens and the Vissim user interface with the **Start page** tab is displayed (see "Using the Start page" on page 57).

Program elements of the user interface

The following figure shows the program interface with a **.inpx* network file opened and individually arranged program elements:



By default, the user interface contains the following elements for viewing, editing, and controlling the network, data and simulation:

Element	Description
(1) Title bar	- If a network file **.inpx* is open: file name of the network file - Program name - Version number including service pack number - Opened version: **Demo**, **Uni**, **Viewer**, **Academic license** When an **Academic license** is opened, diagnostic and usage data is collected. You require an Internet connection. In the **License** window, in the **Version** section, under Product variant **Academic License** is displayed.
(2) Menu bar	You can call program functions via the menus (see "Overview of menus" on page 121). Network files used most recently in Vissim are shown in the **File** menu. Click on the entry if you want to open one of these network files.
(3) Toolbars	You can call program functions via the toolbars. Lists and network editors have their own toolbars (see "Using toolbars" on page 135).
(4) Network Editors	Show the currently open network in one or more Network Editors. You can edit the network graphically and customize the view in each Network Editor (see "Using network editors" on page 75).

Element	Description
(5) Network objects toolbar	By default, the network objects toolbar, level toolbar and background image toolbar are shown on individual tabs in a window. **Network objects toolbar** (see "Using the Network object toolbar" on page 61): - Select the Insert mode for network object types - Select visibility for network objects - Select selectability for network objects - Editing graphic parameters for network objects - Show and hide label for network objects - Context menu for additional functions
(6) Levels toolbar	- Select visibility for levels (see "Using the Level toolbar" on page 65) - Select editing option for levels - Select visibility for vehicles and pedestrians per level
(7) Background toolbar	- Select visibility for backgrounds (see "Using the background image toolbar" on page 66)
(8) Project explorer	Displays projects, base networks, scenarios and modifications of scenario management
(9) Lists	In lists, you show and edit different data, for example, attributes of network objects. You can open multiple lists and arrange them on the screen (see "Using lists" on page 93).
(10) Quick View	Shows attribute values of the currently marked network object. You can change attribute values of the marked network objects in the **Quick View** (see "Using the Quick View" on page 68).
(11) Smart Map	Shows a small scale overview of the network. The section displayed in the Network Editor is shown in the **Smart Map** by a rectangle or a cross-hair. You can quickly access a specific network section via the **Smart Map** (see "Using the Smart Map" on page 71).
(12) Status bar	Shows the position of the cursor in the Network Editor. Shows the current simulation second during a running simulation.

Arranging program elements

You can arrange the program elements of the user interface according to your requirements, such as by moving, displaying from other program elements or hiding from available program elements (see "Changing the display of windows" on page 89).

You can also arrange program elements on multiple screens. Thus you can edit the network and data in a structured way, such as by opening a network and adding more network objects, building a new network from network objects, or running a simulation.

Saving the user interface layout

The layout of the user interface is saved by default when the network is saved. The program elements are arranged accordingly the next time you open Vissim (see "Saving and importing a layout of the user interface" on page 146).

Without a Vissim network file loaded, the Network Editor is empty.

If no network file is open, the Network Editors show an empty Vissim network (see "Using network editors" on page 75).

Labeling mandatory fields and invalid data formats

Vissim highlights entry fields with a red in the following cases:

- Entry field is mandatory. You must enter a value or string in a valid data format.
- Part of the value or string entered is in an invalid data format. For example, for the attribute **No** of a link, a number must be entered. Characters are not accepted.

You must fill in all mandatory fields of a window to be able to close it with **OK** and make the data available in Vissim.

Point the mouse pointer at the red triangle to open a quick info about the cause of error.

Quick info displays a description

For many program elements, a small window opens temporarily, if you point the mouse pointer to the program element, e.g. a toolbar button or a network object type in the network object toolbar. The window displays a brief description of the program element.

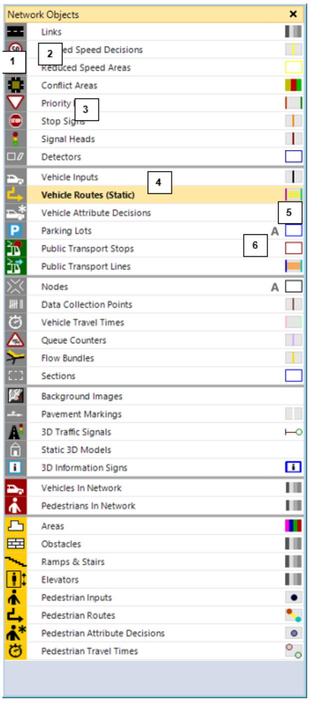
If in an attribute list, you point the mouse pointer to an attribute name in a column header, a small window opens displaying a description of the attribute (see "Structure of lists" on page 94).

In some windows, the symbol is displayed next to an entry box, list box or option. If you click the symbol, a small window opens displaying information on the program element.

2.5 Using the Network object toolbar

The network object toolbar contains a list with the network object types. In a network editor, you can use icons and the context menu to access functions for the display, selection, and editing of network objects. Your settings are only active in the last network editor used. If you click another network editor to activate it, in the network objects toolbar, the settings for this network editor are displayed.

Tip: By default, the network object bar, the level bar, 3D information sign bar and background bar are aligned together. Using the tabs at the bottom of a toolbar, you can switch to another toolbar.



Elements in the network objects toolbar

(1) Network object type button (visibility)

- The icon (and name) of the network object type are shown in color: Network objects of this network object type are shown in the Network editor.
- The icon (and name) of the network object type are grayed out: Network objects of this network object type are not shown in the Network editor.

(2) Lock button (selectability)

The icon is shown only when you position the mouse pointer before the name or on the name of the network object type or when the lock is locked.

	Lock unlocked: You can select and edit network objects of this network object type in the Network editor.
	Lock locked: You cannot select or edit network objects of this network object type in the Network editor.

(3) Button with the name of the network object type (Insert mode)

Click the button with the name of the network object type to activate the Insert mode. If the Insert mode is activated, the entire row is highlighted in orange.

- You can insert new network objects of this network object type into network editors.
- The network objects of this network object type are visible and selectable in all network editors.
- You can switch off visibility and selectability only for the other network object types.

(4) Selecting variants of a network object type

If you click the button of a network object type that has variants, the variant that is currently selected is shown in brackets together with the icon . If you click the button or icon again, you can select a different variant to add a network objects of this network object type to a network editor.

- Vehicle Routes:
  - Static
  - Partial
  - Partial PT
  - Parking Lot
  - Dynamic
  - Closure
  - Managed Lanes
- Areas:
  - Polygon
  - Rectangle
- Obstacles:
  - Polygon
  - Rectangle
- Sections:
  - Polygon
  - Rectangle

(5) Edit graphic parameters icon: Open preview and graphic parameters

This icon shows you how network objects of this type are displayed in the Network editor. The icons of network object types, displayed differently depending on the attribute values, consist of several colored bars. Four gray bars indicate that the Display type is activated (see "Defining display types" on page 320).

Click the symbol to open a list of graphic parameters of the network object type. In the list, you can select and edit graphic parameters (see "Editing graphic parameters for network objects" on page 158), (see "List of graphic parameters for network objects" on page 161).

(6) Label icon

This icon is shown only when you position the mouse pointer on the name or next to the name of the network object type.

Click this icon to show or hide the label of the network objects of this network object type:

	Label is hidden
	Label is shown

Context menu in the network object toolbar

In the context menu, you can select functions for visibility and selectability and call the list of network objects of the selected network object type (see "Context menu in the network object toolbar" on page 64).

Changing the display of the network object toolbar

You can customize the position, size, and visibility of the window (see "Changing the display of windows" on page 89).

2.5.1 Context menu in the network object toolbar

1. In the network object bar, right-click the network object type for which you want to select a command from the short menu.
2. Choose the desired entry from the context menu.

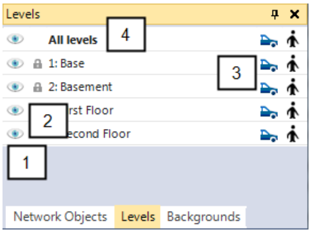
Function	Description
Show List	If you have already opened a list of the network object type, it will be displayed in the foreground. If no list of the network object type has been opened, the list will be opened and displayed in the foreground.
Open new list	Show the list with network objects of network object type, including the network objects' attributes If you have already opened a list of the network object type, another list of the network object type will be opened.
Create Chart	Open the **Create Chart** window (see "Creating charts" on page 1113). In the **Network object type** list box, the object type you right-clicked is automatically selected.

Function	Description
Creating a user-defined attribute	Open the **User-defined attribute** window (see "Using user-defined attributes" on page 210). In the **Object type** list box, the object type you right-clicked is automatically selected.
Edit graphic parameters	Show list of graphic parameters of the network object type (see "Editing graphic parameters for network objects" on page 158)
Make All Types Visible	Display all network objects of all network object types in the Network editor
Make No Types Visible	Hide all network objects of all network object types in the Network editor
Make All Types Selectable	All network objects of all network object types can be selected in the Network editor
Make No Types Selectable	Not all network objects of all network object types can be selected in the Network editor
Selectability Column	Show or hide the **Selectability** column on the network objects toolbar
Label Column	Show or hide the **Label** column on the network objects toolbar
Graphic Parameters Column	Show or hide the **Graphic parameters** column on the network objects toolbar
All Object Types	On the network objects toolbar, show all network objects types for vehicle and pedestrian simulation
Vehicle Object Types Only	On the network objects toolbar, only show network object types for vehicle simulation and hide network object types for pedestrian simulation
Pedestrian Object Types Only	On the network objects toolbar, only show network object types for pedestrian simulation and hide network object types for vehicle simulation

2.6 Using the Level toolbar

The Level toolbar shows the defined levels.

Tip: By default, the network object bar, the level bar, 3D information sign bar and background bar are aligned together. Using the tabs at the bottom of a toolbar, you can switch to another toolbar.



(1) Visibility button

- Button is colored and activated: Static objects on this level are displayed in the Network editor. In addition, on the network objects toolbar, the visibility of the network object types must be active.
- Button is gray and deactivated: Static objects on this level are not displayed in the Network editor.

(2) Selectability button

- activated: You can edit static objects on the this level.
- deactivated: You cannot edit static objects on the this level.

(3) Buttons for vehicles and pedestrians

Display or hide vehicles or pedestrians in the active Network Editor at this level.

(4) All levels

Activate or deactivate visibility and selectability of static objects for all levels.

You can customize the position, size, and visibility of the level toolbar (see "Changing the display of windows" on page 89).

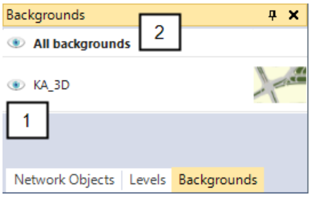
Shortcut menu of the Level toolbar

From the shortcut menu, you can choose to open the **Levels** list (see "Defining levels" on page 922).

2.7 Using the background image toolbar

The background image toolbar shows the names of the loaded backgrounds in succession.

Tip: By default, the network object bar, the level bar, 3D information sign bar and background bar are aligned together. Using the tabs at the bottom of a toolbar, you can switch to another toolbar.



(1) Visibility button

- Button is colored and open: background is displayed in the Network Editor. The visibility of backgrounds must be activated in the network object toolbar.
- Button is gray and closed: background is not displayed in the Network Editor.

(2) All backgrounds

Select or deselect visibility for all backgrounds.

You can customize the position, size, and visibility of the background image toolbar (see "Changing the display of windows" on page 89).

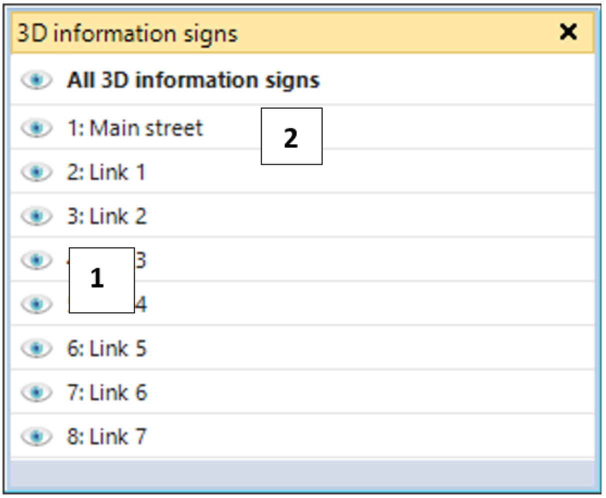
Shortcut menu of Background images bar

From the shortcut menu, you can choose to open the **Backgrounds** list (see "Attributes of background images" on page 404).

2.8 Using the 3D info sign bar

The 3D information sign bar allows you to display the 3D info signs defined.

Tip: By default, the network object bar, the level bar, 3D information sign bar and background bar are aligned together. Using the tabs at the bottom of a toolbar, you can switch to another toolbar.



(1) Visibility button

- Button is colored and activated: 3D info signs are displayed in 3D mode in the network editor. In addition, on the network objects toolbar, the visibility of the 3D info signs must be active.
- Button is gray and deactivated: 3D info signs are not displayed in the network editor.

(2) All 3D information signs

Select or deactivate the visibility of the 3D information signs.

You can adjust the position, size, and visibility of the 3D info sign bar (see "Changing the display of windows" on page 89).

Shortcut menu of the Level toolbar

From the shortcut menu, you can choose **3D Information Signs** (see "Defining levels" on page 922).

2.9 Using the Quick View

The **Quick View** contains a selection of attributes and attribute values for the network objects currently selected. When you globally select network objects, your selection is the same in all network editors and synchronized lists.

- If several network objects of the same network object type are selected, the name of the network object type is shown in the title bar of the Quick View.
- If several network objects with different attribute values are selected, the symbol * is shown with the different attribute values.

- If several network objects of different network object types are selected, the following message will be displayed in Quick View: **Multi-type selection, no quick view available**.
- If several network objects of different network object types are selected, no attribute values are shown and no name is shown in the title bar of the Quick View.
- If synchronization is activated in a list with network objects and you select network objects in the list, these are shown in all network editors and in **Quick View** (see "List toolbar" on page 97).

Data cells that you can or cannot edit are highlighted in color or hatched (see "Structure of lists" on page 94). Hatched cells: Irrelevant attribute values that due to other attribute values have no effect.

- You can select the attributes that are displayed (see "Selecting attributes for the Quick view display" on page 69).
- Using global selection, you can change the attribute values of all network objects (see "Editing attribute values in the Quick view" on page 70).
- You can customize the position, size, and visibility of the **Quick View** (see "Using toolbars" on page 135).

Tip: The **Smart Map** and **Quick View** are displayed in the same section by default. To switch between the **Smart Map** and **Quick View**, use the tabs.

2.9.1 Showing the Quick View

- In the **View** menu, choose > **Quick View**.

The Quick View opens as a tab. The position depends on how your user interface is set up.

2.9.2 Selecting attributes for the Quick view display

You can select network objects in a Network Editor or list and specify which attributes are shown in the quick view. When you press the CTRL key to select multiple network objects, these may belong to diifferent network object types.

1. If you want to select network objects in the Network editor, you must ensure that visibility and selectability have been activated for the network object type in the network object toolbar.
2. In a network editor or list, select at least one network object.
3. Select the **Quick view**.
4. Click on the **Attribute selection** icon.

   *The **Select Attributes** window opens (see "Selecting attributes and subattributes for columns of a list" on page 112).*
5. Select the desired attributes.
6. Deactivate the attributes that you do not want to display in the **Quick view**.
7. Confirm with **OK**.

The attributes selected and their values are displayed in the Quick view window, in the units and sequence specified.

Note: The selection of attributes in the **Quick view** is saved separately for all objects in a **.layx* file (see "Saving and importing a layout of the user interface" on page 146).

2.9.3 Editing attribute values in the Quick view

1. Make sure that the network objects are selected in the network whose attribute values you want to edit.
2. In the Quick view, mark one or more cells.

Note: If several network objects with different attribute values are selected, the * symbol is shown instead of the value of the attribute.

3. Into the field selected, enter the value of your choice.
4. Confirm with ENTER.

Notes: When entering numerical attribute values with a reference to length, time, speed and acceleration, you must take the units into account:

- If you enter just one digit, the value is interpreted in the format in which the attribute is displayed. Example: If an attribute is displayed in the minutes format, the value 5 is read as 5 minutes. You can change the format in the attribute selection window (see "Selecting attributes and subattributes for columns of a list" on page 112).
- You can enter a number with a unit. In this case, Vissim converts the value in such a way that it is displayed accordingly in the selected format. You can enter length units either with metrical (m, km) or imperial (ft, mi) units. For example, if an attribute is shown in the format **Minutes**, you can enter 2 h. After the confirmation, the value 120 will be displayed.

*The new attribute value is shown in all highlighted fields and applied to all objects that were selected. This also applies to fields in which the symbol * referenced different attribute values.*

2.9.4 Editing attribute values in the Quick view with arithmetic operations

Make sure that the network objects are selected in the network whose attribute values you want to edit.

1. In the Quick view, mark one or more numerical cells.
2. In a marked cell, enter one of the below arithmetic operations:

Operation	Syntax
Addition	=+2
Subtraction	=-2
Multiplication	=*2

Operation	Syntax
Division	=/2
Raise to power	=^2
Setting the lower bound value	=min2
Setting the upper bound value	=max2
Use an exponential function	=exp
Take logarithm	=log
Form reciprocal value	=1/

The operation is run for all marked cells with a numerical value. The result is entered as attribute value.

Warning: If you have selected an alphanumerical cell, the text of the arithmetic operation is entered in the cell!

2.10 Using the Smart Map

The **Smart Map** displays a small scale overview of the network. A rectangle shows the section which is currently displayed in the Network Editor. If the zoom factor in the Network editor is so large that the section in the **Smart Map** is not displayed as a rectangle, cross hairs are displayed instead.

If a network is displayed in several Network Editors, the **Smart Map** uses rectangles to indicate the different sections. The border line of the rectangle that displays the section of the active Network Editor is formatted in bold.

Smart Map is linked to the Network editor. Modifications which are made in the Network editor view have an effect on the **Smart Map** and vice-versa. For instance, if you zoom into the network or move the current view, the position of the rectangle or cross hairs is moved in the **Smart Map** as well. To change the network view, change the position or size of the colored rectangle in the **Smart Map**.

Dynamic objects such as vehicles and pedestrians are not shown in the Smart Map. Network objects selected in Network Editors are not highlighted in the Smart Map.

You can customize the position, size, and visibility of the **Smart Map** (see "Using toolbars" on page 135).

Tip: The **Smart Map** and **Quick View** are displayed in the same section by default. To switch between the **Smart Map** and **Quick View**, use the tabs.

2.10.1 Displaying the Smart Map

- In the **View** menu, choose > **Smart Map**.

The Smart Map opens as a tab. The position depends on how your user interface is set up.

2.10.2 Displaying the entire network in the Smart Map

Note: If a network is displayed in several Network editors, the **Smart Map** uses rectangles to indicate the different views.

1. Right-click on the **Smart Map**.
2. Select **Display Entire Network** from the context menu.

   *The entire network is shown in the **Smart Map**.*

2.10.3 Moving the Network Editor view

To move the view shown in the Network Editor, in the **Smart Map**, move the rectangle. If the network is greatly enlarged in the **Smart Map**, the rectangle may lie outside of the **Smart Map** and is not shown in the **Smart Map** (see "Displaying the entire network in the Smart Map" on page 72).

Note: If a network is displayed in several Network editors, the **Smart Map** uses rectangles to indicate the different views.

Note: Make sure that you choose the desired rectangle in the next steps.

1. In the **Smart Map**, point the mouse pointer to a position inside the colored rectangle.

   The mouse pointer becomes a  *symbol.*
2. Hold down the mouse button.
3. Drag the rectangle to the position of your choice and release the mouse button.

*The Network Editor view is adjusted according to the **Smart Map**.*

2.10.4 Showing all Smart Map sections

Note: If a network is displayed in several Network editors, the **Smart Map** uses rectangles to indicate the different views.

1. Right-click on the **Smart Map**.
2. From the shortcut menu, choose **Zoom All Sections**.

   *In the **Smart Map**, all colored rectangles of the Network Editors opened are displayed in full.*

2.10.5 Zooming in or out on the network in the Smart Map

1. Right-click on the **Smart Map**.
2. Select the entry **Zoom In** or **Zoom Out** in the context menu.

Tip: Alternatively, you can use:

- the PAGE UP and PAGE DOWN keys
- mouse scroll wheel

The network is adjusted in the ***Smart Map****.*

2.10.6 Redefining the display in the Smart Map

You can redefine the display shown in the Network Editor in 2D mode by drawing a new rectangle in the **Smart Map**.

Note: If a network is displayed in several Network editors, the **Smart Map** uses rectangles to indicate the different views.

1. Click on the Network Editor.

   The Network Editor is activated.

2. In the **Smart Map**, point the mouse pointer to a position outside of all rectangles.

Note: If you want to start drawing the rectangle within a rectangle, you also have to hold down the SHIFT key in the next step.

3. Hold down the mouse button.

   The pointer appears as magnifier.

4. Drag the rectangle to the desired size and release the mouse button.

The network window view is adjusted according to the ***Smart Map****.*

Tip: Alternatively, you can select **Redefine section** in the context menu and draw a rectangle.

2.10.7 Defining a Smart Map view in a new Network Editor

In the **Smart Map**, you can drag open a rectangle, automatically open a new Network Editor, and show the network in the rectangle map section.

Note: If a network is displayed in several Network editors, the **Smart Map** uses rectangles to indicate the different views.

1. Right-click on the **Smart Map**.
2. From the shortcut menu, choose **Define Section in New Network Editor**.

   The mouse pointer becomes a cross.

3. In the **Smart Map**, point the mouse pointer to a corner of the desired map section.
4. Hold down the mouse button.

5. Drag the rectangle to the desired size and release the mouse button.

*A new Network editor is opened. The network window view is adjusted according to the **Smart Map**.*

2.10.8 Moving the Smart Map view

1. Click into the **Smart Map**.
2. On your keyboard, press the arrow button of your choice.

Tip: Alternatively, you can hold down the mouse wheel.

*The **Smart Map** view is moved.*

2.10.9 Copying the layout of a Network Editor into Smart Map

The layout of the smart map is saved additionally to the layouts of the network editors in the **.layx* layout file. You can change the layout of the **Smart Map** by copying the layout of a network editor.

1. Click on the Network Editor.

   The Network Editor is activated.

2. Right-click into the **Smart Map**.
3. In the context menu, choose **Apply Layout** > **From Current View**.

*The **Smart Map** is updated.*

2.10.10 Displaying or hiding live map for the Smart Map

You can select a live map for the Smart Map and activate or deactivate it.

1. Right-click on the **Smart Map**.
2. From the **Background Maps Configuration** context menu, choose the desired entry:

Element	Description
Bing Maps (aerophoto)	Display Bing Maps in Smart Map
Open Street Map (Mapnik)	Display Open Street Map (Mapnik) in Smart Map
Disable Background Maps	Hide the currently selected live map from the Smart Map

*The **Smart Map** is updated.*

3. At the bottom of the overview map, click the hyperlink of the map provider.

   The Internet page with the license terms of the map provider opens.

4. Read the license terms.

2.11 Using network editors

In Vissim you can display one or multiple network editors, e.g. to show the network in the 2D and 3D mode or in different scales.

In each network editor you can display and edit the open network, zoom out for an overview or zoom in to show more details. You can display the network with different graphic parameters in the Network Editors; for example, two dimensional in one editor and three dimensional in another.

If you run simulations, these are shown in the open Network Editors. You can control the display of network objects for each network editor using the visibility (see "Using the Network object toolbar" on page 61).

In Vissim, you can move and arrange Network editors to other positions on the desktop or anchor them in other windows of Vissim (see "Changing the display of windows" on page 89). The positions are saved in the **.layx* file.

Above each open Network Editor, a toolbar with functions that you can use to control the appearance of the network in that Network Editor is shown.

Note: The display in the network editor is linked to the display in **Smart Map** (see "Using the Smart Map" on page 71). If you change the section in the Network editor, the position of the colored rectangle or cross hairs in **Smart Map** also changes and vice-versa.

2.11.1 Showing Network editors

In Vissim, you can show one or more Network editors. In each Network editor, you can open, show and edit the network, reduce its size for an overview or enlarge it to show more details. You can further watch dynamic objects, e.g. vehicles, pedestrians or signal heads changing their status, during a simulation run.

- In the **View** menu, choose > **Open New Network Editor**.

The new Network Editor opens as a tab. The position depends on how your user interface is set up.

Tip: You switch between open lists and Network Editors with CTRL+TAB.

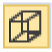
2.11.2 Network editor toolbar

Toolbar button	Name	Description	Keyboard / mouse
Network editor layout selection list		- Save specified network editor layout (see "Saving the user interface layout" on page 146) - Select specified network editor layout	

Toolbar button	Name	Description	Keyboard / mouse
	Edit basic graphic para-meters	Edit basic graphic parameters (see "List of base graphic parameters for network editors" on page 171)	
	Toggle wire-frame on	If wireframe display is enabled, disable wireframe display. The icon remains selected and the function remains active, until you click the icon again. You cannot activate or deactivate the Wireframe view in 3D mode. In the Wireframe view, only the middle lines are shown for links and connectors. Standard colors: ▸ Blue: Links ▸ Pink: Connectors In the Wireframe view in 3D mode, areas are shown in the display type selected.	
	Toggle wire-frame off	If wireframe display is disabled, change to wireframe display. The icon remains selected and the function remains active, until you click the icon again.	
	Toggle back-ground maps on	If background map display is enabled, hide the background map. The icon remains selected and the function remains active, until you click the icon again.	
	Toggle back-ground maps off	If the background map is hidden, display the background map (see "Using live maps from the Internet" on page 394). The icon remains selected and the function remains active, until you click the icon again.	
	Legend on	The legend appears. The legend is displayed if the **Show legend** attribute has been selected in the graphic parameters of the network editor and the **Legend includes <Attribute >** attribute has been selected in the graphic parameters for at least one network object type.	
	Legend off	Legend is hidden.	

Toolbar button	Name	Description	Keyboard / mouse
	**Toggle selection**	If network objects are overlapping each other at the position that you clicked, select the next network object. For instance, for a link with the attribute **Is pedestrian area** (option **Use as pedestrian area**), you can then select the other respective direction. The icon is only active if network objects lie on top of each other and one of the network objects on the top is selected.	TAB
	**Copy selection**	Copy the selected network objects to the Clipboard (see "Selecting and copying network objects" on page 340)	
	**Paste from clip-board**	Paste network objects from the Clipboard into the Network Editor (see "Pasting network objects from the Clipboard" on page 341)	
	**Synchronization: Auto pan on**	The display of the selected network objects is centered automatically in the Network Editor. If you change the selection of network objects in a different Network editor or synchronized list, the section in the Network editor is automatically adjusted to the new selection. The icon remains selected and the function remains active, until you click one of the following icons: - the  **Auto pan** button again -  **Auto zoom** button	
	**Synchronization: Auto pan off**	In the Network Editor, the section is not automatically adjusted to the selected network objects.	

Toolbar button	Name	Description	Keyboard / mouse
	Synchronization: Auto zoom on	Display the selected network objects centered in the Network Editor and select the section so large that all the selected network objects are displayed. If you change the selection of network objects in a different Network editor or synchronized list, the section in the Network editor is automatically adjusted to the new selection. The icon remains selected and the function remains active, until you click one of the following icons: - the **Auto zoom** button again - **Auto pan** button	
	Synchronization: Auto zoom off	In the Network Editor, the section is not automatically adjusted.	
	Show entire network	Choose a section size large enough to display the entire network.	
	Zoom in	(see " Zooming in" on page 82)	PAGE DOWN or move mouse wheel forward
	Zoom out	(see " Zooming out" on page 82)	PAGE UP or move mouse wheel backward
	Previous view	Display previous view. You must have previously displayed views.	ALT+ left arrow key
	Next window section	Display next view. You must have displayed some previous views beforehand.	ALT+ right arrow key
	Pan	Move entire network section. Selected and non-selected network objects are moved. The icon remains selected and the function remains active, until you click the icon again.	Arrow keys or pressed mouse wheel
	Measuring distances	Distances are measured in 2D. The height of areas and z values at link spline points are not taken into account (see "Measuring distances" on page 84).	CTRL+M

Toolbar button	Name	Description	Keyboard / mouse
	**Rotate mode (3D) on**	only in 3D mode: Tilt or rotate the network display level vertically or horizontally (see "Navigating in 3D mode in the network" on page 193). The icon remains selected and the function remains active, until you click one of the following icons: - the  **Rotate mode (3D)** button again - the  **Flight mode** button - the  **2D/3D** button	Hold down left mouse button and drag the mouse, or press the ALT key + hold mouse wheel down
	**Rotate mode (3D) off**	Rotate mode is not selected	
	**Flight mode (3D) on**	only in 3D mode: Move current camera position over network (see "Flight over the network" on page 195) The icon remains selected and the function remains active, until you click one of the following icons: - the  **Flight mode** button again - the  **Rotate mode (3D)** button - the  **2D/3D** button	
	**Flight mode (3D) off**	Flight mode is not selected	
	**Export image (Screenshot)**	Opens the **Save as** window. You can save a screenshot of the Network editor as a graphic file to a folder of your choice. You can saved data in the following file formats: - **.png* - **.jpg* - **.tiff* - **.bmp* - **.gif*	

Toolbar button	Name	Description	Keyboard / mouse
	2D/3D	2D mode is currently displayed. Switch to 3D mode The icon remains selected and the function remains active, until you click the icon again.	
	2D/3D	Currently shows 3D mode. Switch to 2D mode. The icon remains selected and the function remains active, until you click the icon again.	
	Edit 3D graphic parameters	Edit 3D graphic parameters (see "Editing 3D graphic parameters" on page 194), (see "List of 3D graphic parameters" on page 194) The 3D graphic parameters only affect the network objects in 3D mode.	
Selection of camera position list		only in 3D mode: ▸ Saving camera positions ▸ Select saved camera position	
Tip: Alternatively, in the **Camera Positions** list, select one of the saved camera positions. Then from the context menu, choose **Apply to current Network Editor** to assign the camera position to the current Network editor.			

2.11.3 Network editor context menu

The context menu in a network editor can display different features. They depend on whether network objects have been selected, you right-click a network object, or right-click a section in which no network object has been inserted.

2.11.3.1 Commands that do not depend on a previous selection of objects

You can always perform the following functions via the context menu, regardless of whether you click with the right mouse button on a network object or whether network objects are selected:

- **Add <network object type>**: Add a network object of the type selected in the Network objects toolbar, if the network object does not have to be added to another network object. Possible options are links, areas or nodes.
- **Paste**: Paste network objects that you have copied to the Clipboard.
- **Read Additionally Here**: Read a network file **.inpx* additionally.
- **Show List**: Open the network objects list of the network object type that is selected on the Network Objects toolbar.
- **Map this Point to Background Position**: Specify a point in the background map as a reference point (see "Mapping Vissim network to background position" on page 396).

2.11.3.2 Commands that depend on a previous selection of objects

The following functions are displayed in the context menu depending on whether network objects are selected:

Network objects selected in the network editor	Functions
No network objects selected	**Show List**: Show the network objects list of the network object type that is selected for the Insert mode in the network objects toolbar.

One network object is selected	- The standard editing functions are shown, for example:
	  - **Zoom To Selection**
	  - **Delete**
	  - **Duplicate**
	  - **Copy**
	  - **Paste**
	  - **Edit**
	  - **Show In List**
	  - **Reset label position**
	  - **Create Chart for Selected Objects**
	- Network object type specific functions, e.g. **Split link here**

Multiple network objects of a network object type or different network object types are selected	- Only the standard editing functions that can be carried out when multiple network objects are selected are shown, for example:
	  - **Zoom To Selection**
	  - **Delete**
	  - **Duplicate**
	  - **Copy**
	  - **Paste**
	  - **Show In List**
	
	Editing functions that are available for a single selected network object are not displayed.
	- Only functions specific to the network object type that are useful when multiple network objects are selected are shown.

2.11.3.3 Deselecting all network objects

If network objects have been selected and you right-click a section (not a network object) into which no network object has been inserted to open the shortcut menu, all network objects are deselected.

2.11.4 Zooming in

To maximize the view, on the toolbar, click the respective button or use the scroll wheel of the mouse.

Tip: You can also change the Network editor view using the **Smart Map** (see "Zooming in or out on the network in the Smart Map" on page 72).

2.11.4.1 Enlarging the view to a certain section

In 2D mode, click the **Zoom in** symbol to enlarge the view of a specific section. In 3D mode, click the **Zoom in** symbol to gradually enlarge the view.

1. On the Network editor toolbar, click the **Zoom in** button.

   The pointer appears as magnifier.

2. Click the position in the network from which you want to zoom into a section, and hold the mouse button down.
3. Drag the desired section and release the mouse button.

The selected section will be zoomed in. The mouse pointer is reset to standard display.

2.11.4.2 Zooming in with a scroll wheel

1. In the Network editor, point the mouse pointer to the position you want to zoom in on.

   Next, simultaneously hold down the SHIFT key. Then Vissim will enlarge the view faster.

2. Turn the scroll wheel forwards.

The view is zoomed in at the position of the mouse pointer.

If you zoom in until the maximum magnification, the contents of the Network editor are displayed in black.

Tip: Alternatively, press the PAGE UP key to zoom in gradually.

2.11.5 Zooming out

To minimize the view, on the toolbar, click the respective button or use the scroll wheel of the mouse.

Tip: You can also change the Network editor view using the **Smart Map** (see "Zooming in or out on the network in the Smart Map" on page 72).

2.11.5.1 Zooming out using icon

In 2D mode, click the **Zoom out** symbol to reduce the view of a specific section. In 3D mode, click the **Zoom out** symbol to gradually reduce the view.

- On the Network editor toolbar, click the **Zoom out** button.

  The view is reduced.

2.11.5.2 Zooming out using a scroll wheel

1. In the Network editor, point the mouse pointer to the position you want to zoom in on.

   Next, simultaneously hold down the SHIFT key. Then Vissim will reduce the view faster.

2. Turn the scroll wheel backwards.

The view is zoomed out at the position of the mouse pointer.

Tip: Alternatively, you can zoom out by gradually. To do so, press the PAGE DOWN key.

2.11.6 Displaying the entire network

- On the Network Editor toolbar, click the **Show entire network** button.

  The full network is shown in the network editor.

Tip: Alternatively you can also display the entire network by pressing the HOME key.

2.11.7 Moving the view

1. On the Network editor toolbar, click the **Shift mode** button.

   *The **Shift mode** button is activated: . The mouse pointer becomes a symbol. This designates the Pan mode.*

2. Click any position in the network and keep the left mouse button pressed.
3. Drag the view to the desired position and release the left mouse button.

   The view is shifted.

4. In the Network Editor, on the toolbar, click the **Shift mode** button to deactivate the 'Pan' mode.

*The **Shift mode** button is deactivated: . The mouse pointer is reset to the standard depiction.*

Note: Only the display is shifted, the network itself is not. Coordinates remain unchanged.

Tips: Alternatively, you have these possibilities to shift the view:

- Hold down the middle mouse button or the scroll wheel and move the view.
- Press an arrow key on your keyboard. The section moves to a greater extent if you simultaneously hold down the SHIFT key.
- You can also move the view in the **Smart Map** (see "Moving the Network Editor view" on page 72).

2.11.8 Measuring distances

In the network editor, you can set measurement points in 2D mode. Vissim measures and displays the distance between consecutive measuring points. This allows you to determine the length of a network object, the length of a link or the distance between network objects, for example. Vissim displays the following values:

- distance between successive measurement points
- at each measurement point, the distance to the first measurement point

measurement point in the network or on a network object

You can set measurement points at the following positions:

- at the desired position in the network editor
- at the desired position on a network object
- precisely in the middle of the corner point of a construction element

  When you try to place the measurement point precisely in the middle of a corner point, Vissim assists you by showing the relevant corner points. In this case, however, the following criteria must be fulfilled:

  - On the network editor toolbar, **Measure Distance** is selected.
  - The mouse pointer is close to one or multiple corner points of construction elements.
  - The construction element is not selected, as then the corner points would already be displayed.

  When you point the mouse pointer to a corner point, the point is enlarged. This makes it easier for you to click a point. Vissim sets the measurement point precisely in the middle of the corner point.

  When you point the mouse pointer near the start or end of a link, it jumps to the respective section. This makes it easier for you to precisely click the beginning or the end of the link.

Direction of measurement

Vissim measures the distance between successive measurement points. For each measurement point you set, you can change the direction of measurement before setting the next measurement point. This is not the case when measuring the distance on a route, connector or link. With these network objects, you can only measure the distance in the direction of travel.

Pictorial representation of the distance measured

The measurement points are connected via a yellow line. This line is how Vissim depicts the distance between two measurement points in a text box. The text box is aligned parallel to the yellow line.

Vissim indicates the distance from a measurement point to the first measurement point in a horizontally aligned text box, immediately next to the measurement point.

Vissim indicates the distance in the unit selected by default in the network settings.

You can also measure distances during a simulation run.

Setting measurement points and measuring distances

1. Zoom in on your Vissim network until you reach an enlargement where your Vissim network is shown in enough detail to set a measurement point at where you want to start the measurement.
2. Ensure that on the toolbar of the network editor, the **Measure distance** button is selected.

Tip: Alternatively,press CTRL+M to measure a distance.

3. Click the position where you want to start the measurement.

   A measurement point is inserted. On links and connectors, the measurement point is displayed as a yellow cross section that extends across all lanes.

   You may perform the next step multiple times and thus measure successive distances between multiple measurement points:

4. Click the position up to which you want to carry out the measurement.

   A yellow line connects the measurement points. The distance between the measurement points is displayed centered on the line. If you have inserted multiple measurement points, the distance to the first measurement point is displayed at each measurement point.

   With the next step, you stop measuring the distance and hide measurement points, yellow lines and text boxes.

5. Perform a double-click.

Tip: Alternatively, to stop measuring the distance, press the ESC or ENTER key.

2.11.9 Defining a new view

In the Network Editor, you can specify a different network section by dragging open a rectangle in the **Smart Map** (see "Using the Smart Map" on page 71).

1. In the **Smart Map**, click the position at which you want to set a corner of the new display outside of a colored rectangle and hold down the left mouse button.

The mouse pointer becomes a ![zoom] *symbol.*

2. Diagonally drag open the rectangle of your choice.
3. Release the mouse button.

The Network Editor view is adjusted according to the ***Smart Map****.*

2.11.10 Displaying previous or next sections

If you have changed the section in the network editor, you can return to the previous section and display it. If a previous section is displayed, you can go to the next section and display it.

1. In the Network editor, on the toolbar, click the **Previous view** button.

   The previous section is displayed.

   Tip: Alternatively, you can change to the previous view with the ALT + LEFT ARROW key.

2. In the Network Editor, click on the icon **Next window section** in the toolbar.

   The next section is displayed.

   Tip: Alternatively, you can change to the previous view with the ALT + RIGHT ARROW key.

2.11.11 Zooming to network objects in the network editor

In the network editor you can select the size of the network so that the selected network objects are automatically completely displayed.

- In the context menu, select **Zoom To Selection**.

2.11.12 Selecting network objects in the Network editor and showing them in a list

In the network editor you can select network objects of a particular network object type and show them, together with their attributes, in a list of network objects with the particular network type.

1. In the Network Editor, right-click the network object of your choice.
2. From the context menu, choose entry **Show In List**.

   The list of defined network objects for the network object type opens.

The objects selected in the Network editor are marked in the list, if the list is synchronized (see "List toolbar" on page 97).

2.11.13 Using named Network editor layouts

The layout of a Network editor is defined by graphic parameters and settings in the network objects toolbar. You can assign the current layout of a Network editor a name, under which you can import the layout again later on. You can save these so-called named Network editor layouts to the **.layx* file. In a Network editor, you can choose a Network editor layout from all the named ones available, to change the graphic parameters and network object toolbar settings of your window accordingly.

2.11.13.1 Creating a Network editor layout

1. Adapt the graphic parameters of a Network editor via its toolbar and/or the network objects toolbar (see "Editing base graphic parameters for a network editor" on page 171), (see "Using the Network object toolbar" on page 61).
2. On the Network editor toolbar, in the **Network editor layout selection** box, enter a unique name.
3. Confirm with ENTER.

On the toolbar of all Network editors, in the ***Select layout*** *list box, the new network editor layout is displayed and can be selected.*

2.11.13.2 Assigning a Network editor layout

1. In the Network editor, in the **NetworkEditorLayout** list, click the symbol.
2. Select the Network editor layout of your choice.

The network editor layout is assigned to the network editor and the display is adjusted. The new settings are shown on the network objects toolbar.

2.11.13.3 Saving Network editor layouts

1. In the menu, select **File** > **Save Layout as**.
2. Enter a unique name.
3. Click the **Save** button.

*The current layout of the entire Vissim user interface, and thus all named Network editor layouts, are saved to the *.layx layout file.*

If in the User Preferences, the option **Auto-save layout when network file (inpx) is saved** is selected, the layout file is saved automatically under the name of the currently loaded network file, each time the network file is saved (see "Specifying automatic saving of the layout file *.layx" on page 154).

2.11.13.4 Reading in saved Network editor layouts additionally

You may read all Network editor layouts, saved to a **.layx* file, into your currently opened file. Then you can apply these Network editor layouts to the Network editors in the file currently open.

1. From the **File** menu, choose > **Read Additionally** > **Named Network Editor Layouts**.
2. Select the desired **.layx* file of your choice.

3. Click the **Open** button.

*The network editor layouts are read in. On the toolbar of the Network editors opened and the Network editors you open later on, in the **Select layout** list, you can select the Network editor layouts read in.*

2.11.13.5 Deleting a named Network editor layout

1. On the Network editor toolbar, right-click in the **Network editor layout selection** list box.
2. Position the mouse pointer on the **Delete** entry in the context menu.

   The named Network editor layouts are displayed in the context menu.
3. Click the desired Network editor layout in the context menu.
4. Confirm with **Yes**.

2.12 Selecting simple network display

To gain a better overview of complex networks, switch to the simple network display that hides all network objects lying on links, areas, ramps and stairways.

1. In the **View** menu, select > **Simple Network Display**.

Tip: Alternatively, press CTRL+N or on the **Edit** toolbar, click **Simple Network Display**.

The following network objects and their labels are hidden in the Simple Network Display. This is also the case for network objects of the network object type selected during a simulation run in the network objects toolbar, which allows network objects to be added:

Desired Speed Decisions Reduced Speed Areas Conflict Areas Priority Rules Stop Signs Signal Heads Detectors Vehicle Inputs VehicleAttributeDecisions Parking Lots Public transport stops	Public Transport Lines Nodes Data Collection Points Vehicle Travel Times Queue Counters Flow bundles Pavement Markings 3D Information Signs Elevators Pedestrian Inputs Pedestrian Routes Pedestrian Attribute Decisions Pedestrian Travel Times

In the Simple Network Display, all other objects are displayed, if you do not individually set them to invisible:

Links	Vehicles In Network
Vehicle Routes	Pedestrians In Network
Backgrounds	Areas
3D Traffic Signals	Obstacles
Static 3D Models	Ramps & Stairs

2. To show the hidden objects again, from the **View** menu, choose > **Simple Network Display** again.

2.13 Using the Quick Mode

In the Quick Mode, all dynamic objects (e.g. vehicles, pedestrians, dynamic labels, and colors) are hidden in all network editors. In addition, in the Quick Mode, list windows and the Quick view are only then updated when you scroll or click in them. The **Messages** window is not updated and does not display any messages. This allows for a maximum simulation speed. The simulation speed set is not used.

- In the **View** menu, choose > **Quick Mode**.

Tip: Alternatively, press CTRL+Q or on the **Edit** toolbar, click **Quick Mode**.

You can deactivate the Quick Mode again:

- Press CTRL+Q again.
- From the **View** menu, choose **Quick Mode**.
- Again click the **Quick Mode** icon.

All dynamic objects are shown again. All lists and the Quick view are updated.

2.14 Changing the display of windows

You can significantly change the display of windows showing network editors, lists, network object toolbars, the Smart Map and Quick View by using standard program functions:

Name or function	Symbol	Description
Minimize		Minimize user interface
Maximize		Maximize user interface
Auto Hide: hide		The window is hidden. A tab with the name of the hidden window is displayed at the edge of the user interface.
Use Auto Hide to show hidden window again		At the edge of the user interface, point the mouse to the tab of the desired window.
Auto Hide: show permanently		The window is permanently shown.

Name or function	Symbol	Description
Close		The window closes.
Move		Moving and rearranging a window in Vissim or arranging a windows outside of Vissim (see "Arranging or freely positioning program elements in PTV Vissim" on page 91)
Anchoring		Anchor windows or show them together in one section (see "Anchoring windows" on page 91)
Releasing from anchors		Release anchored window (see "Releasing windows from the anchors" on page 92)
Change size		Change the size of the window using the corner drag points. The size of windows cannot always be changed.

You can restore the default settings (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).

2.14.1 Showing program elements together

If you open several program elements, you can group them into one area, for example:

- multiple lists
- multiple network editors
- one or multiple lists together with one or multiple network editors
- the Smart Map together with the Quick View
- network objects toolbar and level toolbar together with the background image toolbar

You cannot show lists and network editors together with the Smart Map, the Quick or a toolbar in one area.

Tabs under each area display the program elements inside:

- For lists, the name of the network object types or base data type
- For Network editors, numbers in the order they were opened
- Smart Map and Quick View
- Network object toolbar, level toolbar, background image toolbar: Network object types, levels, background images

The name of the visible program element is highlighted in the tab. The name of the active program element is highlighted in the title bar.

You can change the arrangement of program elements (see "Changing the display of windows" on page 89), (see "Anchoring windows" on page 91), (see "Arranging or freely positioning program elements in PTV Vissim" on page 91).

2.14.2 Arranging or freely positioning program elements in PTV Vissim

You can move and arrange program elements in Vissim or move them on your screen or screens independently of Vissim. You can also group multiple program elements together, for example, multiple lists or multiple network editors.

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

1. Click the title bar of the program element, keep the mouse button held down, and drag the program element to the position of your choice.

   The icon for anchoring the program element is shown until you drag the program element out of Vissim.

2. Release the mouse button.

Outside of Vissim, the display of the program element floats freely.

You can restore the default settings (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).

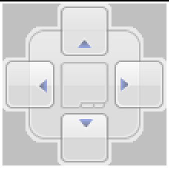
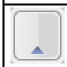
2.14.3 Anchoring windows

You can anchor windows such as network editors, lists, bars, Smart Map and Quick View or display them together in one area. Vissim assists you with visual aids. An icon and a colored shadow show you where you can anchor the window.

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

1. Click on the title bar or the tab for the window and hold the mouse button pressed.

   If you move the mouse, an icon is displayed, which shows you the possible anchoring positions:

Element	Description
	➤ Outer icons: This anchors the window in the destination area at one of the four edges. ➤ Icon in the middle: This anchors the window as a tab. Note: You cannot anchor all windows with all other windows as a tab. You cannot anchor network editors and lists with the **Quick View**, **Smart Map**, **Network objects toolbar**, **Levels toolbar** or the **Background toolbar** as a tab. If anchoring as a tab is not possible, the following symbol is displayed: 🚫
	Anchoring the window at the top edge of the destination area
	Anchoring the window at the bottom edge of the destination area
	Anchoring the window at the left-hand edge of the destination area
	Anchoring the window at the right-hand edge of the destination area

2. Drag the mouse pointer to the desired icon.

   The desired target area is given a colored shadow.

   Note: The icon which you drag with the mouse pointer is decisive, not the position of the window.

3. Release the mouse button.

The window is anchored at the desired position.

You can restore the default settings (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).

If several window are anchored together in an area, they are shown as tabs.

2.14.4 Releasing windows from the anchors

You can release anchored windows from each other.

1. If in a window, you have attached multiple tabs next to each other and want to detach them, click the title bar and hold down the mouse button.
2. If in a window, you have attached multiple tabs next to each other and want to detach one, click the tab window and hold down the mouse button.
3. Drag the mouse pointer to the desired area of the desktop and release the mouse button.

The window is released from the anchor.

You can restore the default settings (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).

2.14.5 Restoring the display of windows

1. Choose **Edit** > **User Preferences**.

   *The **User Preferences** window opens.*

2. Choose the entry **GUI** > **General**.
3. Click the **Reset dialog positions** button.

The next time you start Vissim without a network, the default settings are used to arrange the windows.

2.14.6 Switching between windows

If you have opened multiple windows, you can switch between them to place another window in the foreground. This also includes the start page. The **View** toolbar list shows the windows opened in Vissim.

- On the **View** toolbar, in the **Windows Selection** list, click the window of your choice.

The window selected is placed in the foreground and is activated.

2.15 Using lists

In a list, you can show all objects of a type together with a selection of the object's attributes (see "Opening lists" on page 95):

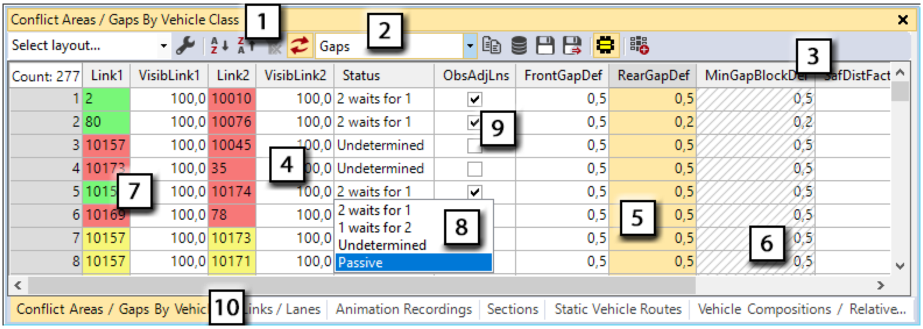
- You can display input attributes, e.g. for links these are the length, name, link behavior type, display type, etc. You can change input attributes in the list, e.g. when you create new network objects or make changes later on (see "Editing attributes of network objects" on page 350), (see "Selecting and editing data in lists" on page 100). This is not always possible during a simulation run. In this case, a corresponding message is displayed when you try to do so.
- You can display attributes calculated from input attributes, e.g. a distance or travel time measurement. The values of these attributes cannot be changed in the list.
- You can display result attributes, whose values are determined during a simulation run (see "Displaying result attributes in attribute lists" on page 1017). Result attributes can have subattributes, e.g. a different value per simulation run and time interval. The values of these attributes cannot be changed in the list.

There are two types of lists:

- "Normal" attribute lists contain the attributes of network objects and base data. You can choose these lists from the **Lists** menu (see "Opening lists" on page 95): A list contains a row for each object of the respective type, e.g. for a defined link. There is one column for each attribute, e.g. **name** or **length** of the link. There is also a column for each subattribute, e.g. time intervals or vehicle classes. "Normal" attribute lists can be extended to coupled lists, if the network object type has relations to other objects, e.g. a link to lanes (see "Using coupled lists" on page 119).

- To select result lists, from the **Lists** menu, choose > **Results** or from the **Evaluation** menu, choose > **Result Lists** (see "Configuring evaluations of the result attributes for lists" on page 1014): A results list contains rows for each object of the respective type per time interval of each simulation run. Moreover, you can show additional rows for static results: the mean, the standard deviation and minimum and maximum values of all simulation runs. You can further view the mean, the standard deviation, the minimum and maximum values and total of all time intervals. Only the subattributes **Vehicle class** or **Pedestrian class** are shown in the attribute columns.

2.15.1 Structure of lists



Element	Description
(1) Title bar	Name of list **Auto Hide**: Hide and show lists **Close list**
(2) Toolbar	Functions for displaying and editing (see "List toolbar" on page 97)
(3) Column header	Short name of attribute Sort, move and mark columns Functions are available via the context menu (see "Editing lists and data via the context menu" on page 103) To open a quick info with an attribute description, point the mouse pointer to the attribute name in the column header. Right-click the column header to select the data in the column.
(4) Data	Attribute values. Functions are available via the context menu. If you point to the first column of a list with the cursor, the row number is shown.
(5) Orange cells	Data is selected in one or more cells, rows or columns, and can, for example, be copied or changed.

Element	Description
White cells	Values you can change, e.g. of input attributes - In cells for selecting values, the button is shown when you click in the cell. - For cells that allow the direct entry of values, click the cell and enter the data.
Gray cells	Values of calculated attributes or result attributes. You cannot change them.
(6) Hatched cells	Values of irrelevant attributes with no impact due to the values of other attributes, e.g. a Wiedemann 74 parameter, when for a driving behavior, the type Wiedemann 99 is selected.
(7) Red, green, yellow cells	Only for conflict areas (see "Defining the right of way at conflict areas" on page 564)
(8) Cells with list boxes	For some attributes, in the cells, in list boxes, you can select or add attribute values. - If you point the cursor to a cell that contains a selection list, the button is displayed. - Click the symbol to select attribute values in the list box or click **Add** to define a new attribute. Tip: If no cell is selected, alternatively, double-click a cell. A list box opens. Select the entry of your choice. Note: Do not double-click a cell that is already selected! By doing so, you add the first entry of the list box into the cell.
(9) Options	Some cells of attributes contain a list of options that allows you to select or add attribute values. Options will be displayed, if the selection of multiple entries is allowed. ☑ If this option is selected, the entry is considered.
(10) Tab	When multiple lists are open, they are shown as tabs. You can thus quickly show the desired list in the foreground. Tip: Alternatively you can press CTRL+TAB. You thus switch between open lists and Network Editors.

2.15.2 Opening lists

You can open lists using the following elements of the user interface: If a list of the same type is open, this list will be shown in the foreground.

Element	Description
Lists menu	If no list of the same type has been opened yet, a list showing all objects of the selected entry will be displayed.
Menu: **Base Data**, **Traffic**, **Signal Control**, **Evaluation**, **Presentation**	Opens a list with all objects of the selected entry
Context menu in the Network objects toolbar > **Show List**	Opens a list with the network objects, attributes and attribute values to the selected network object type
Shortcut menu of the Level toolbar > **Show List**	Opens the **Levels** list with the defined levels, attributes and attribute values
Shortcut menu of the Background image toolbar > **Show List**	Opens the **Backgrounds** list with the defined backgrounds, attributes and attribute values
Shortcut menu in Network editor > **Show In List**	▶ If no network object is selected in the Network Editor: a list of network objects, attributes and attribute values of the network object type opens, which is selected in the network object toolbar. ▶ If one or more network objects of a network object type is selected in the Network Editor: A list with the network objects, attributes and attribute values to the selected network object type opens. The selected network objects are marked in the list.
Tabs with list names	If you open additional lists, these are shown as tabs at the bottom of the first list.
Network editor	For network object types, whose attributes you can only edit in the network objects list of the respective object type: Double-click the network object to open the list.

If you open a list with network objects, you activated synchronization in this list, and in the Network editor, you selected the network objects of the network object type of the list, these network objects are highlighted in the list.

Tips:

- You can open multiple lists and arrange them on the user interface or on multiple screens.
- You can select which attributes are displayed in the list.

2.15.3 Selecting network objects in the Network editor and showing them in a list

In the network editor you can select network objects of a particular network object type and show them, together with their attributes, in a list of network objects with the particular network type.

1. In the Network Editor, right-click the network object of your choice.
2. From the context menu, choose entry **Show In List**.

   The list of defined network objects for the network object type opens.

The objects selected in the Network editor are marked in the list, if the list is synchronized (see "List toolbar" on page 97).

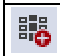
2.15.4 List toolbar

Your settings are saved to a **.layx* file as soon as you save the network file.

Icon	Name	Description
Select layout...	**List layout selection** list	➤ Save named list layout (see "Using named list layouts" on page 111) ➤ Select named list layout and apply to list
	Attribute selection	Open the **Select Attributes** window and select attributes which are shown column by column in the list or whose columns you want to hide (see "Selecting attributes and subattributes for columns of a list" on page 112).
	Add	Adds a new row to the list to create a new object in it. If there is a window for the object in which attributes can be entered, this window automatically opens, provided that the respective setting has been selected under user preferences (see "Right-click behavior and action after creating an object" on page 152).
	Edit	For the object selected, opens the window in which you can edit attributes of the object. If there is no such window for the network object type or the base data type, the symbol is not shown.
	Delete object(s)	Deletes selected objects from the list. If the object is a network object, it is also deleted from the network editor. Conflict areas cannot be deleted.
	Duplicate object (s)	Copies the object from the list. If the object is a network object, it is also copied in the network editor. The object is inserted as a duplicate: ➤ In the list, in a new row, with a new unique number. ➤ If the object is a network object, in the network editor, the duplicate is placed on the original network object and can then be moved. Conflict areas cannot be duplicated.
	Sort ascending	Sorts a list by one or multiple columns in ascending order (see "Sorting lists" on page 106)
	Sort descending	Sorts a list by one or multiple columns in descending order

Icon	Name	Description
	Set all column filters passive	If in the list columns are filtered, deactivate the filters to show all data in the list again (see "Filtering data of a column" on page 107).
	Synchronization	Synchronizes list with all network editors, other synchronized lists and the Quick View. If you select or deselect network objects in the list, these are also selected or deselected in other windows. If the **Auto pan** icon is selected in a Network editor, the selected network objects are shown automatically centered in this Network editor. If you change the selection of network objects in a different Network editor or synchronized list, the section in the Network editor is automatically adjusted to your selection. If the **Auto zoom** icon is selected in a Network editor, the selected network objects are shown centered in this Network editor, and the section is selected so large that all selected network objects are shown. If you change the selection of network objects in a different Network Editor or synchronized list, the section in the Network Editor is automatically adjusted to your selection. You can synchronize the result lists of simulation runs, vehicles in the network, pedestrians in the network, and paths. Synchronization, however, only has an effect on the Quick View (see "Using the Quick View" on page 68). For other result lists, the **Synchronization** icon is not available.
	No synchronization	The list is not synchronized with other program elements. Synchronization is not selected as standard in lists with measurement results.
<Single List> **Relations** list		➤ In simple lists: Select a relation for the objects of the list with other objects, and show the other objects in a coupled list on the right, e.g. the lanes of a link. ➤ In coupled lists: Select a relation for the objects of the left list with other objects, and show these objects in the list on the right (see "Using coupled lists" on page 119). The entry **Single List** only displays the left list and hides the right list with the relation.
	Copy	Copy content of tab-separated rows selected to the Clipboard.

Icon	Name	Description
	**Save to data-base**	Save list as database. Opens the **Evaluations (Database)** window. A database connection must be configured (see "Configuring the database connection for evaluations" on page 1018). ➤ In the **Database Connection** section, select > **Data Link Properties...**: the database connection last set ➤ In the **Database table** section, select > **Table name**: by default name of current table. Avoid using spaces in a table name. If in the list data is hidden through active filters, only the data shown is saved. Hidden data will not be saved.
	**Save to file**	Opens a window where you can specify a file name under which the list is saved as a Vissim attribute file **.att*. In the Attribute file, the data is output in rows and is separated by a semi-colon for each column. In the header of the attribute file, the legend specifies the attributes in the attribute file, listing their short and long name. The short and long names are displayed on the GUI in the language currently selected under User Preferences. If in the list data is hidden through active filters, only the data shown is saved. Hidden data will not be saved.
	Selected: **Autosave after simulation**	For the network object type of the current list, data and result attributes of the simulation run completed are automatically saved to the current evaluation output directory, to a file and/or database. At the end of the simulation, the list must be open to save the data. If under Result Management, you selected **Keep result attributes of previous simulation runs**, Vissim will add the number of the simulation run to each file name, according to the following convention: *<File name *.inpx>_<attribute name>_<number of simulation run>.att*. If under Result Management, you selected **Delete previous simulation runs** and only perform one simulation run, the naming convention is as follows: *<File name *.inpx>_<attribute name>_001.att*. If under Result Management, you selected **Delete previous simulation runs** and using the parameter **Number of runs:**, perform several simulation runs, the data of each simulation run is saved to a separate file. The file names then include the number of the respective simulation run. For automatic output into a database, the database configuration is used that has been defined for evaluations (see "Configuring the database connection for evaluations" on page 1018).

Icon	Name	Description
		If in the list data is hidden through active filters, only the data shown is saved. Hidden data will not be saved.
	Not selected	For the network object type of the current list, do not automatically save data and result attributes of the simulation after the expiry of the simulation.
	**Show passive conflict areas**	only for conflict areas: shows also the passive, yellow (by default) conflict areas in the list which have no effect on traffic
		only for conflict areas: shows only active conflict areas (which have no effect on traffic)
	**Create User-Defined Attrib-ute**	Open the **User-Defined Attribute** window and create an attribute (see "Using user-defined attributes" on page 210)

2.15.5 Selecting and editing data in lists

To a large extent, you can select the common functions and key combinations in spreadsheet programs and subsequently edit, copy, delete or paste the data. Some attributes cannot be changed; these cells are displayed in gray in the lists.

2.15.5.1 List sections in which you can select data

You can select the following areas in lists:

- all cells
- an individual cell
- adjacent cells in multiple rows
- adjacent cells in multiple columns
- adjacent cells in multiple rows and columns
- non-adjacent cells
- row by row
- column by column
- non-adjacent rows or columns

You edit data in lists depending on their data type. The following types of cells exist:

- Cells, in which you enter text or values
- Cells, in which you select one or more options or for which the selection can be can-celed
- Cells, in which you select one or multiple entries from list box .

2.15.5.2 Entering text or values in a cell

To change the text or value of cell that can be edited, you have to mark the cell.

1. Click or double-click into the desired cell.

   An entry is marked.

   When you click or double-click a selected cell, the cell content is marked and an insertion marker is displayed at the end of the cell.

2. In the cell, click the position to which you want to move the insertion marker.
3. Enter the desired data.
4. To complete your entry, press ENTER.

   The entry is completed.

5. If desired, click another cell.

2.15.5.3 Entering text or values in multiple cells

If you highlight multiple cells that contain similar data, changing the data in one cell will automatically change the data in all highlighted cells.

1. Mark the desired cells in the list.
2. Enter the desired data.
3. To complete your entry, press ENTER.
4. If desired, click another cell.

2.15.5.4 Selection options in cells

Using the mouse, you can only select options for individual data sets or you can cancel the selection.

- Click in the list of the desired option:

  The option is selected.

  The option is not selected.

Using the keyboard, you can also activate or deactivate options for multiple cells at the same time.

1. Mark the desired cells in the list.
2. Press the space bar.

   All options are selected.

   Not all options are selected.

2.15.5.5 Selecting data in cells via list boxes

In cells with list boxes, you can click a value in the list box (which contains various values), e.g. a reference to other objects. Cells with list boxes are marked with the button ▾.

1. Point the cursor at the right border of the cell.
2. Click on the ▾ symbol.

   A list box opens.

You have the following options to highlight the entry:

- Click the desired entry.

  The entry is selected.
- Enter the first or the first few characters of the entry.

  The first entry that begins with the characters entered is selected.

2.15.5.6 Defining data in cells with list boxes

In some cells that contain list boxes, you can create a new entry. The first entry of the list boxes is **Add**.

1. If the desired cell is not selected, double-click the cell. If the desired cell is selected, click the cell.

   A list box opens.
2. In the cell, in the list box, click the first entry **Add**.

   If there is a window for editing the attributes of this object type, and it should be shown according to your user preferences, this window opens. You can enter attributes for the new object. Otherwise, the new object is created with the default values.
3. If the window for editing has opened, enter the data of your choice.
4. Confirm with **OK**.

2.15.5.7 Opening the *Edit <network object name>* window from a list

If for a network object, there is an **Edit <network object name>** window, in the **Edit <network object name>** window, you have the following options to open an attribute list:

- Double-click the row header of the desired entry.
- Double-click a non-editable cell of the desired row.
- In the row of your choice, double-click a non-editable cell next to the selected option ☑ or the deselected option ☐.

2.15.6 Editing lists and data via the context menu

Using shortcut menus, you can edit lists and data in lists. The following elements of a list have a shortcut menu:

- column header
- row header
- cells

Notes:

- Depending on the cells, network object type or base data you selected, some functions might by grayed out or hidden and thus cannot be used.
- Depending on the network object type or base data used, the context menu also contains functions that only allow you to edit specific attributes. These functions are described under network object types and base data (see "Creating and editing a network" on page 334), (see "Base data for simulation" on page 202).

2.15.6.1 Functions available in the shortcut menu of the column header

If you have selected multiple columns and right-click the column header of one of these columns, the selection remains unchanged. The context menu opens.The function you select has an impact on the selected columns.

If you have selected multiple columns and right-click the column header of a column not yet selected, this column will be selected and the selection of the other columns will be undone. The context menu opens.The function you select has an impact on the selected column.

Function	Description
Sort Ascending	Sorts a list by one or multiple columns in ascending order
Sort Descending	Sorts a list by one or multiple columns in descending order
Adding aliases	Opens the **Alias** window (see "Using aliases for attribute names" on page 217). The corresponding network object type and the attribute name will be displayed. These entries cannot be changed. In **Name** field an Alias is suggested. You can overwrite this value.
Editing aliases	For the Alias column only: Opens an **Alias** window(see "Using aliases for attribute names" on page 217). You can change the name of the alias.
Removing aliases	For the Alias column only: You may remove the alias. The original attribute name is shown in the column header.
Set Optimum Width for All Columns	Adjusts column width for each column to accommodate the longest column title and longest cell entry

Function	Description
Set Optimum Column Width	Adjusts column width for each selected column to accommodate the longest column title and longest cell entry
Adjust Column Widths To Window Size	Adjusts column width for all columns to the window width.
Remove Column	Hides the column.
Add Column To The Left	Opens a window that allows you to select an attribute for a column that is inserted to the left of a highlighted column.
Attribute Selection	Opens a window that allows you to select attributes for all columns.

2.15.6.2 Functions available in the shortcut menu of the row header

Function	Description
Add	Add a new row with a new object and selects the row. Some attribute values are set by default. For other attribute values, a window is opened. You can then change attribute values. The **Add** command is not available for all object types.
Edit	A window opens that allows you to edit the input attributes of the object. The **Edit** function is not available for all object types.
Delete	Deletes all objects in the rows selected.
Duplicate	Copies the object and its data and inserts a copy of it into a new row. For objects with a geographic position in the network, the copy lies exactly on the position of the original.
Create User-Defined Attribute	Opens the Create User-Defined Attribute window and automatically adopts the object type of the list (see "Using user-defined attributes" on page 210).
Zoom	Sets the section in the network editor last active, so that the selected network objects are all fully displayed.
Create Chart	The **Create Chart** window opens. The network objects selected in the list are displayed in the **Create Chart** window.

2.15.6.3 Functions available in the shortcut menu of cells

Function	Description
Copy cells	Copies the cells selected to the Clipboard. You can paste the data into Vissim or another program.
Paste cells	Pastes the content of the Clipboard to where the cursor is or to a selected area. Note: Make sure that the data and rows copied to the Clipboard match the data and cells of the list you want to paste the Clipboard content to.
Sort Ascending	Sorts a list by one or multiple columns in ascending order
Sort Descending	Sorts a list by one or multiple columns in descending order
Edit	A window opens that allows you to edit the input attributes of the object. The **Edit** function is not available for all object types.
Delete	Deletes all objects in whose rows at least one cell is selected.
Duplicate	Copies the object and its data and inserts a copy of it into a new row. For objects with a geographic position in the network, the copy lies exactly on the position of the original.
Create User-Defined Attribute	Opens the Create User-Defined Attribute window and automatically adopts the object type of the list (see "Using user-defined attributes" on page 210).
Zoom	Sets the section in the network editor last active, so that the selected network objects are all fully displayed.
Create Chart	Create chart for selected attributes

2.15.6.4 Functions available in the shortcut menu outside of cells

1. Right-click the empty area next to or below the list.
2. Choose the desired entry from the context menu.

Function	Description
Opening new list for <Network object type>	Show another list with network objects of network object type including the network objects' attributes. You can select the function in the shortcut menu of the list of a relation, if the coupled list has been opened and the relation list contains network objects of the network object type with their own list.
Creating user-defined attributes	(see "Using user-defined attributes" on page 210)

2.15.7 Selecting cells in lists

In a list, you can use the mouse and keyboard to select cells:

Purpose	Description
Selecting a cell in a list	Click the cell.
Select all cells in the rows and columns which lie between two cells, including the latter	You have the following options: ▶ Click in a cell, hold down the SHIFT key and click in another cell. ▶ Click in a cell, hold down the left mouse button and drag the mouse.
Select additional cells	Click in a cell, hold down the CTRL key and click in another cell.
Undo the selection	Right-click into an empty area below or next to the list.
Selecting all cells of a column	Click the column header.
Select an additional column.	Hold down the CTRL key and click another column header.
Selecting all cells of a row	Click the row header.
Selecting additional rows	Hold down the CTRL key and click another row header.

If on the toolbar you selected Synchronization, in the Network Editor, in 2D mode, the network objects you select in the list are highlighted.

This is also the case for coupled lists, in the list on the right, for instance for the following relations:

- for **Lanes**, if the list on the left shows **Links**
- for **Pedestrian routes (static)**, if the list on the left shows **Pedestrians In Network**.

2.15.8 Sorting lists

You can sort a list by the data in one or multiple columns. This is also possible for lists that are embedded in windows.

2.15.8.1 Sorting lists according to a column

1. Click on the column header of the desired column.
2. Click on the A/Z↓ **Sort ascending** or Z/A↑ **Sort descending** icon.

The entries are sorted.

2.15.8.2 Sorting lists according to several columns

1. Click on the column header of one of the desired columns
2. Press the CTRL key and click on further column headers.

3. Click on the **Sort ascending** or **Sort descending** icon.

The list is first sorted according to the entries in the selected column which is furthest to the left. If several of these entries are identical, these rows are sorted according to the next column, etc.

2.15.9 Filtering data of a column

You can select filter criteria for the data in a column. This allows you to only display the data that meets these filter criteria. You can thus reduce the amount of data displayed in the list, for example, to the data that currently seems particularly relevant to you. This may be particularly useful when dealing with large amounts of data, as it helps create a clearer overview of the data in the list for a limited period of time. When you filter several columns in a list, each of the filters is applied.

You can deactivate filters individually. Then only the filters applied to the other columns remain active. You can also set all filters of a list to passive. Then all data is displayed again (see "List toolbar" on page 97).

Filters can also be used in the list layout. That means you can also apply filters to the lists of a network object type (see "Using named list layouts" on page 111).

2.15.9.1 Selecting filter criteria and displaying filtered data

1. Point the mouse pointer to the column header of the desired column.

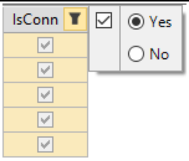
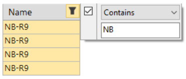
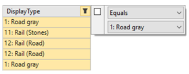
   *At the right edge of the column header, the symbol **Filter by data of this column** is displayed.*

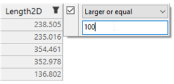
2. Click the symbol **Filter by data of this column**.

   If your list contains network objects and a network editor is open, in the network editor, the network objects are selected. In the list, all fields below the column are selected. Next to the symbol, the filter opens and displays the following elements:

- **Activate filter** check box: The check box is automatically selected when you choose or enter filter criteria.
- Filter criteria: The filter criteria displayed depend on the data type of the column. For text entered, list box data, numerical values and check boxes, suitable logical operators, options or input fields are displayed:

Data type of column	Filter criteria
Check boxes, for example in the **Links** list, in the **Is connector** column:	➤ **Yes**: Shows data for which the check box in the column of the list is selected. **Yes** is a filter criterion selected by default, even if the filter has not yet been set to active. To use **Yes** as a filter criterion and activate the filter, in the filter, select the **Activate filter** check box to the left of **Yes** check box. For example, if in the **Links** list, for the **Is connector** column, the filter criterion **Yes** remains selected and

Data type of column	**Filter criteria**
	you select the **Activate filter** check box next to it, in the **Links** list, only the connectors are shown and the links are hidden. - **No**: Shows data for which the check box has been deselected in the column of the list. For example, if in the **Links** list, for the **Is connector** column, you select the filter criterion **No**, the **Activate filter** check box next to it is automatically selected. In the **Links** list, then only links are shown and connectors are hidden.
Entered text 	The filter criterion consists of a logical operator and entered characters. 1. In list box of the filter, select the desired logical operator: - **Equal to** - **Not equal to** - **Begins with** - **Ends with** - **Contains** - **Does not contain** 2. Into the text box of the filter, enter the desired characters. *The logical operator compares the characters entered into the text box with the content in the fields of the column in the list. The list is filtered accordingly.*
List box 	The filter criterion consists of a logical operator and the object selected in the column. 1. In the upper list box of the filter, select the desired logical operator: - **Equal to** - **Contains** - **Does not contain** - **Contains text** - **Does not contain text** 2. In the bottom list box of the filter, select the desired object. *The logical operator compares the object selected in the bottom list box with the objects in the fields of the column in the list. The list is filtered accordingly.*

Data type of column	Filter criteria
Numeric value 	The filter criterion consists of the logical operator and value entered. 1. In the upper list box of the filter, select the desired logical operator: - **Equal to** - **Not equal to** - **Greater than** - **Greater than or equal to** - **Smaller than** - **Smaller than or equal to** - **Between** - **Between or equal to** - **Top n**: Maximum number of rows that are displayed - **Bottom n**: Minimum number of rows that are displayed - **Contains** - **Larger than average** - **Smaller than average** 2. Into the bottom filter box, enter the desired value. *The logical operator compares the value entered into the text box below it with the content in the fields of the column in the list. The list is filtered accordingly.*

*When you choose the filter criterion, in the filter, the **Activate filter** check box is selected.*

3. To apply the filter, ensure that in the filter, the **Activate filter** check box is selected.

   After you have chosen filter criteria that allow for a filtering of the data in the list, the filter takes immediate effect. In the list, only the data that matches the filter criterion is shown. The filter is thus active. In the column header on the right, the symbol is displayed.

2.15.9.2 Deactivating the active filter of a column

A filter is active, if in the filter, the **Activate filter** check box is selected and the symbol is displayed in the column header on the right. You can deactivate the filter to show all data in the list that was hidden based on the filter.

1. In the column of your choice, click the symbol **Filter by data of this column**.

   *To the right of the symbol, the filter opens. In the filter, the **Activate filter** check box is selected.*

2. Deselect the **Activate filter** check box.

   In the column header on the right, the symbol is hidden. In the list, all data that was hidden based on the filter of this column is shown again. Additional filters remain active and *still hide data. The previously selected filter criteria of the deactivated filter are saved. This allows you to apply the filter, with the same filter criteria you chose, to the column again later on. To do so, in the filter, select the **Activate filter** check box.*

2.15.10 Deleting data in lists

You can select and delete data in lists. Some attributes cannot be changed; these cells are displayed in gray in the lists. These attributes are deleted when the respective network object is deleted.

1. Right-click the entry, section, column header or row header of your choice (see "List sections in which you can select data" on page 100).

   The context menu opens.

2. From the shortcut menu, choose **Delete**.

Tip: Alternatively, click the symbol **Delete object(s)** to delete a selected entry (see "List toolbar" on page 97).

When you delete network objects, this might affect other network objects, e.g. if they lie on top of the deleted network objects, are assigned to them or vice versa. A message is displayed for the first network object affected.

3. When the message is shown, click the desired button:

Button	Description
Continue	Deletes network object according to the message. Shows the next message. When you select **Do this for all messages**, then all network objects are deleted for which afterwards corresponding messages are displayed. The window closes.
Skip	Does not delete network object according to the message. Shows the next message. When you select **Do this for all messages**, then no more network objects are deleted. The window closes.
Cancel	Closes the window without deleting any network objects. You can then, e.g., assign other network objects to the network objects affected, so that they no longer have objects assigned to them that you want to delete.

2.15.11 Moving column in list

1. In the column of your choice, click on the desired column header and keep the mouse button pressed.
2. Move the mouse pointer to the desired position between two column headers.

Between the column headers, a blue bar is displayed, showing the position of the moved column.

3. Release the mouse button.

2.15.12 Using named list layouts

You can adjust the columns of the list and specify a name under which the current settings are saved, so that the list layout can be used again later. You can save the named list layouts to the **.layx* file. In a list, you can choose a list layout from all named list layouts in order to show the columns in your list accordingly.

A list layout always refers to lists of a specific network object type. You cannot use a list layout for lists of other network object types.

2.15.12.1 Creating a list layout

1. Open the list of your choice.
2. Adjust the list layout:

Adjusting an element	Description
Hiding columns	Context menu **Remove Column**
Showing columns	Symbol **Attribute Selection**: Opens the window **<Name network object type>**. In the list on the left, double-click the desired attribute (see "Selecting attributes and subattributes for columns of a list" on page 112).
Sorting columns	(see "Sorting lists" on page 106)
Moving columns	Move column header horizontally between the columns you want to move (see "Moving column in list" on page 110).

3. On the List toolbar, into the **List layout selection** box, enter a unique name.
4. Confirm with ENTER.

*On the toolbar of lists of the same type, in the **List layout selection** box, the new list layout is displayed and can be selected.*

2.15.12.2 Assigning a list layout

1. Open the list of your choice.
2. In the list, in the **List layout selection** list box, click the icon .
3. Select the desired list layout.

The list layout is assigned to the list. The columns of the list are adjusted.

2.15.12.3 Saving list layouts

1. In the menu, select **File** > **Save Layout As**.
2. Enter a unique name.
3. Confirm with **Save**.

*The current layout of the entire Vissim user interface, and thus all named list layouts, are saved to the *.layx file.*

If in the User Preferences, the option **Auto-save layout when network file (inpx) is saved** is selected, the layout file is saved automatically under the name of the currently loaded network file, each time the network file is saved (see "Specifying automatic saving of the layout file *.layx" on page 154).

2.15.12.4 Reading in saved list layouts additionally

In your currently open file, you can import all list layouts that are saved to a **.layx* layout file for the individual network object types. Then you can apply these list layouts to the list of network object types in the currently opened file.

1. In the **File** menu, choose > **Read Additionally** > **Named List Layouts**.
2. Select the desired **.layx* file of your choice.
3. Click the **Open** button.

*The list layouts are imported. When you open lists of the corresponding network object types, on the toolbars of the lists, in the **List layout selection** list box, you can select the newly imported list layouts.*

2.15.12.5 Deleting a named list layout

1. On the list toolbar, right-click on the **List layout selection** list box.
2. Position the mouse pointer on the **Delete** entry in the context menu.

   The named list layouts are displayed in the context menu
3. Click the desired list layout in the context menu.
4. Confirm with **Yes**.

2.15.13 Selecting attributes and subattributes for columns of a list

You can define the content and display of each list in the attribute selection list:

- the attributes displayed in columns
- the sequence of the columns
- the alignment of the attribute value in the table field
- whether the unit is displayed
- number of decimal places if the attribute value is a numerical value with decimal

places
- formatting if the attribute value has a unit

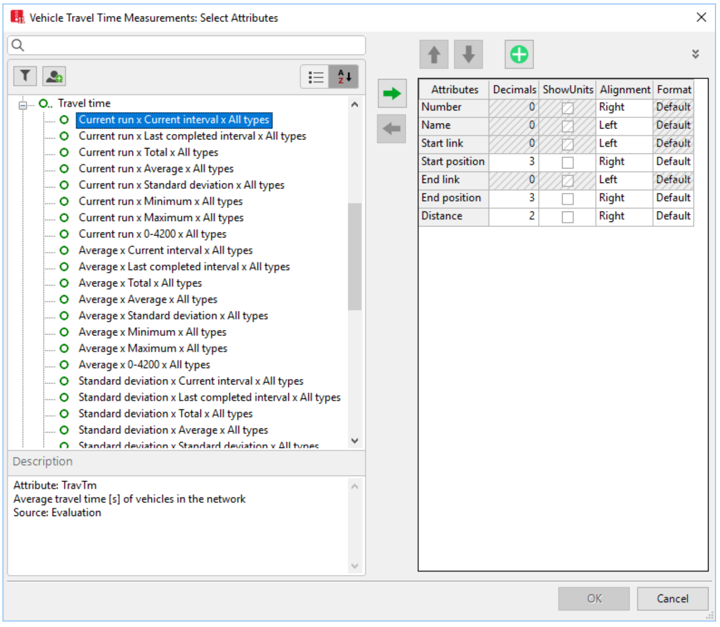
You can also define an alias for an attribute, change the name of the alias or delete the alias (see "Editing aliases in the Attribute selection list" on page 219).

2.15.13.1 Specifying an attribute or subattribute as a column

1. On the toolbar, click the **Attribute selection** button.

   *The attribute selection window **<Name Network object type>: Select Attributes** opens.*

   On the right, the program indicates the attributes and their formats for each column in the list.



   *You can edit the data in the rows of the list on the right. The changes are accepted in the columns of the list, when you click **OK** to close the **<Name Network object type>: Select Attributes** window.*

   The attributes, which you can display as columns in the list, are displayed in an Explorer pane on the left, in alphabetical order of the attributes' long names. Symbols indicate the properties of the attributes:

Symbol	Description
Disk	• Green: attribute without subattribute • Green with two ellipsis points: attribute, for which you must select at least one subattribute.
Ring	• Green: attribute, without subattributes, that only has values during a simulation run • Green with two ellipsis points: attribute, for which you must choose at least one subattribute and that only has values during a simulation run
	User-defined attribute
	User-defined: attribute, without subattributes, that only has values during a simulation run
	Alias of an attribute (see "Using aliases for attribute names" on page 217)
	Alias for an attribute that only has a value during a simulation run
→	Indirect attribute of a corresponding network object: The network object has exactly one network object belonging to it in the respective category. You can edit the indirect attribute if the target object can be edited.
- →	Indirect attribute of a corresponding network object: The network object either has or does not have a network object belonging to it in the respective category
	Indirect attribute of a corresponding network object: The network object has many corresponding network objects in the respective category. This is summarized with aggregation functions.
	Aggregation function
	Scenario comparison <Name Scenario> under attribute **Exists in scenario**: In the scenario currently open, network objects of this type have relations to a scenario selected for comparison whose network has been opened in the background (see "Comparing scenarios" on page 1148).
	Scenario comparison <Name Scenario>: Contains attributes, from which you can select subattributes of scenarios for scenario comparison and add them to the list (see "Selecting attributes for scenario comparison" on page 1149)
For 0..n relations, the following aggregation functions are provided :	
Count	Determine the number of associated network objects.
Min	Determine the minimum value of all associated network objects for the selected attribute.
Max	Determine the maximum value of all associated network objects for the selected attribute.
Sum	Determine the total of the values of all associated network objects for the selected attribute.

Symbol	Description
Average	Determine the mean of the values of all associated network objects for the selected attribute.
Histogram	Contrary to the **Concatenate** aggregate function, each occurring value is issued only once along with the frequency of its occurrence.
Concatenate	String all values of the associated network objects together for the selected attribute.
Distinct	Contrary to the **Histogram** aggregate function, each occurring value is issued only once regardless of the frequency of its occurrence.
Compare	shows a value that is the same in the related network objects.

2. If desired, filter the displayed subattributes (see " Setting a filter for selection of subattributes displayed" on page 117).
3. Repeat the following steps for all attributes that you want to show in the list:
4. In the section on the left, select the attributes of your choice:

an attribute	▸ Click the desired entry.
several individual attributes	▸ Hold down the CTRL key and click the desired entries one after another.
multiple adjacent attributes	▸ Hold down the SHIFT key and click the first and last entry.

5. Click on the icon .

   Each attribute selected on the left is listed on the right in an additional row. You cannot edit hatched cells or the attribute name. The sequence of the rows in the section on the right defines the sequence of the columns in the attribute list.

6. If required, in the section on the right, edit the properties for display of the attribute values and the sequence of the rows.
7. Confirm with **OK**.

2.15.13.2 Editing properties for displaying attribute values

Each attribute selected on the left is listed on the right in an additional row. You cannot edit hatched cells or the attribute name.

1. Click the cell of your choice.
2. Make the desired changes:

Column	Description
Attribute	Attribute name

Decimals	Number of decimal places. This is also possible with integer result attributes, which allows you to set the desired precision of the aggregated parameters.Mean and Standard deviation.
ShowUnits	☑ If this option is selected, in the list, next to the attribute value, the unit is displayed that has been selected in the **Format** column.
Alignment	vertical alignment of the attribute value in the list
Format	Unit of the attribute value

3. If you do not wish to change the **view or the arrangement of the rows in the section on the right**, confirm with **OK**.

2.15.13.3 Defining the sequence of columns in the attribute list

1. In the section on the right, select the desired row.
2. To select further rows, hold down the CTRL key and click the other rows of your choice.
3. Click on the desired icon:

Symbol	Description
⬆	Moves rows up and moves columns in list to the left
⬇	Moves rows down and moves columns in the list to the right

2.15.13.4 Functions of the symbols in the attribute selection list

Button	Description
➡	Adds the attributes selected in the Explorer on the left as rows to the section on right and as columns to the list.
⚖➡	Adds the subattributes, including those from other scenario comparisons, selected in the Explorer under ⚖ **Scenario comparison <Name Scenario>** to the section on the right as rows and to the list as columns.
✖	Deletes the rows selected on the right and the corresponding columns of these attributes from the list.
☰	**Classic sorting**: Sort attribute list: The most frequently required attributes are listed at the top of the list and indirect attributes at the end.
A..Z	**Alphanumerical order**: Sort the attribute list alphanumerically and in the following order: ➤ displays attributes that begin with a numerical value at the beginning of the list ➤ under the latter, displays direct attributes in alphabetical order ➤ under the latter, displays indirect attributes in alphabetical order ➤ Attributes at lower levels are also displayed in alphabetical order

	Insert user-defined attribute: Open the **User-defined attribute** window and define an attribute for the selected network object type (see "Creating user-defined attributes" on page 211). The user-defined attribute is displayed in the section on the left.
,	Filters (see " Setting a filter for selection of subattributes displayed" on page 117)

4. Confirm with **OK**.

The columns are adjusted in the list.

2.15.13.5 Edit aliases

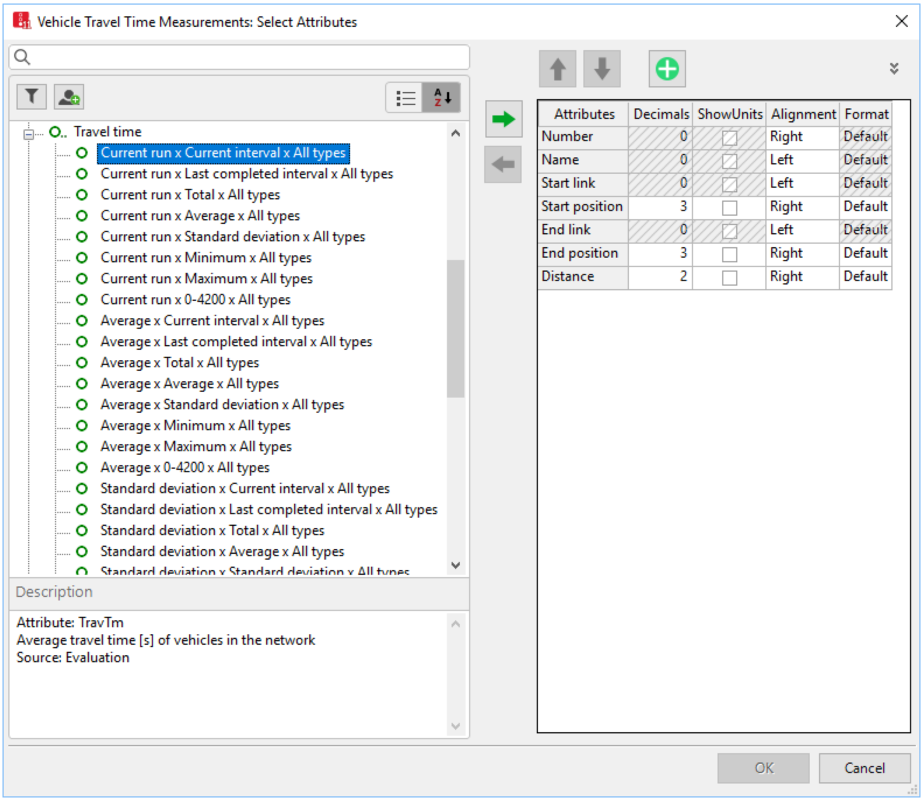
In the attribute list, you can edit aliases (see "Editing aliases in the Attribute selection list" on page 219), (see "Using aliases for attribute names" on page 217).

, ,	**Add alias**, **Edit alias**, **Delete alias** (see "Editing aliases in the Attribute selection list" on page 219)

2.15.14 Setting a filter for selection of subattributes displayed

If attributes have subattributes, the **<Name Network object type> Select attributes** window might contain many entries for selection on the left, e.g. for each time interval of the simulation runs.

2.15.14 Setting a filter for selection of subattributes displayed



If you want to transfer only certain types of subattributes to the right side, e.g. only for the current simulation run, you can filter the entries on the left by subattributes. On the left, only the filtered subattribute types are displayed. From these, you can choose the ones you want to adopt into the right side. This is particularly useful for the evaluation of result attributes.

1. In the list, click the icon **Attribute selection**.

   The Attribute selection window opens.

   On the left, all attributes are shown that you can display in columns in the list (see "Selecting attributes and subattributes for columns of a list" on page 112).

   On the right, the attributes are shown that are displayed with the current list layout.

2. Click the **No active filters** button.

   *The **Preselection Filter** window opens.*

3. Select the desired filter criteria:

Element	Description
Simulation Runs	Setting filters for the simulation runs performed: - **Current run**: show only attributes of the current simulation run - **Average**: arithmetic and thus unweighted average of all simulation runs - **Standard deviation** between all simulation runs - **Minimum** of all simulation runs - **Maximum** of all simulation runs - **Number** of each simulation run started
Time Intervals	Set filter for time intervals: - **Last completed**: Shows last time interval completed of the simulation run selected - **Total**: Sum of all time interval values - **Maximum** of all time intervals - **Minimum** of all time intervals - **Standard deviation** between all time intervals - **Average**: arithmetic and thus unweighted average of all time intervals - **x - y**: specific, individual time intervals
Vehicle Classes	Set filter for all vehicle classes or select vehicle classes. Only those vehicle classes are displayed that were selected for the collection of separate results during global configuration of the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).
Pedestrian Classes	Set filter for all pedestrian classes or select pedestrian classes. Only those pedestrian classes are displayed that were selected for the collection of separate results during global configuration of the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

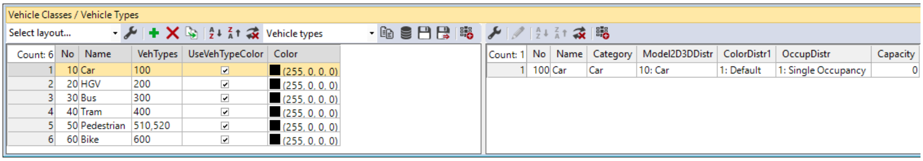
4. Confirm with **OK**.

The display and name of the ***No active filters*** *button is changed in* . ***Filtered by: <Filter criterion>*** *Point the mouse pointer to the symbol to display the selected filter criteria in a quick info.*

2.15.15 Using coupled lists

Many network object types and base data types have relations other network object types or base data types. For instance, a vehicle class refers to the vehicle types it contains. For convenient assignment and editing, Vissim automatically displays two lists next to each other:

- In the list on the left, network objects of the type from which the relation begins are displayed together with the network objects' attribute values.
- For the list on the right, you can select one of the network object types or base data types with relations to the network object type in the left list.



The left hand and right hand lists are linked to each other. If you select an object in the left list, the right list automatically only shows objects with a relation to the object you selected. If there are no such relations, only the column title is displayed. In the list, you can also select multiple or all objects. Then the right list shows all objects that refer to the objects selected in the left list.

You may edit attribute values in both coupled lists, with the exception of calculated attributes and result attributes.

In the right list, you cannot add or delete objects unless they only exist within the objects in the left list, e.g. the lanes of a link. Independent objects, to which multiple other objects might refer (e.g. vehicle types of a vehicle class), can only be added or deleted in their own list or in the Network editor. To add or delete such a reference, the relevant attribute must be shown in the left list and changed there.

Synchronization with the global selection (in network windows and the Quick view) can be activated in the lists on the left (see "List toolbar" on page 97). If on the toolbar you selected Synchronization, the network objects you select in the list on the left or right are highlighted in the Network Editor in 2D mode.

2.15.15.1 Selecting relations in coupled lists

Many network object types have relations to other network object types, such as to a link or to multiple lanes. To show the objects of such a relation in both sections next to each other, open the attribute list of the desired network object type. Then in the toolbar of the attribute list, in the **Relations** list, click the network object type whose attributes you want to display in a list on the right.

1. Open the list of the desired network object type.

   If there is a relation, on the list toolbar, the ***Relations*** *list box is displayed. By default, the entry* ***Single List*** *is shown in the* ***Relations*** *list box.*

2. Select the desired network object type in the **Relations** list box.

   On the right, the coupled list, with the network object type to which the relation refers, is opened. Only those objects are shown to which the objects selected in the left list refer.

   By default, you can edit the list *(see "Using lists" on page 93)**.*

2.15.15.2 Showing the simple list only

When a coupled list is displayed, you can close the right list and only show the list on the left:

- On the list toolbar, in the **Relations** list box, click **Single List**.

2.16 Using the Menu bar

You can call program functions via the menus in the menu bar. You can change the menu entries. You can redo the default settings.

Note: Your user preferences are saved in the Windows registry and in the **.layx* layout file when Vissim is ended. The settings are used automatically when the program is restarted.

2.16.1 Overview of menus

You can also move the menus to the user interface and change the sequence of entries (see "Editing menus" on page 133).

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

2.16.1.1 File menu

Menu commands / key combinations	Description
New	If no network has been opened and the tab on the Start page has been moved to the foreground, the tab is moved to the background and the network editor is displayed in foreground. If a network has been opened, it is closed. You can create a new network. If data was changed, a message is displayed asking whether you want to save the network file before you close the network. In any case, the user interface settings, graphic parameters of the network editors and list settings are saved in a layout file with the same name as the network file if this option is activated under User Preferences (see "Specifying automatic saving of the layout file *.layx" on page 154).
Open CTRL+O	Import network file - You can drag an **.inpx* network file from the Microsoft Windows Explorer to the user interface by drag&drop in order to open the file. - In the Windows Explorer, you can also double-click a **.inpx* network file to open it. If you open a network file containing 2D/3D model segments, with a width that differs from the current Visum 2D/3D model segments, a message is displayed. This message allows you to copy the widths of the current Visum 2D/3D model segments to all imported 2D/3D model segments.

Menu commands / key combinations	Description
Open Layout	Select and read in the **.layx* layout file, then apply it to the elements of the user interface, the graphic parameters of network editors, and the list settings (see "Saving and importing a layout of the user interface" on page 146).
Open Default Layout	Open and read in the *defaults.layx* layout file, and apply it to the elements of the user interface, the graphic parameters of network editors, and the settings in lists.
Read Additionally	- **Network**: Read network file **.inpx* additionally (see "Reading a network additionally" on page 361), (see "Importing INPX files including building data" on page 890) - **Named List Layouts** (see "Using named list layouts" on page 111) - **Named Network Editor Layouts** (see "Using named Network editor layouts" on page 86) - **Named Chart Layouts** (see "Using named chart layouts" on page 1130) - **Simulation run (.sdf file, .db file)** (see "Reading a simulation run additionally" on page 856) - **Simulation Runs (Entire Folder)** (see "Reading simulation runs additionally" on page 856)
Save CTRL+S	Save network file **.inpx* under the same path and name. In addition, the settings for the user interface, graphic parameters of the network editors, and the settings of lists are saved in a layout file of the same name as the network file (if this option is activated under User Preferences.
Save Base Network **Save Scenario** **Save Modification**	If in scenario management the base network is opened, **Save Base Network** is shown, and you can only save the base network. The same applies when you open a scenario or modifications in scenario management.
Save as	Open the **Save File As** window, and save the network file under a new file name and/or to a different directory. The **File name** box automatically shows the file name of the network file. If a different directory than the previous directory is chosen, additional files needed for the network must be copied manually to the new directory, e.g. supply files for signal control. If there is a *..\\<Name of network file>.results* directory for result data, a query opens asking whether you want to save a copy of the *..\\<Name of network file>.results* directory under the same name as the **.inpx* file. If you select **No**, the folder is not copied. This way, you can avoid saving identical result data to different directories when saving multiple versions of your network.

Menu commands / key combinations	Description
Save as Default Network	The opened network is saved as a *defaults.inpx* file to the following directory: *C:\Users\<Username>\AppData\Roaming\PTV Vision\PTV Vissim 11* If a *defaults.inpx* file has been saved to this directory, it is overwritten. The default network is loaded when Vissim is opened and no other network file is opened, e.g. when you double-click an **.inpx* file. If you delete the *defaults.inpx* file from the *..\AppData\Roaming\..* path, the next time you open Vissim, the *defaults.inpx* in your *Exe* installation directory of Vissim is used. This command is not available when a scenario management project has been opened.
Save Scenario as	When in Scenario Management a scenario is opened, you can save the scenario under a different name. The scenario saved under another name is displayed in the project explorer under **Scenarios**.
Save selection as	Open the **Save File As** window, and save the selected network objects to an **.inpx* network file under a new file name and/or to a different directory.
Save Layout as	Save the current arrangement of user interface elements, graphic parameters of network editors and the settings of lists to a **.layx* layout file in the following directory: *C:\Users\<Username>\AppData\Roaming\PTV Vision\PTV Vissim 11* If a *defaults.layx* file has been saved to this directory, it is overwritten. If you delete the *defaults.layx* file from the path displayed, the next time you open Vissim, the *defaults.layx* in your *Exe* installation directory of Vissim is used.
Save Layout as Default	Save the following settings in the default layout file *defaults.layx*: - the current arrangement of the user interface elements; - the graphic parameters of the network editor; - the current section of the background graphic and the background map, if a background graphic is loaded or a background map provider has been selected. In future Vissim will open with the saved map section. - List settings

Menu commands / key combinations	Description
	The *defaults.layx* file is normally located in the *C:\Users\<user name>\AppData\Roaming\PTV Vision\PTV Vissim 11* folder. Vissim uses these settings for the default layout, when after starting the program, you do not load a network, but create a new network file.
Compare and Transfer Networks	➤ **Create Model Transfer File**: (see "Creating model transfer files" on page 1152) ➤ **Apply Model Transfer File...**: (see "Applying model transfer files" on page 1153)
Scenario Management	Executing functions for scenario management: ➤ **Place Under Scenario Management** (see "Placing a network under scenario management" on page 1144) ➤ **Open Base Network** (see "Opening and editing the base network in the network editor" on page 1146) ➤ **Open Scenario** (see "Opening and editing scenarios in the network editor" on page 1147) ➤ **Open Modification** (see "Opening and editing modifications in the network editor" on page 1148) ➤ **Save in highlighted scenarios** (see "Project explorer toolbar" on page 1138) ➤ **Project Structure** (see "Editing the project structure" on page 1139)
Import	➤ **ANM (Vistro)/Visum**: Import ANM file (e.g. from Visum) (see "Importing ANM data" on page 366) ➤ **ANM Adaptive** (see "Adaptive import of ANM data" on page 369) ➤ Import **Synchro 7** data (see "Importing data from the add-on module Synchro 7" on page 377) ➤ **Synchro 7 Adaptive** (see "Importing Synchro 7 network adaptively" on page 379) ➤ **CAD for Pedestrian Areas**: Import CAD data that is used in pedestrian simulation to represent obstacles and walkable areas (see "Importing walkable areas and obstacles from AutoCAD" on page 882) ➤ **BIM (*.ifc)** (see " Starting conversion in Viswalk" on page 885) ➤ **openDRIVE**: Import **.xodr* file (see "Importing openDRIVE network *.xodr" on page 379)
Export	➤ Export **Visum (Nodes/Edges)** (see "Exporting data" on page 384) ➤ Export **3ds Max** data (see "Exporting static network data for

Menu commands / key combinations	Description
	3ds Max" on page 391)
Open Working Directory	Open Windows Explorer with the current working directory in which the network file *.inpx* is saved.
Show Log File	Show *vissim_msgs.txt* log file (see "Using the vissim_msgs.txt log file." on page 1181)
List of recently opened files	Open one of the recently opened *.inpx* network files. The list is updated each time you open a network file *.inpx* and save it under a new file name. The update takes place before the **File** menu is opened. *.inp* files saved as a Vissim 6 version in any previous program versions installed in parallel are not displayed.
Exit	Close Vissim. If data was changed, you are prompted whether you want to save the network file. In any case, the user interface settings, graphic parameters of the network editors and list settings are saved in a layout file with the same name as the network file if this option is activated under User Preferences (see "Saving and importing a layout of the user interface" on page 146).

2.16.1.2 Edit menu

Menu command	Description
Undo	▸  **Undo** with name of the last operation performed: discards this operation ▸ **Undo** with subordinate menu, if multiple operations can be discarded: discards all operations, including the one selected
Redo	▸  **Redo** with name of the last operation undone: performs this operation again ▸ **Redo** with subordinate menu, if multiple operations can be redone: performs all operations undone, including the one selected
Rotate Network	Enter angle around which the network is rotated counterclockwise (see "Rotating the network" on page 392)
Move Network	Enter the distances for the x-axis, y-xis or z-axis by which the network is moved (see "Moving the network" on page 393)
User Preferences	(see "Setting user preferences" on page 149)

2.16.1.3 View menu

Menu commands / key combinations	Description
Open New Network Editor	Open new Network editor. When a Network editor has been opened, a new Network editor is added as a tab.
Start page	Open start page (see "Using the Start page" on page 57)
Create Chart	Open **Create Chart** window (see "Creating a chart without preselection" on page 1123)
Network Objects	Open network objects toolbar (see "Using the Network object toolbar" on page 61)
Levels	Open level toolbar (see "Using the Level toolbar" on page 65)
Backgrounds	Open background toolbar (see "Using the background image toolbar" on page 66)
Quick View	Open Quick View (see "Using the Quick View" on page 68)
Smart Map	Open Smart Map (see "Using the Smart Map" on page 71)
Project Explorer	Open project explorer for scenario management (see "Using the project explorer" on page 1136)
Messages	Open window in which messages and warnings are displayed (see "Showing messages and warnings" on page 1178).
Simulation time	Switching the simulation time format for the status bar (see "Specifying the simulation time format for the status bar" on page 148) - **Simulation second** - Simulation **time** based on start time in the simulation parameters. Format [`hh:mm:ss,f`]
Quick Mode CTRL+Q	Activate or deactivate **Quick Mode** (see "Using the Quick Mode" on page 89)
Simple Network Display CTRL+N	Activate or deactivate **Simple Network Display** (see "Selecting simple network display" on page 88)

2.16.1.4 Lists menu

The menu commands open lists with base data, network objects or result data.

Menu command	Description
Base Data	Lists for defining or editing the base data (see "Base data for simulation" on page 202)
- **Network** - **Intersection Control** - **Private Transport** - **Public Transport** - **Pedestrian Traffic**	Lists with attributes of network objects of the selected network object type (see "Creating and editing a network" on page 334)

Menu command	Description
Graphics & Presentation	Lists for defining or editing network objects and data, which are used for the graphical preparation and realistic representation of the network as well as the creation of presentations from simulations.
Event-Based Scripts	List of event-based scripts (see "Using event based script files" on page 1172)
Measurements	Lists for defining or editing network objects or collecting simulation results
Results	Result lists with data from evaluations of simulations (see "Performing evaluations" on page 1001)

2.16.1.5 Base Data menu

Menu items open a window for basic network settings and/or open lists with basic objects for simulation (see "Base data for simulation" on page 202).

Menu command	Description
Network Settings	Basic network settings (see "Selecting network settings" on page 202)
User-Defined Attributes	List for defining or editing user-defined attributes (see "Using user-defined attributes" on page 210)
Aliases	List for defining and editing alternative attribute names (see "Using aliases for attribute names" on page 217)
2D/3D Model Segments	Axles, shafts, clutches, and doors of vehicle models (see "Attributes of 2D/3D model segments" on page 227)
2D/3D Models	2D models and 3D models for vehicles and pedestrians (see " Using 2D/3D models" on page 219)
Functions	Acceleration and deceleration behavior (see " Defining acceleration and deceleration behavior" on page 230)
Distributions	Distributions for desired speed, power, weight, time, location, distance, occupancy, 2D/3D model, colors (see "Using distributions" on page 237)
Vehicle Types	Combine vehicles with similar technical driving characteristics in vehicle types (see "Using vehicle types" on page 267)
Vehicle Classes	Combine vehicle types (see "Using vehicle classes" on page 280)
Driving Behaviors	Driving behavior parameter sets (see "Defining driving behavior parameter sets" on page 282)
Link Behavior Types	Link behavior types for links and connectors (see "Defining link behavior types for links and connectors" on page 318)
Pedestrian Types	You can combine pedestrians with similar properties into pedestrian types (see "Using pedestrian types" on page 876)

Menu command	Description
Pedestrian Classes	Group pedestrian types and combine them into pedestrian classes (see "Using pedestrian classes" on page 879)
Walking Behaviors	Walking behavior parameter sets (see " Modeling area-based walking behavior" on page 932)
Area Behavior Types	Area behavior types for areas, ramps & stairs (see "Defining area behavior types" on page 934)
Display Types	Display for links, connectors and construction elements in the network (see "Defining display types" on page 320)
Levels	Levels for multistory buildings or bridge structures for links (see "Defining levels" on page 922)
Time Intervals	Time intervals (see "Defining time intervals for a network object type" on page 326)

2.16.1.6 Traffic menu

Menu command	Description
Vehicle Compositions	Define and edit vehicle compositions (see "Modeling vehicle compositions" on page 452)
Pedestrian Compositions	Define and edit pedestrian compositions (see "Modeling pedestrian compositions" on page 930)
Pedestrian OD Matrix	Define pedestrian demand on the basis of OD relations (see "Selecting origins and destinations in the Pedestrian OD Matrix" on page 978)
Dynamic Assignment	- **Parameters**: Defining parameters for dynamic assignment (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771) - **Matrices**: Opening the **Matrices** list (see "Matrix attributes" on page 724) - **Zones**: open the list **Zones** (see "Modeling parking lots and zones" on page 698) - **OD pairs**: open and display the **OD Pairs** list, origin zones and destination zones - **Edges**: open the list **Edges** (see "Attributes of edges" on page 718) - **Paths**: Open the list **Paths** (see "Attributes of paths" on page 752) - **Read Paths**: Import the path file **.weg*(see "Attributes of paths" on page 752), (see "Setting volume for paths manually" on page 786) - **Write Paths**: Save current paths to path file **.weg* - **Create Static Routing from Assignment**: convert the current state of the dynamic assignment and thus also the paths found and their congestions into a Vissim model with inputs and static routes (see "Generating static routes from assignment" on page 791) - Calculate **PTV Visum Assignment** (see "Using an assignment from Visum for dynamic assignment" on page 793)

Menu command	Description
Toll Pricing Calculation Models	Opens the list **Toll Pricing Calculation Models / Elements** (see "Defining toll pricing calculation models" on page 331)
Managed Lanes Facilities	Opens the list **Managed Lanes Facilities / Pricing Models** (see "Defining managed lane facilities" on page 327)

2.16.1.7 Signal Control menu

Edit input data for signal control

Menu command	Description
Signal Controllers	Open the **Signal Controllers** list: Define or edit SC (see "Using signal control procedures" on page 602)
Signal Controller Communication	Opens the **SC Communication** list (see "Linking SC" on page 672)
Optimize All Fixed Time Signal Controllers	Start the green time optimization of all stage-based fixed time controllers in the network (see "Performing green time optimization of stage-based fixed time controllers" on page 609)

2.16.1.8 Simulation menu

Menu command / key	Description
Parameter	Enter simulation parameters (see " Selecting simulation method micro or meso" on page 840), (see "Defining simulation parameters" on page 840), (see "Selecting the number of simulation runs and starting simulation" on page 845)
Continuous F5	Starts continuous simulation run or switches from **Simulation single step** mode to **Simulation continuous** mode.
Single Step F6	Starts simulation in **Simulation single step** mode or switches from **Simulation continuous** mode to **Simulation single step** mode or executes the next single step.
Stop Esc	Quit simulation run
Check net-work	Check Vissim network for inconsistencies (see "Checking the network" on page 857)

2.16.1.9 Evaluation menu

Activate and parameterize evaluations (see "Performing evaluations" on page 1001)

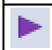
Menu command	Description
Configuration	Define settings for the evaluation of simulation runs: - **Result Management**: (see "Managing results" on page 1007) - Define output directory for evaluation files - **Result Attributes**: Configure data collection in result attributes: (see "Configuring evaluations of the result attributes for lists" on page 1014) - **Direct Output**: Configure output into files or databases (see "Configuring evaluations for direct output" on page 1018)
Database Configuration	Configure the database connection (see "Configuring the database connection for evaluations" on page 1018)
Measurement Definition	Show lists of network objects for measurements (see "Showing results of measurements" on page 1014)
Window	Configure the display of signal times table, the SC detector record and signal changes in windows (see "Showing evaluations in windows" on page 1022)
Result Lists	Open Result lists of evaluations (see "Showing result attributes in result lists" on page 1016)

2.16.1.10 Presentation menu

Create presentation (see "Creating simulation presentations" on page 1158)

Menu command	Description
Camera Positions	Open list **Camera Positions** (see "Attributes of camera positions" on page 1159)
Storyboards	Open list **Storyboards / Keyframes** (see "Using storyboards and keyframes" on page 1160)
3D Information Signs	Open the list **3D Information Signs** (see "Attributes of 3D information signs" on page 683)
Record AVIs	Only in 3D mode: Record a 3D simulation as a video file in the file format **.avi* (see "Starting AVI recording" on page 1164).
3D Anti-Aliasing	Enable or disable 3D anti-aliasing (see "Selecting the 3D mode and 3D recording settings" on page 151)
Continuous	Starts continuous animation run or switches from **Animation single step** mode to **Animation continuous** mode.
Continuous (without ANI file)	Starts a continuous animation run for the current simulation, without using an ANI file.
Single Step	Starts animation in single step mode or switches from continuous animation run to single step mode or executes the next single step.
Stop	Finish animation run
Single Step Reverse	Runs animation in reverse, single step mode.

Menu command	Description
Continuous Reverse	Run animation reverse continuously
Animation with ANI file	selected by default. Run animation with or without animation file. If the command is not selected, only the animation of the simulation in the network file currently open is run. Aggregated result attributes are used for visualization, e.g. for the color of link segments or link bars (see "Running the animation" on page 1169).

Symbol	Animation with ANI file is selected	Animation with ANI file is not selected
 **Animation continuous**	Opens a window in which you can select the **.ani* animation file of your choice. The animation then runs continuously.	When a network file is loaded, the animation of the simulation runs continuously.
 **Animation single step**	Opens a window in which you can select the **.ani* animation file of your choice. The animation then runs in single step mode.	When a network file is loaded, the animation of the simulation runs in single step mode.

While the animation is running, into the **Go to second** box, you can enter a simulation time of your choice. This triggers an update of the Network editor and the result lists to the state of the simulation, which corresponds to the specified simulation time period. Only aggregated result attributes are used for visualization.

Menu command	Description
Animation Recordings	Open list **Animation Recordings** (see "Defining an animation recording" on page 1167)
Record Animations	only in 3D mode: Switch recording on or off

2.16.1.11 Test menu

Perform testing of logic without simulation (see "Testing logics without traffic flow simulation" on page 1154)

Menu command	Description
Continuous	Starts continuous test run or switches from **Test run single step** mode to **Test run continuous** mode.
Single Step	Starts simulation in **Test run single step** mode or switches from **Test run continuous** mode to **Test run single step** mode or executes the next single step.
Stop	Quit test run
Record Macros	Enabling and disabling macro creation. Enabled: A **.m_i* macro file is saved to the working directory.
Run Macro	Select macro file **.m_i*, enter simulation second until when you want to run macro file, then start macro file
Edit Macro	Open Macro Editor and edit macro (see "Editing a macro" on page 1156)

2.16.1.12 Scripts menu

Using the scripts, you manage script files and define the times at which you want to run the script files during the simulation (see "Using event based script files" on page 1172)

Menu command	Description
Event-Based Scripts	Open list of event-based scripts
Run Script File	Execute script file (see "Starting a script file manually" on page 1173)
Stop Running Script	Stop initiated script file

2.16.1.13 Help menu

Access to information and documents about Vissim. For example, Help, information on current Vissim installation, service and contact (see "Service and support" on page 46). Register Vissim as COM server.

If during the installation of Vissim, a document was not selected for installation, the menu command is grayed out and is not available.

Various documents are only available in English. If during the installation of Vissim, you select the installation of documents for another language, these will still be installed in the *..\doc in* folder of the selected language.

Menu command	Description
PTV Vissim Help	Opens the Vissim Help located in the directory *..\exe* (see "Using the manual, Help and FAQ list" on page 46).
COM Help	Opens reference documentation of the COM interface located in the *..\exe* directory.
PTV Vissim Manual	Opens the user guide in *pdf* file format that is located in the *..\doc\<language>* directory.
Introduction to the Vissim COM API	Opens an Introduction to the VissimCOM interface *Vissim <version> - COM intro.pdf* in English located in the *..\doc\<language>* directory.
Open document directory	Opens the directory *..\Doc* in the language selected under **User Preferences** > **General** > **Language** > **main language**. All files saved to your computer during the installation process are saved to this directory.
FAQ (Online)	Show PTV Vissim FAQs on the web pages of PTV GROUP.
Service Pack Download	Show the **PTV Vissim & PTV Viswalk Service Pack Download Area** on the webpages of PTV GROUP (see "Services by the PTV GROUP" on page 47).
Technical Support	Show the support form of the Vissim Technical Hotline on the webpages of PTV GROUP (see "Service and support" on page 46).
Examples	➤ **Readme Examples**: Open the file *Overview of examples.pdf*. The file con-

Menu command	Description
	tains an overview of demo examples and training examples in the file format **.inpx* that can be installed during the installation of Vissim. By default, the file *Overview of examples.pdf* is saved to the following path: *C:\Users\Public\Public Documents\PTV Vision\PTV Vissim 11* ▸ **"First Steps" Tutorial**: Open the file *PTV Vissim - First steps.pdf*. The file contains a tutorial that provides a first insight into Vissim and practical examples of how to use it. By default, the file is saved to a path similar to the following: *C:\Users\Public\Public Documents\PTV Vision\PTV Vissim 11\Tutorial First Steps* ▸ **Open Demo Directory**: Open the Windows Explorer and the directory **Examples Demo** . The examples in the directories below illustrate typical use cases of Vissim in subprojects. Each example is explained in **.pdf* file. The **.pdf* file is always saved to the same directory as the example. ▸ **Open Training Directory**: Open the Windows Explorer and show the directory **Examples Training**. The examples in the directories below demonstrate program functions or their combination in an easy to understand context. Each example is explained in **.pdf* file. The **.pdf* file is always saved to the same directory as the example.
License	Open the **License** window (see "Showing program and license information" on page 49).
Register COM Server	Register Vissim as COM server.
About PTV Vissim	Open the **Info about PTV Vissim** window (see "Showing program and license information" on page 49).

2.16.2 Editing menus

You can move or delete menus and menu entries.

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

You can restore the default settings (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).

2.16.2.1 Showing menu entries several times

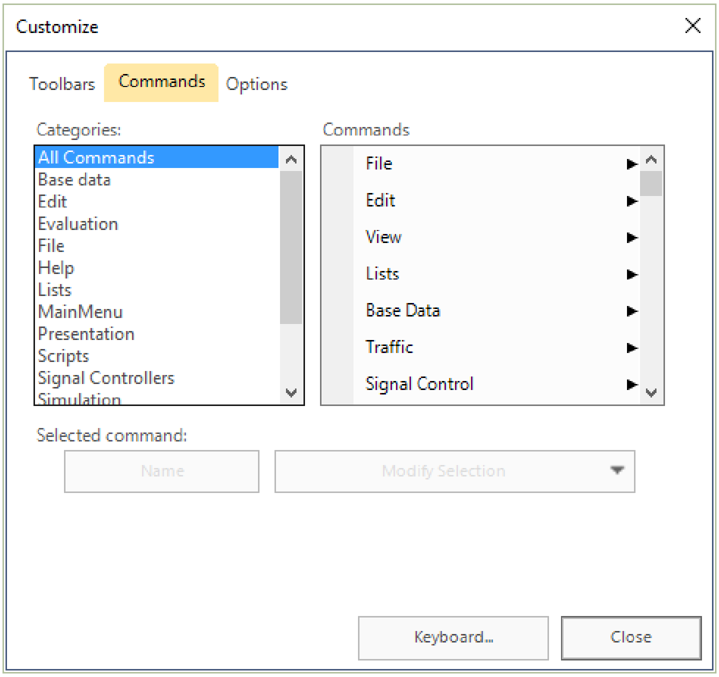
You can show menu entries in several menus or toolbars.

1. Right-click next to the menu bar.

   A context menu opens.

2. Then select **Customize**.

*The **Customize** window opens.*



3. Select the **Commands** tab.
4. Under **Categories**, select the desired category.
5. Select the desired entry in the corresponding**Commands** list box.
6. Hold down the left mouse button and drag the entry to the position of your choice in the Vissim menu.
7. Release the mouse button.

The entry is additionally displayed at the new position.

2.16.2.2 Deleting menu or menu entry

1. Right-click next to the menu bar.

   A context menu opens.

2. Then select **Customize**.

   *The **Customize** window opens.*

3. Hold down the left mouse button and drag the menu or menu entry from the user interface of Vissim.
4. Release the mouse button.

The menu or menu entry is deleted from the user interface of Vissim.

2.16.2.3 Changing display of menus

1. Right-click next to the menu bar.

   A context menu opens.

2. Then select **Customize**.

   *The **Customize** window opens.*

3. Select the **Options** tab.
4. Make the desired changes.

2.17 Using toolbars

You can call program functions via the toolbars. You can change the position of toolbars and edit toolbars. You can redo default settings of toolbars.

> Note: Your user preferences are saved in the Windows registry and in the **.layx* layout file when Vissim is ended. The settings are used automatically when the program is restarted.

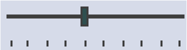
2.17.1 Overview of toolbars

By default, Vissim allows you to call functions via the following toolbars:

2.17.1.1 File toolbar

Symbol	Name	Description	Hotkeys
	**New**	If no network has been opened and the tab on the Start page has been moved to the foreground, the tab is moved to the background and the network editor is displayed in foreground. If a network has been opened, it is closed. You can create a new network. If you have changed data, you are prompted whether you want to first save the network file. In any case, the user interface settings, graphic parameters of the network editors and list settings are saved in a layout file with the same name as the network file if this option is activated under User Preferences (see "Specifying automatic saving of the layout file *.layx" on page 154).	
	**Open**	Open saved Vissim network. If in the same folder there is a layout file of the same name, it is also read in.	CTRL+O
	**Save**	Save the Vissim network. In addition, the settings for the user interface, graphic parameters of the network editors, and the settings of lists are saved in a layout file of the same name as the network file (if this option is activated under User Preferences.	CTRL+S

2.17.1.2 Edit toolbar

Symbol	Description
	➤ **Undo** with the name of the last function performed: discards this function. ➤ **Undo** with a list box if several functions can be discarded.
	➤ **Redo** with the name of the last function performed: performs this function again. ➤ **Redo** with a list box if several functions can be restored.
	Time interval after which the display of the current simulation is updated. Value range: Each 0.1 simulation second (each time step) up to the entire 100 simulation seconds (all 1,000 time steps)
	Toggle **Quick Mode** (see "Using the Quick Mode" on page 89)
	Toggle **Simple Network Display** (see "Selecting simple network display" on page 88)

2.17.1.3 Simulation toolbar

Start and stop simulation (see "Running a simulation" on page 840)

Symbol	Name	Description	Key
	**Simulation continuous**	Starts continuous simulation run or switches from **Simulation single step** mode to **Simulation continuous** mode.	F5
	**Simulation single step**	Starts simulation in **Simulation single step** mode or switches from **Simulation continuous** mode to **Simulation single step** mode or executes the next single step.	F6
	**Stop simulation**	Stop started simulation run	Esc

2.17.1.4 Run control toolbar

Symbol	Name	Description
	**Simulation break at**	Stops the simulation run at the time entered in the **Simulation break at** box. You can enter the time in the following formats: ➤ Simulation second (see "Defining simulation parameters" on page 840) ➤ Time [hh:mm:ss] beginning from start time (see "Defining simulation parameters" on page 840). Vissim displays the time in simulation seconds.

2.17.1.5 Animation toolbar

Start and stop animation (see "Running the animation" on page 1169) This toolbar is hidden in the default layout.

- To show the **Animation** toolbar, in the shortcut menu, right-click next to the toolbars (see "Adapting the toolbar" on page 138).

Symbol	Name	Description
	Animation single step reverse	Stops running animation and shows last step before the current simulation second.
	Animation continuous reverse	Stops running animation and plays animation in continuous reverse order.
	Animation continuous	- When no animation run is started: Opens a window in which you can select the **.ani* file of your choice. Then starts the continuous animation run. - When the **Animation single step** mode is started: Lets animation run continuously.
	Animation single step	- When no animation run is started: Opens a window in which you can select the **.ani* file of your choice. Then starts the animation run and shows the step of the first simulation second. - When the **Animation continuous** mode is started: Stops animation run and show next single step.
	Stop animation	Finish animation run
	Go to second list	During an animation run, jump to the entered time from simulation second NULL of the animation run. Possible input formats: - Simulation second, can be entered in unit [s] or without a unit - Point in time, can be entered with or without a unit [hh:mm:ss.f]. You can also enter minutes and/or seconds for this point in time. Enter a unit if this makes the entry unambiguous.

2.17.1.6 Test toolbar

Test signal control logic without simulated vehicles (see "Testing logics without traffic flow simulation" on page 1154). This toolbar is hidden in the default layout.

- To show the **Test** toolbar, in the context menu, right-click next to the toolbars (see "Adapting the toolbar" on page 138).

Symbol	Name	Description
	Test run continuous	Starts continuous test run or switches from **Test run single step** mode to continuous mode.
	Test run single step	Starts simulation in **Test run single step** mode or switches from Test run continuous to **Test run single step** mode or executes the next single step.
	Stop test run	Quit test run

2.17.1.7 View toolbar

If you have opened multiple windows, you can switch between them to place another window in the foreground (see "Switching between windows" on page 93).

2.17.1.8 Toolbar in Network Editors

Change network display and navigate in the network (see "Network editor toolbar" on page 75)

2.17.1.9 Toolbar in lists

Change list layout and export data (see "List toolbar" on page 97)

2.17.2 Adapting the toolbar

By default, the following toolbars are displayed below the menu bar when they are shown. You can position, show, hide and adjust these toolbars.

- File
- Edit
- Simulation
- Run control
- Animation
- Test
- View

The toolbars of network editors and lists cannot be changed.

2.17.2.1 Positioning the toolbar

You can drag the toolbar to a desired position:

- At the edges of the user interface of Vissim
- To any position in the user interface of Vissim

1. Click the left edge  of the toolbar and keep the mouse button held down.
2. Drag the toolbar to the desired position and release the mouse button.

The toolbar is shown with a title line.

2.17.2.2 Showing and hiding the toolbar

1. Click with the right mouse button on the empty area next to or below a toolbar.

   The context menu opens. The selected options indicate which toolbars are shown.

2. Select the desired options in the context menu.

2.17.2.3 Anchoring the toolbar

1. Click with the right mouse button on the empty area next to or below a toolbar.

   The context menu opens.

2. From the context menu, choose the entry **Lock the Toolbars**.

You can no longer position the toolbar.

2.17.2.4 Creating your own toolbar

1. Right-click next to the menu bar.

   A context menu opens.

2. Then select **Customize**.

   The ***Customize*** *window opens.*

3. Select the **Toolbars** tab.
4. Click the **New** button.
5. Enter the desired data.

6. Confirm with **OK**.

The toolbar is shown as empty. You can integrate menu commands (see "Showing menu entries several times" on page 133).

2.17.2.5 Deleting your own toolbar

You can delete only user-defined toolbars.

1. Right-click next to the menu bar.

   A context menu opens.

2. Then select **Customize**.

   The ***Customize*** *window opens.*

3. Select the **Toolbars** tab.
4. In the **Toolbars** section, click the entry you want to delete.
5. Click the **Delete** button.

The toolbar is deleted from the user interface of Vissim.

2.18 Mouse functions and key combinations

Many mouse functions and key combinations correspond to the default settings of your Microsoft Windows operating system in Vissim, for example for highlighting, copying or inserting.

In addition to these standards functions, you can use specific mouse functions and keyboard combinations in Vissim (see "Using key combinations" on page 141). These are mentioned in the description of the relevant functions in the Vissim Help and in the manual.

2.18.1 Using the mouse buttons, scroll wheel and DEL key

<table>
<tr><th>Key</th><th>Description</th></tr>
<tr><td>Right mouse button</td><td>You can select the function of the right mouse button (see "Right-click behavior and action after creating an object" on page 152).
• Open context menu: Opens a context menu. Which functions are shown in the context menu depends on the program element or the list you click on and whether network objects are highlighted. To insert new network objects, press CTRL and click. To insert new links, hold down the right mouse button and drag the mouse.
• Insert network object: To insert links and connectors, hold the right mouse button down and drag the mouse. To open the context menu, press the CTRL key and right-click.</td></tr>
<tr><td>Left mouse button</td><td>• Click in the Network editor to select a network object.
• Hold the mouse button down and drag the mouse to move an object in the Network editor.
• Hold down the CTRL key and the mouse button to create a copy of the object in the Network editor.
• Hold down the ALT key and mouse button to rotate an object in the Network editor. This function is only available for some network objects, for example for areas or obstacles.
• Double-click a network object in a Network editor to open a window in which you can change the attributes of the selected network object (if there is such a window).
• Double-clicking in lists:
  • Edit entry
  • If you can choose one of several attribute values, an attribute list is opened.</td></tr>
<tr><td>Middle mouse button</td><td>Drag the mouse to move the network section in the Network editor.
In the 3-D mode, press ALT to rotate the network display in the Network editor.</td></tr>
</table>

Key	Description
Scroll wheel	Rotating this changes the network display: - Rotate down: enlarge (zoom in) - Rotate up: reduce (zoom out) Hold down the scroll wheel and drag the mouse: moves the network display in the Network editor.
DEL	Deletes all network objects currently selected in the network editor. If this means that other objects must also be deleted, e.g. objects on a link, a message is displayed that you need to confirm before the network objects are deleted. Removes a selected attribute value from a list, if you are allowed to edit or delete the attribute. Removes a network object from a list, if you selected the entire row.

2.18.2 Using key combinations

In Vissim, you can press key combinations to execute certain functions directly. Depending on its function, a key combination affects a selected network object, a selected network editor, the current simulation or Vissim directly. If the key combination has a direct impact on Vissim, Vissim must be selected.

You can create and change your own key combinations for menu commands (see "Customizing key combinations" on page 144).

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

Hotkeys	Description
CTRL+A	In 2D mode: Toggle wireframe (see "Network editor toolbar" on page 75), (see "Setting up a road network or PT link network" on page 335), (see "Moving network objects in the Network Editor" on page 356)
CTRL + B	Show or hide backgrounds (see "Modeling the network for background images" on page 405)
CTRL +C CTRL+INS	Copy selected network objects to the Clipboard (see "Selecting and copying network objects" on page 340)
CTRL+D	Toggle 3D mode (see "Using 3D mode and specifying the display" on page 193)
CTRL+E	During simulation run: Toggle color of vehicle status (see "Dynamically assigning a color to vehicles during the simulation" on page 175)
CTRL+M	Measure distance (see "Measuring distances" on page 84)
CTRL+N	Activate or deactivate Simple Network Display (see "Selecting simple network display" on page 88)

Hotkeys	**Description**
CTRL+O	Open file. You can save the currently open network and load a saved network file (see "Overview of menus" on page 121).
CTRL+Q	Activate or deactivate Quick Mode (see "Using the Quick Mode" on page 89)
CTRL+R	Recalculate Spline (see "Recalculating the spline" on page 434)
CTRL+S	Save network file (see "Overview of menus" on page 121)
CTRL+T	In 3D mode: If rotate mode (3D) or flight mode (3D) is selected, switch to the other mode respectively (see "Navigating in 3D mode in the network" on page 193), (see "Flight over the network" on page 195)
CTRL+U	Switching the simulation time format for the status bar (see "Switching the simulation time format for the status bar" on page 148)
CTRL+V SHIFT+INS	Pasting network objects from the Clipboard (see "Pasting network objects from the Clipboard" on page 341)
CTRL+Y	Perform last operation undone again
CTRL+Z	Undo last operation performed
TABULATOR	Switch between overlapping network objects at the click position and select the next network object (see "Selecting a network object from superimposed network objects" on page 360)
CTRL+TABULATOR	Switch between open lists and network editors (see "Structure of lists" on page 94). Shows each of these in the foreground.
F5	- Start continuous simulation. - Switch to continuous simulation, if simulation was started in **Simulation single step** mode (see "Selecting the number of simulation runs and starting simulation" on page 845).
F6	- Start the simulation in **Simulation single step** mode. - Switch to single-step mode simulation, if continuous simulation has been started. - Perform a single step, if simulation was started in **Simulation single step** mode.
ESC	Stop simulation (see "Selecting the number of simulation runs and starting simulation" on page 845)
ENTER	Opens the window with the network object attributes, if a network object has been selected in the network editor.
SPACE BAR	- During a simulation in **Simulation single step**, execute the next step. - Switch to single-step mode simulation, if continuous simulation has been started (see "Selecting the number of simulation runs and starting simulation" on page 845).
+	If a continuous simulation has been started, increase the speed of the simulation.

Hotkeys	Description
-	If a continuous simulation has been started, reduce the speed of the simulation.
*	Maximum speed of the simulation. Alternatively, press SHIFT++.
/	During a continuous simulation: Use the same speed as the last simulation speed set. Alternatively, press SHIFT+7.
1	Execute simulation in real time, simulation speed = 1.0
POS1	Show entire network (see "Displaying the entire network" on page 83)
PAGE UP	Zoom in (see " Zooming in" on page 82)
PAGE DOWN	Zoom out (see " Zooming out" on page 82)
Arrow key	Move the observer position across the network, in the desired direction
A	In 3D mode: Move the current observer position horizontally to the left. If you additionally press the SHIFT key, the speed is increased. Alternatively for A, you can press the left arrow key.
D	In 3D mode: Move the current observer position horizontally to the right. If you additionally press the SHIFT key, the speed is increased. Alternatively for D, you can press the right arrow key.
E	In 3D mode: Move current observer position vertically downwards in terms of camera coordinates. If you additionally press the SHIFT key, the speed is increased. It can correspond to the function of S, depending on the orientation of the map.
F	In 3D mode: Move current observer position vertically downwards in terms of z coordinate of the model. If you additionally press the SHIFT key, the speed is increased. It can correspond to the function of S, depending on the orientation of the map.
I	In 3D mode: Make observer position over the network flatter
J	In 3D mode: Rotate observer position clockwise around z axis
K	In 3D mode: Make observer position over the network steeper
L	In 3D mode: Rotate observer position counter-clockwise around z axis
Q	In 3D mode: Move current observer position vertically upwards in terms of camera coordinates. If you additionally press the SHIFT key, the speed is increased. It can correspond to the function of W, depending on the orientation of the map.
R	In 3D mode only: Move current observer position vertically upwards in terms of z coordinate of the model. If you additionally press the SHIFT key, the speed is increased. It can correspond to the function of W, depending on the orientation of the map.

Hotkeys	Description
S	In 3D mode: Zoom out. Instead of S, if you press the PAGE DOWN key, it is zoomed out at larger increments.
W	In 3D mode: Zoom in Instead of W, if you press the PAGE UP key, it is zoomed in at larger increments.

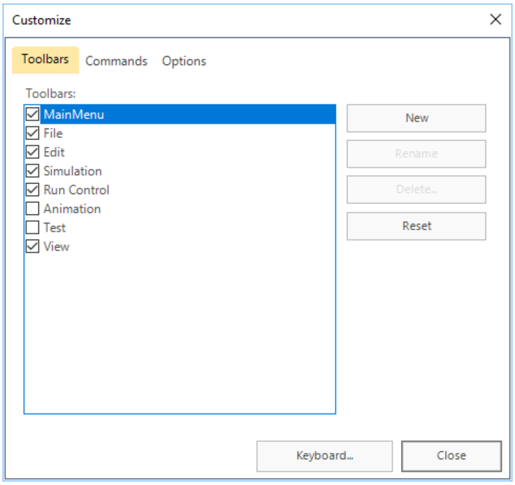
2.18.3 Customizing key combinations

You can customize the keyboard mapping by specifying keyboard shortcuts.

Note: Changing program elements or shortcut keys can make it more difficult for you to find and use commands. The documentation and the PTV Vision Support assume that the standard settings are used.

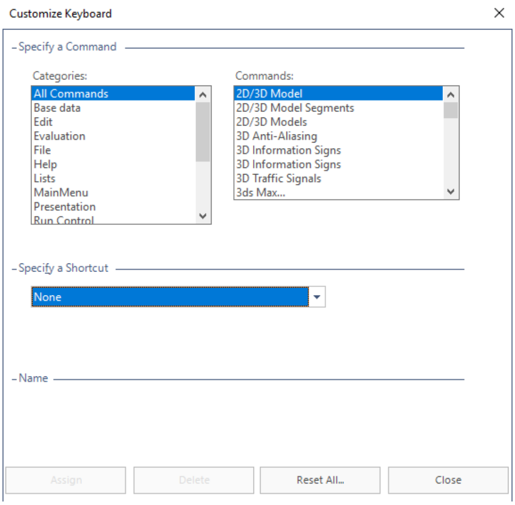
1. Right-click under the title bar on the empty area next to the menu bar and the toolbar.
2. From the shortcut menu, choose **Adapt**.

   The ***Customize*** *window opens.*



3. Click the **Keyboard** button.

   The ***Customize keyboard*** *window opens.*



4. Under **Categories**, select the desired menu.
5. Under **Commands**, select the menu command which you would like to apply for a new key combination.
6. In the **Specify a Shortcut** list, click the desired key combination.

   In the case that this key combination has already been used for a command, this command is shown in the list.
7. Click **Assign** to confirm the selection.

   If the key combination is allocated a different command, the combination is canceled.
8. Click the **Close** button.

9. Click the **Close** button.

2.18.4 Resetting menus, toolbars, shortcuts, and dialog positions

1. Choose **Edit** > **User Preferences**.

   The ***User Preferences*** *window opens.*
2. Choose the entry **GUI** > **General**.
3. Click the button of your choice:
   - **Reset Menu/Toolbar/Shortcuts**
   - **Reset dialog positions**

A message opens confirming the reset. The next time you start Vissim, the respective interface elements are displayed with their default settings and in their default positions.

2.19 Saving and importing a layout of the user interface

When you rearrange program elements, such as network editors, lists, the Quick view, the Smart Map, toolbars, context menus, graphic parameters of network editors or column settings of a list, you can save these settings in a **.layx* file. You can re-import this layout and use it for the user interface.

When you save a **.inpx* file, the current layout is automatically saved under the same name as the **.layx* file, if this option is activated in the User Preferences (see "Specifying automatic saving of the layout file *.layx" on page 154).

When you open a **.inpx* network file, the layout is automatically imported from the **.layx* file, if it has the same file name as the network file and has been saved to the same folder.

The default network file *defaults.inpx* is read in and provides predefined base objects when you start Vissim or from the **File** menu, choose **New**. The default layout file *defaults.layx* is read in when you start Vissim. Both files are saved in the following directory:

C:\Users\<Username>\AppData\Roaming\PTV Vision\PTV Vissim

If the default network file and/or the default layout file are deleted, they are copied from the *Exe* subfolder of the Vissim installation to the aforementioned directory when you start the program.

In Network editors and lists, you can generate named layouts and use them again later. All named layouts are saved in the layout file. You can also choose to only additionally read in named Network editor layouts or list layouts from the layout file.

Note: Your user preferences are automatically saved in the registry of your computer. These cannot be selected and imported in Vissim.

2.19.1 Saving the user interface layout

When you save a network in a **.inpx* file, a **.layx* file with the user interface layout is automatically saved under the same name (if this option is activated under User Preferences (see "Specifying automatic saving of the layout file *.layx" on page 154). You can also save the user interface layout to a **.layx* file under a different name, e.g. when you want to use the arrangement of program elements and/or graphic parameter settings in network editors in another network.

Note: Your user preferences are automatically saved in the registry of your computer. These cannot be selected and imported in Vissim.

You can also save several **.layx* files with different settings.

1. In the menu, select **File** > **Save Layout as**.

   *The **Save layout file as** window opens. By default the path to the opened network file is selected. The **File name** box automatically shows the name of the layout file. By default, the file type *.layx is selected.*

2. Enter a unique name.
3. Click the **Save** button.

*The layout is saved in the *.layx file. You can reload the layout (see "Importing the saved user interface layout" on page 147).*

2.19.2 Importing the saved user interface layout

You can open a saved user interface layout to organize program elements, graphic parameters of network editors, and the settings of list windows.

1. Select **File** > **Open Layout** in the menu.

   *The **Load layout file** window opens. By default the path to the opened network file is selected. By default, the file type *.layx is selected.*

2. Select the desired **.layx* file of your choice.
3. Click the **Open** button.

Tip: You can also use a Drag-and-Drop operation to drag the file from the Microsoft Windows Explorer to the user interface in order to open it.

2.20 Information in the status bar

The status bar is displayed at the bottom of the Vissim user interface interface. The status bar is divided into the following areas. The information displayed in the areas depends on whether or not a simulation is running:

Area	Description
1st section left	➤ In 2D mode: Current coordinates of the mouse pointer (global coordinates x,y in meters) ➤ In 3D mode: **fov** (**fieldOfView**) **= angle of view** (**FOV**) of camera in degrees from viewer's position.
2nd section	Only during a simulation: Current second or time of the simulation, followed by current cycle second of an SC (see "Specifying the simulation time format for the status bar" on page 148)
3rd section	Only during a simulation: ➤ *Current number of vehicles in the network + current number of pedestrians in the network*
4th section	Only during a simulation: ➤ Real time factor of simulation speed ➤ In parentheses: Number of vehicles that could be simulated close to real time (only if in the simulation parameters, the simulation speed **Maximum** is selected) ➤ if multiple simulations are running: Number of the current simulation run (total of simulation runs) in the simulation parameters.

2.20.1 Specifying the simulation time format for the status bar

You can show the simulation seconds or the time.

Showing the simulation seconds

- In the **View** menu, select > **Simulation Time** > **Simulation Second**.

Showing the time in the time format hh:mm:ss

Here the start time which is set in the simulation parameters is used (see "Defining simulation parameters" on page 840).

- From the **View** menu, choose > **Simulation Time** > **Time of Day**.

2.20.2 Switching the simulation time format for the status bar

During a simulation run, you may switch between the views Simulation Second and Time of Day.

- To do so, in the status bar, in the second box, double-click the simulation time.

Tip: Alternatively, press CTRL+U.

2.21 Selecting decimal separator via the control panel

Vissim shows data by default with the decimal separator that is set in the control panel of your operating system.

1. Choose **Start** > **Control panel**.

   The next step varies depending on your operating system.

2. Double-click on **Time, Language and Region** or **Regional and Language Options**.
3. If you use Microsoft Windows VISTA, click the **Customize** button in the **Formats** tab.
4. If you use Microsoft Windows 7 or 8, click the **Advanced Settings** button in the **Formats** tab.
5. In the **Numbers** tab, check the character in the **Decimal separator** list box.
6. You can change the settings if you wish.

3 Setting user preferences

In the user preferences, you can make the following basic settings:

- Select the language of the user interface of Vissim and the fallback language
- Select the country for the country-specific information displayed on the start page
- Restore default settings
- Set the video compression for AVI recording
- Settings for video compression and 3D mode
- Select function for the right mouse button
- Select the command you want executed after a new network object has been created.
  - Open the window with the attributes of the network object
  - Open the list with network objects of the network object type including the network objects' attributes
  - None of the two commands
- Check network at start of simulation
- Select type of detector activation in the test mode
- Specify default short name or long name for column headers
- Define the number of functions last performed that are to be saved
- Define the automatic saving of the layout file **.layx*
- Define default values for lane width, minimum gap time and minimum headway
- Collect usage data

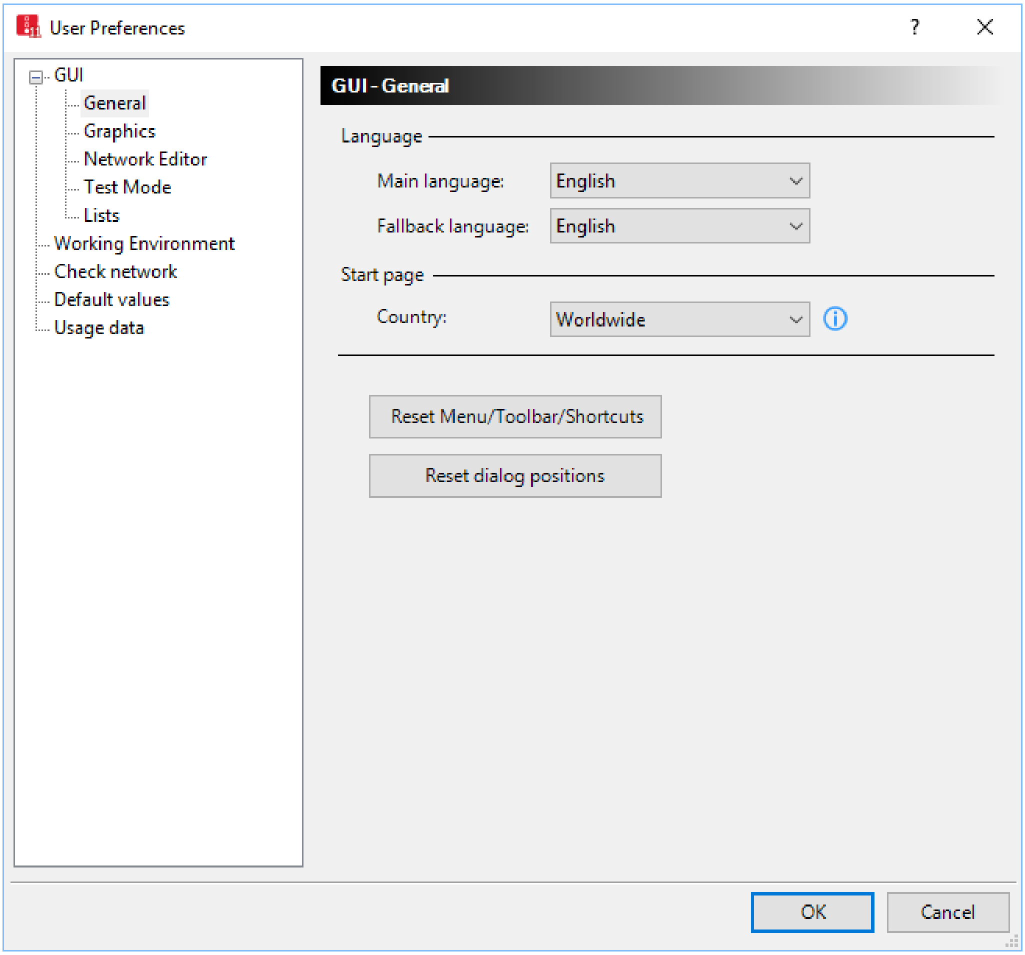
3.1 Selecting the language of the user interface

The languages available depend on the Vissim license.

1. From the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **General**.
3. In the **Main language** list, click the language of your choice.
4. In the **Fallback language** list, click the language of your choice.
5. Confirm with **OK**.

*The language setting is saved and is used in program elements and output files. The **Main language** is used again the next time Vissim is opened.*

If there is no translation available in the primary language for a text in the user interface, in the Quick info describing an attribute or in an output file, Vissim automatically shows the text in the fallback language. If there is no translation available in the fallback language, the text is displayed in English. If there is no English translation available, the text is displayed in the original language. In most cases, the original language is German.



3.2 Selecting the country for regional information on the start page

The information on the start page may vary by region (see "Using the Start page" on page 57). Some of it is available in different languages.

1. From the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **General**.
3. Go to the **Start page** and select the desired entry in the **-Country** list box.
4. Confirm with **OK**.

3.3 Selecting a compression program

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **Graphics**.
3. Make the desired changes:

Element	Description
AVI recording	**Compression** button: Open the **Video compression** window, and select a default compression program for AVI recording (see "Starting AVI recording" on page 1164).

4. Confirm with **OK**.

3.4 Selecting the 3D mode and 3D recording settings

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **Graphics**.
3. Make the desired changes:

Section	Description
3D Mode	☑ **Enable 3D mode**: Select this option to show the 3D mode. After you restart Vissim, the following icons are displayed in the toolbars of the Network editors: - **2D/3D** (see "Network editor toolbar" on page 75) - **Edit 3D graphic parameters** (see "Editing 3D graphic parameters" on page 194) - **Anti-aliasing**: ☑ Select this option to reduce so-called “jaggies”, i.e. pixel edges caused by screen resolution. Using this type of recording produces a video of higher quality, however slows down the recording speed. - **Double-sided lighting**: ☑ Select this option to improve 3D display. You can switch off this option to increase simulation speed in the 3D mode, particularly when using Nvidia-GeForce graphic cards from series 400 (Fermi) on. This, however, can lead to change in the color some vehicle types are displayed in. - **Background texture compression**: ☑ Select this option to compress textures for background images. It might then take longer to load background images. Select this option if your computer does not have sufficient video memory.
	Graphics driver: Graphics driver required for displaying objects during simulation. Ensure that your video card and the driver installed support the standard selected. Settings changes only become effective after you restart Vissim. - **OpenGL 3.0**: Default setting - **DirectX 11**: Allows access to Vissim in 3D mode via remote desktop

4. Confirm with **OK**.

The changes take effect after Vissim is restarted

3.5 Right-click behavior and action after creating an object

In the Network editor, you can define the right-click behavior. This means you can specify whether a window or list for editing the attributes is automatically opened after you insert a network object, or whether neither the list nor the window shall be opened.

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **Network Editor**.
3. Make the desired changes:

Element	Description
Right click behavior	Function, when you right-click in the network editor: • **Right-click opens the context menu**: The context menu of the Network editor opens (see "Network editor context menu" on page 80) To create new objects, press CTRL + right-click. • **Right-click creates a new object**: In the Network editor, a new network object of the type selected in the Network objects toolbar is inserted. To open the context menu, press CTRL + right-click.
Automatic action after object creation	After you have added a new network object in the Network Editor, have opened the window or list for editing the attributes, or have neither opened the window nor the list. For many network object types, you can not only edit the attributes in a list, but also in a window. For some network object types, there is no window but only the list, e.g. for vehicle inputs, pedestrian inputs, routng decisions / routes, and conflict areas. • **Show edit dialog if available, show list otherwise** (default setting): If there is a window for editing the attributes of this network object type, open it, otherwise open the list of network objects of this type. • **Show edit dialog if available, no action otherwise**: If there is a window for editing the attributes of this network object type, open it, otherwise do not open a window or list. • **Always show list**: Always open the list of network objects of this type • **No action**: Do not open the list or window. The network object is inserted into the Network Editor at the desired position.

4. Confirm with **OK**.

3.6 Showing and hiding object information in the Network editor

In the Network editor, in the Quick info, you can show Network editor attributes and attribute values directly next to the network object. Depending on whether you insert the network object, edit it or are point the mouse pointer towards it, different attributes and attribute values are displayed.

You can show or hide the Quick info.

1. From the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **Network Editor**.
3. Select or deselect **Show Quick info in Network editor**.
4. Confirm with **OK**.

3.7 Configuring command history

You can set that Vissim saves the recently executed functions as command history. You can also define the number of the previously executed functions which should be saved.

From the **Edit** menu, choose the **Undo** icon to undo the maximum number of functions. To redo them, click the **Redo** icon (see "Overview of menus" on page 121).

1. In the **Edit** menu, choose > **User Preferences**.
2. From the navigation tree, choose **Working Environment**.
3. Make the desired changes:

Element	Description
Command history (Undo / Redo) active	☑ If this option is selected, the previously executed functions are saved. You can use the **Undo** icon.
Maximum number of entries in command history	Number of functions last performed that were saved and shall be shown in the **Edit** > **Undo** menu. Standard: 20 Maximum number: 49

4. Confirm with **OK**.

3.8 Specifying automatic saving of the layout file **.layx*

1. In the **Edit** menu, choose > **User Preferences**.
2. From the navigation tree, choose **Working Environment**.
3. In the **Auto-save layout** section, make the settings of your choice.

Element	Description
when network file (inpx) is saved	☑ Select this option to automatically save the layout file **.layx* when the network file **.inpx* is saved.
when network is discarded (e.g. File - New)	☑ Select this option to automatically save the layout file **.layx* when you close the current network.

4. Confirm with **OK**.

3.9 Defining click behavior for the activation of detectors in test mode

You can set whether you want to activate detectors in test mode with a single or a double click.

1. Choose **Edit** > **User Preferences**.

*The **User Preferences** window opens.*

2. Choose the entry **GUI** > **Test Mode**.
3. Make the desired changes:
   - **Activate detector on double click**: In the test mode, double-clicking switches the detector call. A single click selects the detector and you can, for instance, look at the attributes of the detector in the Quick View.
   - **Activate detector on single click (selection unavailable)**: Single click changes the detector call. You cannot select detectors (see "Setting detector types interactively during a test run" on page 1154).
4. Confirm with **OK**.

3.10 Checking and selecting the network with simulation start

When you start the simulation, Vissim automatically checks the Vissim network for certain constraints that could prevent the simulation from starting. You can select whether you also want Vissim to check the Vissim network for consistency when the simulation starts.

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **Check network**.
3. Make the desired changes:

Element	Description
Check network at start of simulation	☑ If this option is selected, Vissim checks the Vissim network for certain constraints and for consistency when the simulation is started. ☐ If this option is not selected, Vissim only checks the Vissim network for certain constraints that could prevent the simulation from starting.

3.11 Resetting menus, toolbars, shortcuts, and dialog positions

1. Choose **Edit** > **User Preferences**.

   *The **User Preferences** window opens.*

2. Choose the entry **GUI** > **General**.
3. Click the button of your choice:
   - **Reset Menu/Toolbar/Shortcuts**
   - **Reset dialog positions**

A message opens confirming the reset. The next time you start Vissim, the respective interface elements are displayed with their default settings and in their default positions.

3.12 Showing short or long names of attributes in column headers

In the column header of attribute lists, you can show the short or long name of an attribute.

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigator tree, choose **GUI** > **Lists**.
3. Select the desired option:
   - **Short name**
   - **Long name**

3.13 Defining default values

You can define default values that are assigned to links and priority rules when you add them to your Vissim network.

1. From the **Edit** menu, choose > **User Preferences**.
2. In the navigation tree, select **Default values**.
3. Make the desired changes:

Element	Description
Add link	**Use user-defined lane width**: ☑ Select this option to enter the desired value into the **Lane width** box. New links, with the lane width specified, are added in the Network editor. Default 3.50 m
Add priority rule	• **Use user-defined minimum gap time**: ☑ Select this option to enter the desired value into the **Min. gap time** box. New priority rules, with the width specified, are added in the Network editor. Default 3.0 s • **Use user-defined minimum headway**: ☑ Select this option to enter the desired value into the **Min. headway** box. New priority rules, with the width specified, are added in the Network editor. Default 5.0 m

4. Confirm with **OK**.

3.14 Allowing the collection of usage data

As your PTV Vision team, we want Vissim to provide the functions you actually need and that make work easier for you. To support us in this objective, activate the **Collect usage data** option. We can then examine which parts of the program you prefer and how often you use individual functions, e.g. via the number of clicks on menu items, the size of networks, the time the Network editor is used or the duration of simulation runs. All data collected and sent to our servers is fully anonymized. The data cannot be traced back to you or your workplace.

1. In the **Edit** menu, choose > **User Preferences**.
2. In the navigation tree, choose **Usage data**.
3. Make the desired changes:

Element	Description
Collect usage data	Collect and transfer usage data.
Log usage data locally	Only available if the option **Collect usage data** has been selected. Opens the *Telemetrydata_<yyyymmdd>_<hhmmss>.log* file in the editor that is installed as default on your computer. This file has been saved to the *%TEMP%\VISSIM* folder. Files from the previous day are deleted when starting Vissim.

4 Using 2D mode and 3D mode

In the 2D mode, you can add, show, and edit network objects as well as run vehicle or pedestrian simulations.

The 3D mode is used to show networks during a simulation or presentation in 3D.

4.1 Calling the 2D mode from the 3D mode

- On the Network editor toolbar, click the **2D/3D** button .

The button changes to . The Vissim network and vehicles are shown in 2D.

Tip: You can save your display options to and load them from the **.layx* file.

4.2 Selecting display options

You can use Graphic Parameters to define the display of network objects in Network editors for each network object type. In addition, you can select base graphic parameters for the display of the network for each open Network Editor.

With 3D Graphic Parameters, you define the three-dimensional display of the network for each open Network Editor.

For vehicles, pedestrians, links, areas, ramps and stairs, you can specify a colored display, e.g. a dynamic display based on simulation data.

Simulations can also be run in the 3D mode (see "Using 3D mode and specifying the display" on page 193).

Per default, they are shown in 2D.

4.2.1 Editing graphic parameters for network objects

You can set graphic parameters for the network objects of a network object type. Via graphic parameters, you define the display of network objects in the Network editor last used.

You can also change graphic parameters during the simulation.

In the network object list, a preview button is displayed for the graphic parameter of each network object type. The colors of the preview buttons depend on the graphic parameters chosen.

Note: Graphic parameters can refer to attributes of network objects.

---

1. If several network editors are open, ensure that the network editor in which you want to display the network objects with the modified graphic parameters is active.
2. On the Network Objects toolbar, in the row of the network object type of your choice, click the desired **Edit graphic parameters** button.

Icon	Network object type	Preview icon Graphic Parameters (in default colors)
	Links	
	Desired Speed Decisions	
	Reduced Speed Areas	
	Conflict Areas	
	Priority Rules	
	Stop Signs	
	Signal Heads	
	Detectors	
	Vehicle Inputs	
	Vehicle routes, in the list box :	
	Vehicle Routes (Static)	
	Vehicle Routes (Partial)	
	Vehicle Routes (Partial PT)	
	Vehicle Routes (Parking Lot)	
	Vehicle Routes (Dynamic)	
	Vehicle Routes (Closure)	
	Vehicle Routes (Managed Lanes)	
	Vehicle attribute decisions	
	Parking Lots	
	Public transport stops	
	Public Transport Lines	
	Nodes	

Icon	Network object type	Preview icon Graphic Parameters (in default colors)
	Data Collection Points	
	Vehicle Travel Times	
	Queue Counters	
	Flow bundles	
	Sections	
	Background Images	No graphic parameters
	Pavement Markings	
	3D Traffic Signals	
	Static 3D Models	
	3D Information Signs	
	Vehicles In Network	
	Pedestrians In Network	
	Areas: - Polygon - Rectangle	
	Obstacles, in the list box : - Polygon - Rectangle	
	Ramps & Stairs	
	Elevators	
	Pedestrian Inputs	
	Pedestrian routes, in the list box - Static - Partial	

Icon	Network object type	Preview icon Graphic Parameters (in default colors)
	Pedestrian Attribute Decisions	
	Pedestrian Travel Times	

The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

Tip: Alternatively you can also open the graphic parameters of a network object type via the context menu.

1. On the network object toolbar, right click the network object type of your choice.
2. From the context menu, choose **Edit graphic parameters**.

1. Make the desired changes.
2. If you want to change colors, double-click the row of the desired graphic parameter.
3. Click next to the list when you want to close the list of the graphic parameters of a network object type.

The list closes. The graphic parameters are assigned to the network objects of the network object type.

4.2.2 List of graphic parameters for network objects

You can set graphic parameters for a type of network object for the network editor last used (see "Editing graphic parameters for network objects" on page 158). Depending on the network object type, in addition to general graphic parameters, this may also be network object specific parameters.

If network objects are composed of multiple components, you can assign separate colors to some of them. For example, you can select the color for the mast and the signal arm of each 3D signal head.

You cannot define graphic parameters for backgrounds

When you display network objects in the network editor, the display type settings have priority over the graphic parameter settings for network objects (see "Defining display types" on page 320).

4.2.2.1 General graphic parameters

You can define the following general graphic parameters for most network object types:

Graphic parameter	Description
Object visibility	☑ If this option is selected, the network objects of this network object type are displayed in a network editor

<table>
<tr><th>Graphic parameter</th><th>Description</th></tr>
<tr><td></td><td>If an object of the network object type is selected, the attribute cannot be deactivated.
☐ If this option is not selected, the non-selected network objects of this type are not displayed in the Network editor when a different network object type is selected.
☐ If this option is not selected, but Label visibility and a label attribute have selected.
In the Network editor, only the labels of network objects of this network object type are displayed.
In the Network editor, the network objects of this network object type are not displayed.
The graphic parameters preview symbol is highlighted in a lighter color.</td></tr>
<tr><td>Label visibility</td><td>☑ Select this option to show a 2D label for each network object of this type in the network editor.
By default, the label is displayed in the center of the network object. In the Insert mode, use the mouse to move the position of the respective network object type.</td></tr>
<tr><td>Label color</td><td>Color of label</td></tr>
<tr><td>Label attribute</td><td>Opens the window <Name of network object type: Select Attribute>. Click the attribute whose value you want to show as a label.</td></tr>
<tr><td>Label decimal places</td><td>Number of decimal places in label</td></tr>
<tr><td>Label font size</td><td>Font size of label in points (pt)</td></tr>
<tr><td>Use label color scheme</td><td>Labeling of network objects using a color scheme based on attribute values. Next to the margin on the right, click the symbol to open the Edit Label Color Scheme window:
Classification by color: Attribute by whose values the color is classified
Color for undefined value: Label color when the attribute lies outside the upper and lower limits
: Select a pre-defined color scheme that contains class bounds and colors. The color schemes vary in color and class bound.
Range scale factor:
Enter a factor for the upper bound and lower bound
Apply button: Multiply values of upper bounds and lower bounds by a factor</td></tr>
</table>

Graphic parameter	Description
Label background color	Color of the area in which the label is displayed. The default color for vehicles in the network: transparent gray. The default color for other network objects: transparent
Legend contains <parameter>	The legend contains a description of the symbols and colors of the network object types, for which under Graphic parameters, you selected **Legend contains <parameter>**.

4.2.2.2 Network object specific graphic parameters

Graphic parameters for filling and outlines

The following network object types have graphic parameters for filling and outlines:

- Reduced Speed Areas
- Detectors
- Parking Lots
- Public transport stops
- Public Transport Lines
- Static 3D Models
- Pedestrian Inputs
- Links
- Connectors in the graphic parameters of the network object type **Links**
- Nodes and node segments
- Sections
- Elevators

Graphic parameter	Description
- **Fill style** - **Connector fill style** - **Segment fill style**	- **Solid fill**: Display fill color or color scheme color in the network object - **No fill**: Do not display fill color or color scheme color in the network object. Select this option together with the border line style **No line** to hide the network object, irrespective of the attribute **Object visibility**.
- **Fill color** - **Connector fill color** - **Segment fill color** - **PT line fill color**	Color within outline area of network object or color of line. The graphic parameter **Use display type** must not be selected.

<table>
<tr><th>Graphic parameter</th><th>Description</th></tr>
<tr><td></td><td>Using the fill style Solid fill and a fill color with the Alpha value set to 0 in the color definition, the network objects of the network object type are transparent in 2D and 3D. The transparent option for network object types should only be used where it is absolutely necessary to create transparency. This option reduces the rendering speed and thus the display and visualization quality of large, transparent objects.
Avoid using “Transparency” for a completely transparent floor, for instance.
Use Transparency for windows or individual transparent wall segments, for example. When designing balustrades for escalators, a transparent design creates a modern look. Additionally, pedestrians are more visible. The window panes of vehicle models created as ScetchUp files can be transparent.</td></tr>
<tr><td>Border line style
Connector border line style
Segment border line style</td><td>Object border style:
Solid line
No line</td></tr>
<tr><td>Border color
Connector border color
Segment border color</td><td>Color of object border</td></tr>
</table>

Graphic parameters for line color

The network objects of the following network object types are displayed as a colored line that run via a link or lane, or they contain lines. You define the line color via the graphic parameter **Line color**:

- Desired Speed Decisions
- Stop Signs
- Signal Heads
- Vehicle Inputs
- Queue Counters
- Flow Bundles
- Data Collection Points
- Pavement Markings

Graphic parameters for Start line color, End line color, route course and public transport stops.

The beginning and end of network objects of the following network object types are displayed as a colored line that runs across a link or lane:

- Priority Rules
- Vehicle Travel Times
- Public Transport Lines
- Vehicle routes: For each type of vehicle route you may also specify the color of the route course and of the public transport stops:
  - **Static route color**
  - **Partial route color**
  - **Partial PT route color**
  - **Route closure color**
  - **Parking lot route color**
  - **Managed Lane general route color**
  - **Managed lane route color**
  - **Active stop color**
  - **Inactive stop color**

Graphic parameter	Description
Start line color	Color of section at the beginning of network object
End line color	Color of section at the end of network object

Graphic parameters for headways of priority rules

Graphic parameters	Description
Headway visibility	☑ If this option and the destination section of a priority rule are selected, the headway of the priority rule is displayed in the 2D mode as a triangle across the lane width. The triangle is pointing downstream. The number of lanes is considered for which priority rules have been defined.
Headway color	Color of triangles in which headways are displayed, if the attribute **Headway visibility** has been selected.

Graphic parameters for display of signal heads in 3D mode

Graphic parameter	Description
Signal head display mode 3D	- **as blocks** - **as stop lines**: Display as stop lines on links for vehicles and on links for which the attribute **Is pedestrian area** (option **Use as pedestrian area**) is selected. - **as stop lines (vehicle links only)**: Display as stop lines on links for vehicles. Stop lines are not shown in links for which the attribute **Is pedestrian area** (option **Use as pedestrian area**) is selected.

Graphic parameters for display of 3D signal heads

You may specify the colors used in a 2D schematic diagram of the components of a 3D signal head:

Graphic parameter	Description
Signal arm color 2D	- Color of line that represents the arm - Outline color of circle at the end of the arm
Signal head color 2D	Outline color of circle that represents the signal head
Mast color 2D	Outline color of diamond that represents the mast
Streetlight color 2D	- Color of line that represents the street light arm - Outline color of rectangle at the end of the arm
Color 2D	Outline color of 3D signal head

Graphic parameters for wireframe display of links and connectors

Graphic parameter	Description
Wireframe color	Color of links in wireframe display
Connector wireframe color	Color of connectors in wireframe display

Graphic parameters for display of link bars and lanes

Dynamic attributes can only be shown as link bars, if in the graphic parameter **Link bar configuration**, for the attribute **Link bar representation** > **Segment-based** is selected.

Graphic parameter	Description
Link bar display type	- **No link bars**: Links are displayed without link bars. - **Only link bars**: Links bars are shown instead of links. - **Links and link bars**: Links are displayed together with link bars.
Link bar configuration	Opens the **Edit Link Bar** window:
	Link bar representation::

Graphic parameter	Description
	• **Segment-based**: Link bars for individual segments of the link • **Link-based**: Show link bars for entire link • **Base color**: Color of link bar, default RGB 190, 0, 0
	Classification by width: **Attribute** : The width of the link bars is based on the values of the selected attribute and the following settings. If the value is negative, the width is based on the absolute value. **Width scale**: • **Automatic**: Vissim specifies the width. • **Manually**: For the width, you can enter a minimum value **Scale (minimum):** and a maximum value **Scale (Maximum)**. If the absolute value of the attribute is smaller than the specified minimum value, the link bar is not drawn. If the absolute value of the attribute is greater than the specified maximum value, the link bar is drawn with its maximum width. • **Scale bar width (maximum)**: maximum width for link bars with automatic width • **Lateral offset**: Distance between links and link bars
	Classification by color: **Attribute** : The color of the link bars is based on the values of the selected attribute and the following settings: **Class bounds and colors**: • **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. Based on value range after MIN, on upper bound of the row above. • **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. **Range scale factor**: • Enter a factor for the upper bound and lower bound • **Apply** button: Multiply values of upper bounds and lower bounds by a factor

Graphic parameter	Description
	: Select a pre-defined color scheme that contains class bounds and colors. The color schemes vary in color and class bound.
Color of pavement markings	Color of pavement markings between the lanes of links with multiple lanes
Width of lane markings	Width (in meters) of lane markings between the lanes of links with multiple lanes, 0 = no markings. Lane markings are no longer displayed when you zoom far out of the Vissim network.

Graphic parameters for drawing mode of network objects

The colored display of network objects of the following network object types can be based on graphic parameters that specify a permanent color, display type or color scheme. The colored display of network objects can also be based on simulation data. Define the display via the graphic parameter **Drawing mode**.

- Links, nodes, sections, areas, obstacles, ramps & stairways:

Graphic parameter	Description
Use consistent color	Display network objects of the network object type in the color of the graphic parameter **Fill style**.
Use display type	Show network objects with display type assigned to network object (see "Defining display types" on page 320)

- Vehicles in network, pedestrians in network, links, nodes, areas, ramps and stairs:

Graphic parameter	Description
Color scheme configuration	Opens the **Edit Color Scheme** window: Select classification based on attribute values of an attribute and display network objects in this color scheme (see "Assigning a color to vehicles based on an attribute" on page 177), (see "Assigning a color to pedestrians based on an attribute" on page 178), (see "Assigning a color to links based on aggregated parameters" on page 179), (see "Assigning a color to nodes based on an attribute" on page 191), (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190)

- The display of **Vehicles In Network** and **Pedestrians In Network** can be based on a color distribution. The desired color distribution must be assigned to the desired vehicle type or pedestrian type.

Graphic parameter	Description
Color by color distribution	During simulation, vehicles and/or pedestrians are displayed in color according to the color distribution assigned to your vehicle type or pedestrian type.

- The display of **Vehicles In Network** during simulation may be based on the state of the vehicles.

Graphic parameter	Description
Color by driving state	During simulation vehicles are displayed in a color that represents their state. During a simulation run you may switch between this view and the default view (see "Dynamically assigning a color to vehicles during the simulation" on page 175). You cannot edit the colors.

Graphic parameters for the display of vehicle and pedestrian shapes

Graphic parameter	Description
Shape (2D)	- **Rectangle**: Display vehicles and/or pedestrians as rectangles in 2D mode. Simulations might run faster than possible with the graphic parameter **Rounded**. - **Rounded**: Display vehicles and/or pedestrians as rounded corners in 2D mode. Simulations might run slower than possible with the graphic parameter **Rectangle**.

Graphic parameters for the alignment of vehicle labels

Graphic parameter	Description
Auto-rotate label	The graphics parameter **Label visibility** must be selected. ☑ If the option is selected, the label is automatically rotated so that its base line is always aligned along the vehicle. You cannot move the label.

Graphic parameters for the display of areas and nodes based on their function

Graphic parameter	Description
Color by function	Only for **Nodes** (see "Meaning of node color and line style" on page 713): ☑ Select this option to visualize nodes based on their attribute values **Use for evaluation**, **Use for mesoscopic simulation** and **Use for dynamic assignment**: - If for the **Fill style** attribute, **Solid fill** is selected, nodes are filled with the same color as the border.

<table>
<tr><th>Graphic parameter</th><th>Description</th></tr>
<tr><td></td><td>
If no node is selected, the border color depends on the node type selected:
<table>
<tr><th>Color and style of the border</th><th>node type selected</th></tr>
<tr><td>solid white</td><td>no node type</td></tr>
<tr><td>solid green</td><td>Use for evaluation</td></tr>
<tr><td>solid red</td><td>Use for dynamic assignment</td></tr>
<tr><td>solid black</td><td>Use for evaluation and for dynamic assignment</td></tr>
<tr><td>white, dashed</td><td>Use for mesoscopic simulation</td></tr>
<tr><td>not white, dashed</td><td>Use for mesoscopic and other simulations</td></tr>
</table>
</td></tr>
<tr><td></td><td>
only for areas: This option is selected by default; the displayed area fill color depends only on the area function:
<table>
<tr><th>Color</th><th>Description</th></tr>
<tr><td>Magenta</td><td>Platform edge is selected for the Public transport usage attribute of the area.</td></tr>
<tr><td>Blue</td><td>Waiting area is selected for the Public transport usage attribute of the area.</td></tr>
<tr><td>Green</td><td>A pedestrian input is located in the area.</td></tr>
<tr><td>Red</td><td>A destination section of a pedestrian route is located in the area.</td></tr>
<tr><td>Gray</td><td>All other areas</td></tr>
</table>
If an area has several functions, the order of the colors in the table determines the fill color priority for displaying the area.
In wireframe display the colors are used for the edges of the areas.
</td></tr>
</table>

Graphic parameter for turn value visualization

Visualize turn values graphically in the Network editor using result attributes of a node along turn relations (see "Visualizing turn values" on page 685), (see "Configuring turn value visualization" on page 687).

Graphic parameters for visualizing the volume on paths with flow bundles

Graphic parameters	Description
Show flow bundle bars	If a flow bundle or at least a filter cross section is selected, the volume on the respective paths is displayed using flow bundle bars (see "Visualizing volumes on paths as flow bundles" on page 766), (see "Show flow bundle bars" on page 770). Next to the margin on the right, click the symbol to open the **Edit Flow bundle bars** window: The **classification by width** is based on the flow bundle volume. **Width scale**: - **Automatic**: Vissim specifies the width. - **Manually**: For the width, you can enter a minimum value **Scale (minimum):** and a maximum value **Scale (Maximum)**. If the absolute value of the attribute is smaller than the specified minimum value, the flow bundle bar is not drawn. If the absolute value of the attribute is greater than the specified maximum value, the flow bundle bar is drawn with its maximum width. - **Scale bar width (maximum)**: maximum width for flow bundle bars with automatic width - **Lateral offset**: Distance between links and flow bundle bars

4.2.2.3 Graphic parameters with hatched cells

Hatched cells contain irrelevant attribute values that due to other attribute values have no effect. If, for example, the **Label visibility** is deactivated, the graphic parameters **Label color**, **Label attribute**, **Label decimals** and **Label font size** are shown hatched.

4.2.3 Editing base graphic parameters for a network editor

1. On the toolbar of the Network editor, click the button **Edit basic graphic parameters**.

   The list of graphic parameters opens (see "List of base graphic parameters for network editors" on page 171).

2. Select the desired entries.
3. If you want to change colors, double-click the row of the desired graphic parameter.
4. Click next to the list when you want to close it.

The list closes. The graphic parameters are assigned to the Network Editor.

4.2.4 List of base graphic parameters for network editors

You can set the following graphic parameters independently for every Network editor. You can also change graphic parameters during the simulation.

Long name	Description
Background color	Background color in 2D mode
Show map	Display or hide background map in 2D mode. The desired map must be selected in the base graphic parameter **Map provider**.
Map provider	Select a background map from a background map provider
Map gray scale	Display background map in gray tones or color
Map intensity	Background map color intensity: 0 = no color, 100 = maximum color intensity
Wireframe mode	Toggle wireframe
Show scale	Show or hide scale at the bottom left of the Network editor
Show grid	Show 20 m grid as help to position network objects. If you zoom out the network substantially, the grid is no longer shown.
Showing the simulation time label	In the Network Editor, the current simulation time is displayed in simulation seconds or as the time [hh.mm.ss]. The simulation time is also recorded during the recording of **.avi* files. - Simulation has not been started: 0.00 simulation seconds or the time 00:00:00 - Simulation in **Continuous** mode: Simulation time continues to run - Simulation in **Single step** mode: The simulation time displayed is the time of the current single step. If you continue in the Single step mode, the simulation second displayed will only continue to run after the time intervals specified in the simulation parameters under simulation resolution have ended (see "Defining simulation parameters" on page 840).
Simulation time label position	Display position of simulation time in the Network Editor
Simulation time offset	Relative position of simulation time to the label position in the Network Editor - First value: x position, default 15 - Second value: y position, default 30
Simulation time label font color	Font color of simulation time
Simulation time label font size	Font size in pt of simulation time
Show logo	Display the logo at the **Logo position**. The logo graphic file is selected in the **Logo filename** box.
Logo position	Position of logo in the network editor
Logo offset	- First value: Number of pixels by which the logo is moved towards the x-axis. - Second value: Number of pixels by which the logo is moved towards the y-axis.

<table>
<tr><th>Long name</th><th>Description</th></tr>
<tr><td>Logo file name</td><td>Name of the logo graphic file Click the symbol to select the file. Vissim supports the following file formats for logos:
<table>
<tr><th colspan="2">Raster formats</th></tr>
<tr><td>*.bmp</td><td>Windows bitmap, two-dimensional</td></tr>
<tr><td>*.jpg</td><td>graphic compressed according to JPEG (Joint Photographic Experts Group) standard</td></tr>
<tr><td>*.gif</td><td>Graphics Interchange Format, compressed with no loss in quality</td></tr>
<tr><td>*.tif</td><td>Tagged Image File Format, uncompressed or packbits compressed</td></tr>
<tr><td>*.png</td><td>Portable Network Graphics, compressed with no loss in quality</td></tr>
</table></td></tr>
<tr><td>Automatic Level Transparency</td><td>☑ Select this option to draw network objects on underlying layers with a decreasing level of transparency.
☐ If this option is not selected, the network objects on all layers are displayed the same way.</td></tr>
<tr><td>3D mode</td><td>Toggle 3D mode</td></tr>
<tr><td>Selection color</td><td>Color of selected network objects</td></tr>
<tr><td>Rubberband color</td><td>Color of rectangle dragged open with the mouse for network object selection</td></tr>
<tr><td>Show compass</td><td>The compass rose to show the cardinal direction in the Network editor Red tip = North</td></tr>
<tr><td>Compass position</td><td>Display position of compass rose in the Network Editor You may turn the compass rose.</td></tr>
<tr><td>Compass offset</td><td>Relative position of compass rose in the Network Editor
➤ First value: x position, default 0
➤ Second value: y position, default 0</td></tr>
<tr><td>Show legend</td><td>☑ Select this option to display the legend at the legend position. The legend contains a description of the symbols and colors of the network object types, for which under Graphic parameters, you selected Legend contains <parameter>.</td></tr>
<tr><td>Legend offset</td><td>➤ First value: Number of pixels by which the legend is moved towards the x-axis. Default 10
➤ Second value: Number of pixels by which the legend is moved towards the y-axis. Default 10</td></tr>
<tr><td>Legend position</td><td>Position in Network editor Default value Bottom right</td></tr>
<tr><td>Legend scale</td><td>Factor for enlarged or reduced display of the legend. Value range 0.01 to 100. Default 1.0</td></tr>
</table>

4.2.5 Using textures

Textures are graphic files which you can select instead of the default colors for the display of the following elements in 3D mode:

- Sky: a hemisphere is automatically displayed above the Vissim network. The texture is projected on it.

> Note: A texture is displayed for the sky only if the graphics card of the computer has at least a 16-bit Z-buffer. Otherwise, the insufficient depth resolution causes display problems.

- Land: The texture is displayed on the land area in which the Vissim network is located. If the texture is less than the land area, it is displayed in a tiled format.
- Links, connectors, areas, ramps and stairways: The texture is displayed in a tiled format on these network objects.

You can select textures for network objects in the display types (see "Defining display types" on page 320).

You can select textures for the sky and landscape via the 3D graphic parameters (see "Editing 3D graphic parameters" on page 194).

4.2.6 Defining colors for vehicles and pedestrians

Vissim can visualize individual vehicles and/or pedestrians as well as classified parameters for links and areas. Visualization depends on the following graphic parameters and attributes:

- **Links**: **Links** list > **Show individual vehicles** attribute or **Link** window > **Display** tab > **Individual vehicles** option (see "Attributes of links" on page 409).
- **Areas**: **Areas** list > **Show individual pedestrians** attribute or **Pedestrian Area** window > **Visualization** section > **Individual pedestrians** option (see "Attributes of areas" on page 898).
- **Ramps & Stairs**: **Ramps & Stairs** list > **Show individual pedestrians** attribute or **Ramps & Stairs** > **Display** tab > **Individual pedestrians** (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913)
- Graphic parameter **Use color scheme** for links, vehicles in network, pedestrians in network, areas, ramps & stairs (see "List of graphic parameters for network objects" on page 161).

You have the following options to visualize individual vehicles and pedestrians in color:

- via colors and color distributions of classes and types of the vehicles, pedestrians or public transport lines (see "Static colors of vehicles and pedestrians" on page 175)
- for vehicles, color based on dynamic simulation data (see "Dynamically assigning a color to vehicles during the simulation" on page 175)

- color based on attributes (see "Assigning a color to vehicles based on an attribute" on page 177), (see "Assigning a color to pedestrians based on an attribute" on page 178)

The status of signal heads is displayed at their position by a colored bar in addition to the vehicles, for example red, red-amber, green.

You can assign display types to links and construction elements (see "Defining display types" on page 320).

You can visualize links, areas, ramps, and stairs based on aggregated parameters (LOS) (see "Assigning a color to links based on aggregated parameters" on page 179), (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190).

4.2.6.1 Static colors of vehicles and pedestrians

Static colors are used for vehicles or pedestrians if neither the automatic nor the user-defined dynamic coloring is active. Static colors are defined via the attributes of the vehicle type, the pedestrian type, the vehicle class or the pedestrian class. For PT vehicles, the color is taken from the respective PT line.

The table indicates which color is used, if you have defined colors for multiple objects:

Defined by type	Defined by class	Defined by bus line/tram line	Display color based on
Color distribution	-	-	Vehicle type, pedestrian type
Color distribution	Color	-	Vehicle class, pedestrian class
Color distribution	-	Color	PT line
Color distribution	Color	Color	Vehicle class, pedestrian class

The color of a PT line has priority over the color distribution of a vehicle type and the color of a vehicle class or pedestrian class has priority over the color distribution of a vehicle type and a PT line.

If a vehicle or pedestrian type belongs to multiple classes, vehicles or pedestrians of each type are displayed in the color of the first class of these classes which has a defined color.

At the same time, you can also display the link segments in the simulation with colors based on vehicular parameters (see "Assigning a color to links based on aggregated parameters" on page 179).

4.2.6.2 Dynamically assigning a color to vehicles during the simulation

You can automatically assign vehicles a color based on their current status during the simulation. Thus you obtain information about the movements of a vehicle in the simulation.

If the 3D models used do not only contain polygons with dedicated colors, automatic dynamic coloring also works in 3D mode.

At the same time, you can also display the link segments in the simulation with colors based on vehicular parameters (see "Assigning a color to links based on aggregated parameters" on page 179).

1. Press CTRL+E in the Network Editor during the simulation.

   *The graphic parameter **Color by driving state** is selected. The following colors are used for vehicles and are listed in descending priority. You cannot change these colors:*

Color	Description
White	**In queue**: The vehicle is stuck in a traffic jam. At least one queue counter must be defined in the network.
Light blue	**Waiting for lane change**: Vehicle has been waiting more than 6s at the last position for lane change (emergency stop distance).
Teal	**Ignores priority rule**: Vehicle ignores priority rule to resolve a deadlock situation.
Black	**In priority rule deadlock situation**: Vehicle is part of a deadlock situation caused by priority rules
Orange	**Wants to change lanes**: Due to the vehicle route, a lane change becomes necessary that the vehicle has not yet begun.
Light green	**Is changing lanes**: Vehicle is changing lanes.
Dark green	**After lane change**: Lane change was performed in the last 6 s.
Dark red	**Brakes to change lanes**: Vehicle brakes, as it need to change lanes.
Amber	**Brakes cooperatively**: Cooperative braking for an upcoming lane change maneuver of a vehicle, from an adjacent lane to its own lane.
Red	**Sleep**: Vehicle is currently not paying attention.
Dark yellow	**Ignores signal**: Vehicle has decided in the last 3 seconds of simulation to traverse a red signal head or a blocked section of a priority rule because its speed was too high to come to a stop in advance.
Pink	**Brakes heavily**: Vehicle brakes heavily (< -3.0 m/s²).
Purple	**Brakes moderately**: Vehicle brakes moderately (-3.0 m/s² to -1.0 m/s²).
Navy blue	**Default**: All other states (default)

2. Press CTRL+E again, to deactivate the graphic parameter **Color by vehicle state**. The vehicle color is then no longer assigned based on simulation data.

4.2.6.3 Assigning a color to vehicles based on an attribute

Note: A vehicle color based on an attribute is only displayed if the colors are not assigned automatically during the simulation (see "Dynamically assigning a color to vehicles during the simulation" on page 175).

You may also display link segments in the simulation in colors based on traffic-related parameters (see "Assigning a color to links based on aggregated parameters" on page 179).

1. On the Network objects toolbar, next to **Vehicles In Network**, click the **Edit graphic parameters** button.

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

2. In the **Drawing mode** list box, click **Use color scheme**.
3. In the list of graphic parameters, click **Color scheme configuration**.

   *The window **Edit Color Scheme for Vehicles In Network** opens.*

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.

4. Make the desired changes:

Symbol	Element	Description
	Attribute	Opens an attribute selection window. The result attributes can be filtered (see " Setting a filter for selection of sub-attributes displayed" on page 117).
	Predefined color scheme	Open **Select pre-defined color scheme** list box: Select a defined color scheme and show it in the **Class bounds and colors** list. The color schemes vary in color and class bound. • **Red-yellow-green**: 11 classes, class size by default 0.500, 11 colors • **Acceleration**: 11 classes, 11 colors from pink to red, yel-low, green to MAX = white • **Speed**: 11 classes, 11 colors from pink to red, yellow, green to MAX = white

Element	Description
Class bounds and colors list	Edit color scheme. From the shortcut menu, choose **Add** to add a new row to the list and define additional class bounds and colors. - **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. Based on value range after **MIN**, on upper bound of the row above. - **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

5. If you want to change the RGB values in the **Color** column, double-click the desired row.
6. Select the desired colors.
7. Confirm with **OK**.

4.2.6.4 Assigning a color to pedestrians based on an attribute

In the 2D mode, pedestrians are displayed as colored oval shapes. You may assign individual pedestrians a color based on attribute values.

Areas in which pedestrians move can be displayed based on LOS schemes (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182).

1. On the Network objects toolbar, next to **Pedestrians In Network**, click the **Edit graphic parameters** button .

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

2. Then select **Color scheme configuration**.

   *The window **Edit Color Scheme for Pedestrians In Network** opens.*

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.

3. Make the desired changes:

Symbol	Element	Description
	**Attribute**	Opens an attribute selection window. The result attributes can be filtered (see " Setting a filter for selection of sub-attributes displayed" on page 117).
	**Predefined color scheme**	**Select pre-defined color scheme** list box: Select a defined color scheme and show it in the **Class bounds and colors** list. The color schemes vary in color and class bound. - **Red-yellow-green**: 11 classes, class size by default 0.500, 11 colors - **Speed**: 11 classes, 11 colors from pink to red, yellow, green to MAX = white

Element	Description
Class bounds and colors list	Edit color scheme. From the shortcut menu, choose **Add** to add a new row to the list and define additional class bounds and colors. - **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. Based on value range after **MIN**, on upper bound of the row above. - **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

4. If you want to change the RGB values in the **Color** column, double-click the desired row.
5. Select the desired colors.
6. Confirm with **OK**.

4.2.7 Assigning a color to links based on aggregated parameters

During a simulation, you can display traffic-related parameters classified by color based on link segments. For these traffic-related parameters, you can select result attributes of the current simulation, e.g. volume, density, emissions or delay time (relative). For visualization of the data, you can define colors or select color schemes that have been predefined, e.g. for volume, density, emissions or delay time (relative). Thus you can easily differentiate the vehicular parameters on the individual links during the simulation in the network, for example, to quickly locate a hotspot in a large network.

You can make similar settings for pedestrians in areas (see "Using LOS schemes for showing aggregated pedestrian values" on page 186).

At the same time, you can also display the vehicles in the simulation in color based on their attributes or simulation data (see "Assigning a color to vehicles based on an attribute" on page 177), (see "Dynamically assigning a color to vehicles during the simulation" on page 175).

1. Before starting the simulation, from the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab > **Links** and make the following settings:
2. Select **Collect data**.
3. Into the **From time** and **To time** boxes, enter the simulation seconds during which you want to collect data for a classified display.
4. Into the **Interval** box, enter the length of the time intervals for which you to aggregate data.
5. Click the **More** button.
6. Select how you want the data to be collected:
   - **per lane segment**: separately for each lanes
   - **per link segment**: for all lanes together
7. In the network objects toolbar, next to **Links**, click the **Edit graphic parameters** button .

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).
8. In the **Drawing mode** list box, click **Use color scheme**.
9. In the list of graphic parameters, click **Color scheme configuration**.

   *The **Edit Color Scheme for Links** window opens.*

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.

   *Under **Classification based on**, **Lanes** or **Lane segments** is selected. This depends on the option **per lane segment** or **per link segment** of the configuration of the evaluation for links (see "Showing data from links in lists" on page 1103).*
10. In the **Classification based on** area, select **Segments**. Lanes have static attributes only.
11. Make the desired changes:

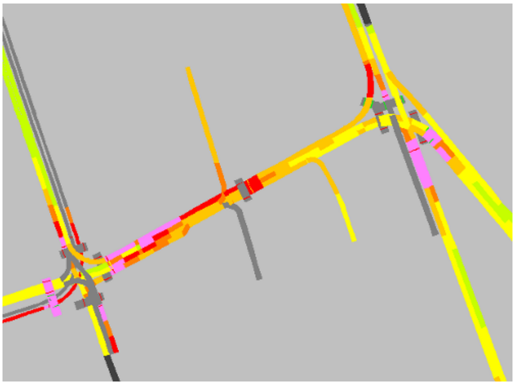
Symbol	Element	Description
	**Attribute**	Opens an attribute selection window. The result attributes can be filtered (see " Setting a filter for selection of subattributes displayed" on page 117).
	**Predefined color scheme**	**Select pre-defined color scheme** list box: Select a defined color scheme and show it in the list below. The color schemes vary in color and class bound: - **Red-yellow-green**: 11 classes, class size by default 0.500, 11 colors - **Speed**: 11 classes, 11 colors from pink to red, yellow, green to MAX = white - **Density**: 11 classes, 11 colors from white to light blue, blue to MAX = white - **Volume**: 11 classes, 11 colors from white to light blue, blue to MAX = white - **Relative delay**: 11 classes, 11 colors from white to light blue, blue to MAX = white

Element	Description
List	- **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. From **MIN**, based on upper bound of the row above. - **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

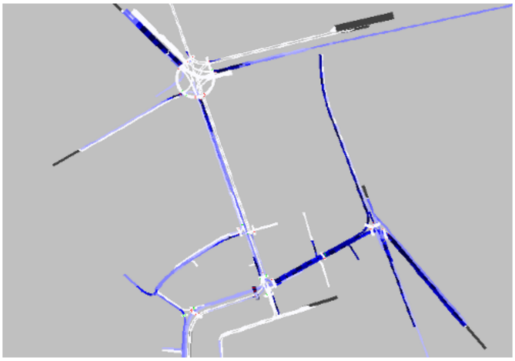
12. If you want to change the RGB values in the **Color** column, double-click the desired row.
13. Confirm with **OK**.

4.2.7.1 Examples of colored links based on aggregated parameters

Speed attribute:



Density attribute:



4.2.8 Assigning a color to areas based on aggregated parameters (LOS)

During a simulation, you can show a grid-based or area-based display of pedestrian attribute values according to different LOS schemes (see "Using LOS schemes for showing aggregated pedestrian values" on page 186). During simulation, this allows you to easily distinguish between different areas in the network, e.g. in order to identify relevant differences in the pedestrian density of a an area or different pedestrian speeds. You can set the display to 2D or 3D mode.

- When you choose **Areas** as a basis for classification, you can also show result attributes during the simulation, namely the parameters **Speed** and **Density** and define classes for them.
- When as a basis for classification, you select **Pedestrian Grid Cell Data**, a grid is placed over the network that is used as a basis to record its density, density of worst interval, expected density, expected density of worst interval, speed and speed of worst pedestrian interval. The parameter values are determined by cell and are visualized through the cell color. For technical reasons, the 2D and 3D display might slightly differ where there is a transition from areas to ramps and stairways.

Note: To record these data, you need to configure the recording in the evaluation of areas and ramps (see "Evaluating pedestrian density and speed based on areas" on page 1034). Otherwise, areas are displayed in their static colors during the simulation.

You can, for example, show the average density in color during the last evaluation interval of the simulation. To do so, select the attribute **Density** / **Current run** x **Last completed**. Afterwards, during the simulation run, you can watch how the colors change depending on the evaluation interval.

If the data of the attribute set is not yet available, because the time interval selected has not begun yset, each area or cell is displayed in its static color. The latter is also used when no color scheme has been activated.

In 2D mode, areas, ramps, and stairs defined for separate levels are displayed in different transparent shades, depending on the height of the level they are on. This means the colors can no longer be directly compared in the LOS display. For a comparison, open a Network editor for each level whose objects you want to compare, and in it show the respective level only. The objects are then shown in opaque colors and can be compared in different Network editors.

In 3D mode, areas, ramps, and stairs on different levels are not displayed in "transparent" in a Network. If you want to hide individual areas, ramps or stairs for comparison, in the attributes of these objects, select a display type with the attribute **invisible** (see "Defining display types" on page 320). Go to the graphic Parameters of the network object type, if you want to select the transparent type of display for all network objects of a 2D and 3D network object type: Fill style **Solid fill** and fill color with the Alpha value set to 0 in the color definition (see "List of graphic parameters for network objects" on page 161).

You can make similar settings for links (see "Assigning a color to links based on aggregated parameters" on page 179).

You can simultaneously show the pedestrians in color. This is based on the attributes of the pedestrians (see "Assigning a color to pedestrians based on an attribute" on page 178).

4.2.8.1 Parameter-based area visualization - grid-based or area-based

You can choose between grid-based or area based for area visualization. You then configure and activate visualization in the graphic parameters of the network object type **Areas**.

Option 1: Parameter-based area visualization - area-based

1. Before starting a simulation, make the following settings:
2. From the **Evaluation** menu, choose **Configuration** > **Result Attributes** tab > **Areas & ramps**.
3. Select **Collect data**.
4. Into the **From time** and **To time** boxes, enter the simulation seconds during which you want to collect data for a classified display.
5. Into the **Interval** box, enter the length of the time intervals for which you to aggregate data.

6. Confirm with **OK**.

Option 2: Parameter-based area visualization - grid-based

1. Before starting a simulation, make the following settings:
2. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
3. In the **Pedestrian Grid Cells** row, select **Collect data**.
4. If desired, change the time and/or the interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
5. Click the **More** button.

   *The **Pedestrian Grid Cells** window opens.*
6. Make the desired changes (see "Grid-based evaluation of pedestrian density and speed" on page 1037).
7. Confirm with **OK**.

8. Confirm with **OK**.

4.2.8.2 Activating parameter-based area visualization

1. On the Network object toolbar, next to **Areas**, click the **Edit graphic parameters** button .

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).
2. In the **Drawing mode** list box, click **Use color scheme**.
3. In the list of graphic parameters, click **Color scheme configuration**.

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.
4. For an area-based classification, in the **Classification based on** section, select **Areas**.
5. For a grid-based classification, in the **Classification based on** section, select **Pedestrian Grid Cell Data**.
6. Make the desired changes:

Symbol	Name	Description
	**Attribute**	Opens a window that allows you to select an attribute or subattribute. - Density - Density - duration exceeding threshold - Density (worst interval) - Level - Density experienced - Experienced density - duration exceeding threshold - Experienced density (worst interval) - Speed - Speed - duration exceeding threshold - Speed (worst interval) - Speed variance - Velocity variance - duration exceeding threshold - Ramp/Stairs If in the evaluation configuration of pedestrian-grid cells, **Only last interval** is selected, this enables the attributes **Density (worst interval)**, **Experienced density (worst interval)** and **Speed (worst interval)** to display the corresponding results aggregated. The result attributes can be filtered (see " Setting a filter for selection of subattributes displayed" on page 117).
	**Predefined color scheme**	Open the **Select pre-defined color scheme** list box (see "Using LOS schemes for showing aggregated pedestrian values" on page 186): Show a defined color scheme in the **Class bounds and colors:** list. The color schemes vary in color and class bound.

Name	Description
Class bounds and colors list	Edit color scheme. From the shortcut menu, choose **Add** to add a new row to the list and define additional class bounds and colors. - **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. From **MIN**, based on upper bound of the row above. - **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

7. If you want to change the RGB values in the **Color** column, double-click the desired row.
8. Select the desired colors.
9. Confirm with **OK**.

4.2.8.3 Using LOS schemes for showing aggregated pedestrian values

Alternatively, or in addition to displaying individual pedestrians in areas, on stairways, ramps, moving sidewalks, and escalators, you can choose the LOS scheme to show aggregated values in color, according to a color code. Using the LOS scheme is a quick and easy way to gain an overview. You may also select a pre-defined red-yellow-green scheme.

Note: Areas that have reached or exceeded a capacity limit are displayed in red in all predefined LOS schemes. This allows you to easily identify them.

You can choose to classify all areas and ramps by color, or only a user-defined square grid. You can also choose the class boundaries for each color and the attribute, whose value determines the class.

Predefined LOS schemes

Both subsequent tables contain predefined LOS schemes, which you can select in the **Edit color scheme** window, in the graphic parameters for areas, ramps & stairs (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190). Under **Classification by color**, you select a suitable attribute. When a predefined LOS-scheme is selected, its predefined colors and class bounds are displayed in the **Class bounds and colors** list. You can edit the colors and class bounds.

Default schemes used in literature

Scheme	Pedestrian movements	Stairway	Waiting situations	Attribute
Fruin	☑	☑	☑	Density [Ped/m²]
There is a different scheme for each pedestrian movement, stairs and waiting situation. According to Fruin the breakpoints that determine the various levels-of-service have been determined on the basis of the walking speed, pedestrian spacing, and the probabilities of conflict at various traffic concentrations. Numerically, these breakpoints are specified as density or flow. By defining both density and flow limits, Fruin provides the traffic planner with the right strategy, as the level-of-service concept is meant to assess walking quality up to a capacity limit. As soon as this limit is exceeded, the capacity is neglected.				
Weidmann	☑			Density [Ped/m²]
Weidmann follows Pushkarev and Zupan and the HCM in stating eight criteria for assessment of pedestrian walkway quality. Using eight further references, he describes the level limits in words. Weidmann does not explain how his description in words is converted into numerical limits.				
HBS	☑		☑	Density [Ped/m²]
There is a different scheme for each pedestrian movement and waiting situation. These level limits are similar to those of HCM (e.g. rounded values of metrical HCM data). The importance of considering the effective width (or area) is pointed out. In addition, a factor is given for calculation of the effective density with contraflows. Vissim calculates the level of service based on the geometric area and does not account for contraflows.				
HCM	☑		☑	Density [Ped/m²]
There is a different scheme for each pedestrian movement and waiting situation. HCM refers to Fruin as originator of the LOS concept, but the breakpoints between levels are set at considerably smaller values.				
Pushkarev-Zupan	☑			Density [Ped/m²]
Pushkarev and Zupan, along with Fruin, are credited in the HCM for having initiated and done the principal work in developing the LOS concept.				
Polus	☑			Density [Ped/m²]
Pre-defined LOS scheme with five levels Breakpoint values are based on measurements recorded in Haifa.				
Tanaboriboon-Guyana	☑			Density [Ped/m²]

Scheme	Pedestrian movements	Stairway	Waiting situations	Attribute
The breakpoint values for this six-level scheme are based on measurements recorded in Bangkok. So this is the only typically Asian scheme. It is the only LOS scheme with all breakpoint values higher than the ones of the walkway LOS of Fruin.				
Teknomo	☑			Speed [km/h]
In contrast to density-based LOS, this speed-based LOS scheme uses the opposite sequence (starting with the worst LOS), as with increasing speed the LOS becomes better.				

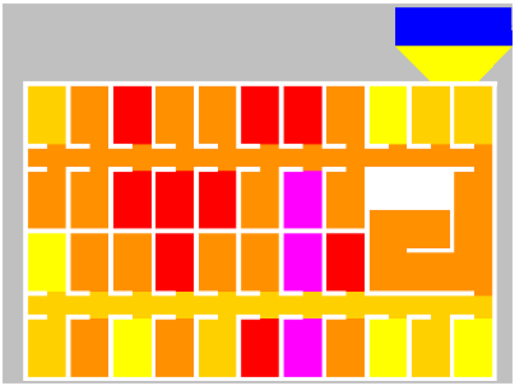
Schemes for user-defined classification

Scheme	Attribute
Density	Density in pedestrians/m²
Speed	Speed in km/h

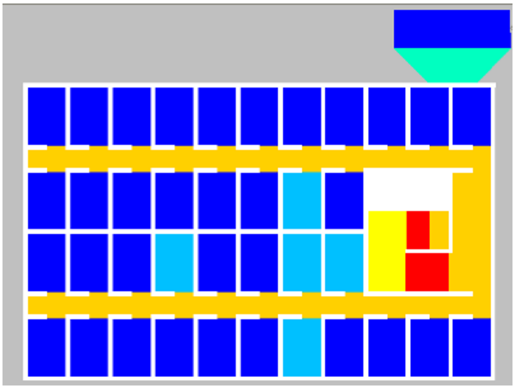
4.2.8.4 Examples of visualization of pedestrian parameters

Typical area-based visualization of essential parameters - examples

Example 1: Parameter **Speed**, with default settings:

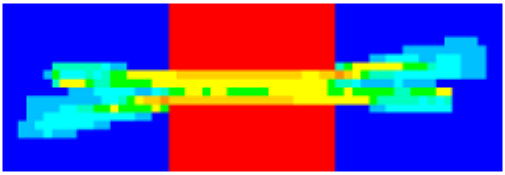


Example 2: Parameter **Density**, with default settings:



Typical grid-based visualization of essential parameters - examples

The parameters are depicted with the help of the simulation in the example file *..\Examples Training\Pedestrians\Counterflow\Counterflow-2Rooms - Simple.inpx*.

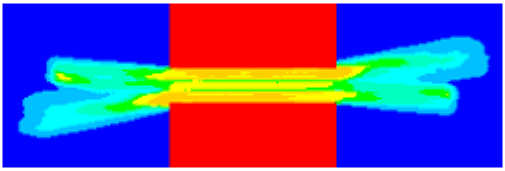


The image above is the result of the following configuration:

Parameter	Value
Scheme	LOS: User-defined: Density
Cell size	1.0 m
Range of influence	1

The image below is the result of the following configuration:

Parameter	Value
Scheme	LOS: User-defined: Density
Cell size	0.2 m
Range of influence	5



4.2.9 Assigning a color to ramps and stairs based on aggregated parameters (LOS)

During a simulation, you can visualize and classify traffic-related parameters of pedestrians on ramps and stairs based on different LOS schemes (see "Using LOS schemes for showing aggregated pedestrian values" on page 186). Thus you can easily distinguish between different traffic parameters at different positions in the network during simulation, for example, to quickly locate a hotspot in a large network.

You can make similar settings for pedestrians in areas and vehicles on links (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to links based on aggregated parameters" on page 179).

You can simultaneously show the pedestrians in color. This is based on the attributes of the pedestrians (see "Assigning a color to pedestrians based on an attribute" on page 178).

1. Before starting a simulation, make the following settings:
2. From the **Evaluation** menu, choose **Configuration** > **Result Attributes** tab > **Areas & ramps**.
3. Select **Collect data**.
4. Set the parameters as required.
5. Confirm with **OK**.

6. Confirm with **OK**.
7. On the Network object toolbar, next to **Ramps & Stairs**, click the **Edit graphic parameters** button .

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).
8. In the **Drawing mode** list box, click **Use color scheme**.
9. In the list of graphic parameters, click **Color scheme configuration**.

   *The window **Edit Color Scheme for Ramps & Stairs** opens.*

   If the window does not open because the option has already been deactivated once since program start, click the  icon at the end of row to open the window.
10. Make the desired changes:

Symbol	Element	Description
	**Attribute**	Opens an attribute selection window. The result attributes can be filtered (see " Setting a filter for selection of subattributes displayed" on page 117).
	**Predefined color scheme**	Open **Select pre-defined color scheme** list box (see "Using LOS schemes for showing aggregated pedestrian values" on page 186): Select a defined color scheme and show it in the list below. The color schemes vary in color and class bound.

Element	Description
List	- **UpperBound** column: Enter values. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

11. If you want to change the RGB values in the **Color** column, double-click the desired row.
12. Confirm with **OK**.

4.2.10 Assigning a color to nodes based on an attribute

You can have nodes displayed in a color based on the values of an attribute. To display nodes in color, you can select a pre-defined color scheme or define one of your own.

1. On the Network object toolbar, next to **Nodes**, click the **Graphic parameters** button .

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).
2. In the **Drawing mode** list box, click **Use color scheme**.
3. In the list of graphic parameters, click **Color scheme configuration**.

   *The **Edit Color Scheme for Nodes** window opens.*

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.
4. Make the desired changes:

Symbol	Name	Description
	**Attribute**	Opens an attribute selection window. The result attributes can be filtered (see " Setting a filter for selection of sub-attributes displayed" on page 117).
	**Predefined color scheme**	Open **Select pre-defined color scheme** list box: Select a defined color scheme and show it in the **Class bounds and colors** list. The color schemes vary in color and class bound: - **Red-yellow-green**: 11 classes, class size by default 0.500, 11 colors - **Speed**: 11 classes, 11 colors from pink to red, yellow, green to MAX = white - **Density**: 11 classes, 11 colors from white to light blue, blue to MAX = white - **Volume**: 11 classes, 11 colors from white to light blue, blue to MAX = white - **Relative delay**: 11 classes, 11 colors from white to light blue, blue to MAX = white - **Level-of-service value**: six classes, six semi-transparent colors from blue to green and yellow to MAX = red

Name	Description
Class bounds and colors list	Edit color scheme. From the shortcut menu, choose **Add** to add a new row to the list and define additional class bounds and colors. - **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. Based on value range after **MIN**, on upper bound of the row above. - **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. - Column **Color**: Select the color. You may also enter RGB values. The unit of the values depends on the attribute selected and is displayed in the **Classification** section, below the attribute selected.
Range scale factor	- Enter a factor for the upper bound and lower bound - **Apply** button: Multiply values of upper bounds and lower bounds by a factor

5. If you want to change the RGB values in the **Color** column, double-click the desired row.
6. Select the desired colors.
7. Confirm with **OK**.

4.3 Using 3D mode and specifying the display

The 3D mode is used to show networks during a simulation or presentation in 3D. You use the 2D mode to edit networks and network objects (see "Calling the 2D mode from the 3D mode" on page 158).

4.3.1 Calling the 3D mode from the 2D mode

- On the Network editor toolbar, click the **2D/3D** button.

The button changes to . The Vissim network and vehicles are shown in 3D. The network can be viewed from any desired perspective.

Notes:

- When you load a bitmap graphic file and then call the 3D mode, the graphic file is converted into a texture. This might delay the 3D display and require more memory.
- Calculation of the size of a rectangular area accounts for static 3D models. This prevents problems arising through the display of very large models and gaps from being created between models and the area.

Tip: When you edit display options in the graphic parameters, your changes are saved to the **.layx* file as soon as you save the network file. You can also save a layout as a **.layx* file and to load it again later.

4.3.2 Navigating in 3D mode in the network

In the 3D mode, in the Network Editor toolbar, some features for navigating in the network differ from the 2D mode:

Button	Name	Description
	Show entire network	Vertical and central top view on the network
	Zoom out	Gradually zoom out network view (see " Zooming out" on page 82)
	Zoom in	Gradually zoom in network view (see " Zooming in" on page 82)
	Rotate mode (3D)	Tilt or rotate the network display level vertically or horizontally. • Hold down the mouse button and drag the cursor up or down: change height and angle of the observer position • Hold down the mouse button and drag the cursor left or right: rotate the viewing point around the network. Tip: Alternatively, press the ALT key and hold down the mouse wheel, while moving the mouse.
	Flight mode (3D)	The observer position is continuously moved through the network (see "Flight over the network" on page 195).

If you click the button **Flight mode (3D)**, the button **Rotate mode (3D)** and the Rotate Mode are deactivated.

If you click the button **Rotate mode (3D)**, the button **Flight mode (3D)** and the Flight Mode are deactivated.

4.3.3 Editing 3D graphic parameters

Note: Graphic parameters can be attributes of the network objects.

1. On the toolbar of the Network Editor, click the button **Edit 3D graphic parameters**.

   The list of 3D graphic parameters opens (see "List of 3D graphic parameters" on page 194).
2. Select the desired entries.
3. If you want to change colors, double-click the row of the desired 3D graphic parameter.
4. Click next to the list when you want to close it.

The list closes. The 3D graphic parameters are assigned.

4.3.4 List of 3D graphic parameters

You can define the following 3D graphic parameters. You can also change 3D graphic parameters during the simulation. 3D graphic parameters use the unit set for short distances **m** or **ft**.

Base graphic parameters are taken into account (see "List of base graphic parameters for network editors" on page 171).

Graphic parameter	Description
Show land	Shows or hides land area
Show map	Display or hide background map in 3D mode. The desired map must be selected in the base graphic parameter **Map provider** (see "List of base graphic parameters for network editors" on page 171). If you zoom out of the map view too far, Vissim no longer displays a background map. The background is then gray.
Land texture	Select graphic file for land area
Land texture horizontal length	Length of graphic in Network Editor
Land color	Color of land area, if no texture is selected

Graphic parameter	Description
Sky texture	Allows you to select graphic file with a texture for the sky. The sky is displayed as a hemisphere with a texture, if the 3D graphic parameter **Show land** is not selected.
Sky color	Color of sky, if no texture is selected
Underground texture	Allows you to select graphic file with a texture for the underground. The underground is displayed as a hemisphere with a texture, if the 3D graphic parameter **Show land** is not selected.
Underground color	Color of the underground, if no texture is selected The underground is displayed in the color selected, if the 3D graphic parameter **Show land** is not selected.
Fog mode	• **Off** • **Linear**: Define display of fog via a range. Enter distances for the 3D graphic parameters **Fog start** and **Fog end**. The 3D graphic parameter **Fog density** is disabled. • **Exponential**: Defines display of fog via a value for the density. The 3Dgraphics parameter **Fog start** and **Fog end** are disabled.
Fog density (exponential mode)	Density value, default value 6.0. With increasing value, the fog looks more dense. The Fog mode **Exponential** must be selected.
Fog start (linear mode)	Distance of viewer position to the front of the fog. The default value is 0 m. The Fog mode **Linear** must be selected.
Fog end (linear mode)	Distance of viewer position to the end of the fog. The default value is 250 m. The Fog mode **Linear** must be selected.
Show shadows	Show or hide shadows in static 3D models, for pedestrians and vehicles
Shadow color	The default value is light gray 80, 0, 0, 0 Set the level of transparency: Double-click into the color box and use the slider under **Alpha** to set the desired level of transparency.

4.3.5 Flight over the network

You can fly forward or backward over the network in 3D flight mode and change direction and speed in the process.

1. Make sure that the icon  **Flight mode (3D)** is selected in the toolbar of the Network Editor.
2. Press the key of your choice or shortcut:

Purpose	Key or shortcut
Execute flight forward over the network	Hold down the W key.
Execute flight backward over the network	Hold down the S key.
Change direction of the flight	You have the following options: - To the left: Additionally hold down the A key during flight simulation. - To the right: Additionally hold down the D. key during flight simulation. - Choose direction freely: Hold down the mouse button during flight simulation, and drag the cursor to the desired position.
Increase speed of flight	Additionally hold down the SHIFT button during flight simulation.

4.3.6 Showing 3D perspective of a driver or a pedestrian

You can select a vehicle or a pedestrian and show the view from their perspective in 3D mode. You can select another vehicle or pedestrian in any opened network editor to show different persepectives at the same time.

You can save the 3D perspective of a driver or pedestrian as a camera position. You can use the driver perspective or pedestrian perspective for AVI recordings (see "Recording a 3D simulation and saving it as an AVI file" on page 1158).

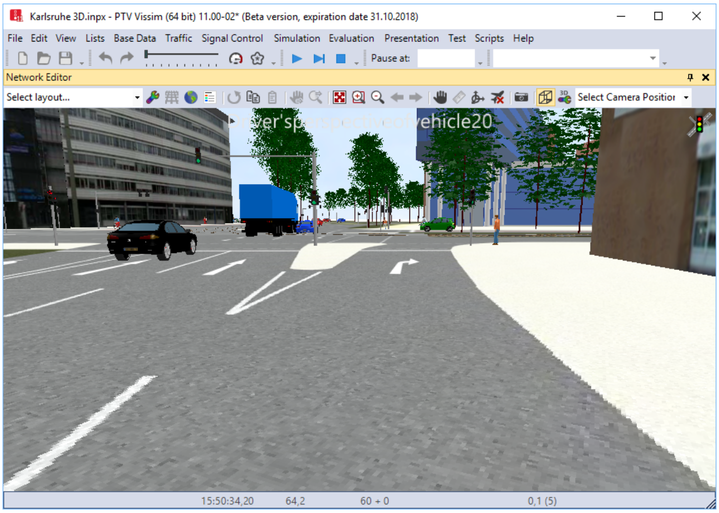
When you choose the camera position driver perspective, as in reality, the vehicle itself is not visible. When you choose a camera position outside the driver perspective, parts of the vehicle are displayed. This allows you to position the camera freely and choose a camera position attached to a vehicle or inside a vehicle. This applies accordingly to pedestrians.

4.3.6.1 Showing 3D perspective via a Network editor

Note: If under User Preferences, you selected **Right-click creates a new object**, hold down the CTRL key for the next step (see "Right-click behavior and action after creating an object" on page 152).

1. In a Network editor, in the single-step mode, right-click the vehicle or pedestrian of your choice.
2. From the context menu, choose the respective command.
   - **View From Driver's Perspective**
   - **View From Pedestrian's Perspective**

   In the Network editor, the perspective of your choice is shown in 3D mode.



Note: If under User Preferences, you selected **Right-click creates a new object**, hold down the CTRL key for the next step (see "Right-click behavior and action after creating an object" on page 152).

3. If you want to exit the perspective, right-click the Network Editor.
4. From the context menu, choose the respective command.
   - **Leave Driver's Perspective**
   - **Leave Pedestrian's Perspective**

Tip: Alternatively, you can switch back to the 2D mode.

4.3.6.2 Showing 3D perspective via a result list

1. From the **Evaluation** menu, choose **Result Lists** > **Vehicles In Network** or **Pedestrians In Network**.

   The result list opens.
2. Click on the desired vehicle or pedestrian in the result list.
3. The vehicle or pedestrian is marked in the Network Editor.

   The next step can be more easily executed in single-step mode or at a lower simulation speed.
4. Double-click the marked vehicle or pedestrian in the Network Editor.

   The perspective changes. When the vehicle or the pedestrian reaches its destination, the perspective is exited automatically.
5. If you want to exit the perspective, right-click or double-click into the Network Editor.

6. From the context menu, choose **Leave Driver's Perspective** or **Leave Pedestrian's Perspective**.

The simulation continues to run in 3D mode.

4.3.7 Changing the 3D viewing angle (focal length)

You can select the viewing angle in 3D mode. This corresponds to the setting of the focal length in photography.

The default perspective in Vissim is 45°. This corresponds to a focal length of 43mm in a 35mm system.

- If you want to reduce the perspective by two degrees, press the CTRL+PAGE UP keys.
- If you want to enlarge the perspective by two degrees, press the CTRL+PAGE DOWN keys.

The current perspective is displayed in the first column of the status bar during the change.

Notes:

- Vissim does not save any changes to the perspective. The default perspective is set again the next time you open Vissim.
- The modification of perspective applies to Rotate mode (3D) and Flight mode (3D) and all keyframes.

The table displays the corresponding focal lengths for different perspectives:

Perspective	Focal length (35 mm)	Perspective	Focal length (35 mm)
4°	500 mm	38°	53 mm
7°	300 mm	40°	50 mm
10°	200 mm	44°	45 mm
11°	180 mm	54°	35 mm
15°	135 mm	65°	28 mm
20°	100 mm	72°	25 mm
24°	85 mm	81°	21 mm
29°	70 mm	90°	18 mm

4.3.8 Displaying vehicles and pedestrians in the 3D mode

You can assign 3D models via model distributions to any type of vehicle or pedestrian (see "Using 2D/3D model distributions" on page 260).

4.3.9 3D animation of PT vehicle doors

- Select the desired 3D model of the PT vehicle (see "Using 2D/3D model distributions" on page 260).

  The 3D model of the PT vehicle is displayed.

For 3D models whose model file **.v3d* includes vehicle doors as modeled areas, the opening and closing of sliding doors or double doors at PT stops is also visualized in the 3D mode.

Tip: Under **Base Data** > **2D/3D Models**, you can use the model **Tram - GT8-2S** as an example. With the **Bus - EU Standard** model, the doors are not modeled as areas. This is why the opening and closing of sliding doors is not displayed in the 3D mode.

4.3.9.1 Requirements for the animation of doors

- Boarding and/or alighting via the door is permitted.
- Boarding and/or alighting at the PT stop is permitted.
- The door is located at a permissible platform position at the PT stop.
- If there is no platform edge and there are thus no passengers, then the door must be located at a permissible position within the PT stop length.

4.3.9.2 Temporal restrictions for opening the doors

The doors are opened for 1.5 seconds as soon as the vehicle comes to a standstill at the PT stop.

4.3.9.3 Temporal restrictions for closing the doors

- For calculated dwell times: The doors start to close 3 seconds prior to the end of the dwell time.
- For passengers in Viswalk: The doors start to close after no passenger has boarded or alighted the vehicle for 3 seconds.

  The closing of the doors always takes two seconds.

  - During the first of these two seconds, the doors will open again if a passenger wants to use one of them.
  - During second of these two seconds, passengers behave as if the doors were already closed.

Once the doors are closed, the vehicle waits another second before it departs.

The number of time steps depends on the simulation resolution.

4.3.9.4 Moving door polygons when doors are opened

- The door is moved perpendicular to the vehicle by 6 cm (duration: 0.3 seconds)
- It is further moved by its width in parallel to the vehicle element heading towards the direction with the greater distance to the next door and/or end of the vehicle.

Closing the doors corresponds to moving the door polygons in reverse order and direction.

4.3.10 Using fog in the 3D mode

For realistic simulations in 3D mode, you can add fog (see "List of 3D graphic parameters" on page 194).

- **Linear** mode from **Fog start** to **Fog end**. From the observer until **Fog start**, visibility is 100%. The fog looks dense if you choose a short distance between **Fog start** and **Fog end**. The fog looks less dense, the longer the distance is.
- **Exponential** The fog begins at the location of the observer based on the **Fog density** defined in the 3D graphic parameter.

Notes:

- Vissim saves no settings with regard to fog. If Vissim is closed and then reopened; the fog is switched off.
- The switching on of fog applies for all keyframes.

Examples of different types of fog effects

Fog mode: **Off**	
Fog mode: **Linear** **Fog start** 0.0 m: Fog starts at the location of the observer. **Fog-end** 100.0 m: Visibility extends until the point of Fog end.	
Fog mode: **Linear** **Fog start** 40.0 m: Fog starts at a distance of 40 m from the observer. **Fog end** 90.0 m: Visibility is limited from the point of fog start at 40.0 m and extends 50.0 m far until the point of fog end at 90.0 m.	

Fog mode Exponential: Fog starts at the position of the viewer. **Fog density** Default value 6.0: Fog density is relatively low, visibility is not limited.	
Fog mode Exponential: Fog starts at the position of the viewer. **Fog Density**: 30.0. This value is five times greater than the default value 6.0. This means visibility is very limited.	

5 Base data for simulation

The stochastic nature of traffic begs the necessity to provide this type of variability in Vissim models. The heart of Vissim, Wiedemann's car-following model, accounts for this by implementing parameters based on stochastic distribution (see "Operating principles of the car following model" on page 32).

The base data for simulation includes the settings for the entire network and all basic objects for modeling vehicle and pedestrian movement, e.g. distributions, functions, and behavior parameters. Base data further contains types and classes. These allow you to group properties that are the same for many network objects, so that you need not set them for each individual object.

In addition to input and output attributes, you can define user-defined attributes for all objects. User-defined attributes may be edited and managed in lists in the same way as predefined attributes.

For each attribute, you can specify an alternative name as an alias (see "Using aliases for attribute names" on page 217).

5.1 Selecting network settings

You can select the network default settings.

- **Vehicle Behavior**: Determine gradient from z-coordinate, choose right-side traffic or left-side traffic, activate Driving simulator add-on module (see "Selecting network settings for vehicle behavior" on page 203)
- **Pedestrian Behavior**: Define various settings for pedestrian behavior (see "Selecting network settings for pedestrian behavior" on page 204)
- **Units**: Select metric or imperial to display the units for length, speed and acceleration (see "Selecting network settings for units" on page 205)
- **Attributes**: Select separator and string length for indirect attributes that are linked using the **Aggregate function Concatenate** (see "Selecting network settings for attribute concatenation" on page 206)
- **Display**: Specify display settings for arrow of 3D traffic signals during red & amber signal, set angle of compass rose to the north direction, show coordinates of reference points in the background maps (see "Selecting network settings for 3D signal heads" on page 206), (see "Showing reference points" on page 208), (see "Selecting angle towards north" on page 209)
- **Standard types**:
  - For elevators and elevator groups, specify standard types for display types and area behavior types (see "Network settings for standard types of elevators and elevator groups" on page 207)

  - for the time a vehicle remains in standstill, between backing out of a parking space and driving forward, after it has left the parking lot: Standard time distribution for the attribute **Direction change duration distribution** of the parking lot.
- **Driving simulator**: Activate interface to your external driving simulator. Select vehicle type and/or pedestrian type you want to control (see "Network settings for the driving simulator" on page 210).

5.1.1 Selecting network settings for vehicle behavior

1. Select from the menu **Base Data** > **Network Settings**.
2. Select the **Vehicle Behavior** tab.
3. Make the desired changes:

Element	Description
Link gradient based on	- **Attribute 'Gradient'**: For driving behavior, the static gradient entered for the link during simulation is used. The z coordinates of the link section are ignored. - **Z-coordinates**: If this option is selected, during simulation the current gradient for driving behavior is always calculated from the z-coordinates of the link section on which the front edge of the vehicle is located. The given static gradient of the link is thereby ignored. Note: When selecting the option **Calculate from z coordinates**, make sure there is no connector, connecting links with a large difference in altitude (e.g. 0.5 m) over a very short distance (e.g. 1 m).
Traffic regulations	Controls the creation of opposite lanes, bus bays, and the driving behavior on freeways with a right/left side rule based on the selected traffic regulation: - **Right-side traffic** - **Left-side traffic**
Specific power for HGV	Power-to-weight ratio of vehicles of the category **HGV** in [kW/t], with t = metric ton: - **Minimum**: Lower limit of specific power. Default: 7.00, value range 1 to 10 - **Maximum**: Upper limit of specific power. Default 30.00, value range 11 s to 1,000 If you change the settings, the simulation results change even if there is no vehicle that has exceeded the upper limit or fallen below the lower limit. Vissim uses the upper and lower limits for interpolation. Then the maximum acceleration for all trucks can change and thus the simulation results.

Element	Description
Driving simulator	Notes: - You must have a license for the add-on module. - Verify that the connection to the driving simulator via the interface is configured correctly. - By default, information for developers in English is saved to the installation directory of your Vissim installation: *..\API\DrivingSimulator_DLL\doc*: *Driving_Simulator_Interface.pdf* - You can find sample files by default in the installation directory of your Vissiminstallation: *..\api\driving simulator_dll\example\driving simulator text client* - **Driving simulator active**: If this option is selected, the interface to your external driving simulator is activated. Up to 1000 vehicles can be added to the simulation using a driving simulator. You must select a vehicle type if you wish to start a simulation. - **Vehicle Type:**: Vehicle type, which is not controlled by Vissim during the simulation, but by your external driving simulator.

5.1.2 Selecting network settings for pedestrian behavior

You can define default values for pedestrian behavior. These serve as a global model parameters for the pedestrian routes that you define. When changing global model parameters, you do not change the parameters of the pedestrian routes already defined.

1. From the menu **Base Data** menu, choose **Network Settings**.
2. Select the **Pedestrian Behavior** tab.
3. Make the desired changes:

Element	Description (see "Defining global model parameters" on page 871)
Social force calculation	
Search neighborhood grid size	Maximum distance up to which pedestrians influence each other, default value 5.00 m
Potential cell size of internal destinations	This parameter specifies the distances at which control points are set by PT vehicles for calculation of the pedestrian route via stairways, ramps or through doors. As you are not required to enter intermediate points for pedestrian routes here, Vissim calculates the pedestrian route based on internal routing points of the pedestrian route. Default 0.15 m (see "Defining global model parameters" on page 871).
Default obstacle distance	only for calculation of the static potential: Specifies the distance up to which the nearby walls have a bearing on the distance potential. Default 0.50 m (see "Defining global model parameters" on page 871).

Element	Description (see "Defining global model parameters" on page 871)
Queuing	
Queue order	The higher this value, the more orderly (one after the other) pedestrians get in line in areas and elevators (see "Attributes of areas" on page 898), (see "Elevator attributes" on page 993): - 0.0: Pedestrians are standing together in groups - 1.0: Queue of pedestrians lined up one behind the other Default 0.70
Queue straightness	The larger this value, the straighter the queue will look that pedestrians in areas are waiting in (see "Attributes of areas" on page 898): - 0.0: snake shaped queue - 1.0: straight queue Default 0.60

Element	Description (see "Defining global model parameters" on page 871)
Behavior with dynamic potential (see "Dynamic potential" on page 968)	
Pedestrians - direction change clipping	☑ Select this option to allow for the angle between the fastest and shortest path to increase at any speed.
Direction change angle	Maximum permitted angle by which the angle between the quickest and the shortest path can increase from one time step to the next, default value 4.0°.

Element	Description (see "Defining global model parameters" on page 871)
Computation of experienced Level of Service	
Radius for the computation of the pedestrians personal area:	Radius around a pedestrian within which other pedestrians are recorded for calculation of pedestrian-based density (see "Showing pedestrians in the network in a list" on page 853), default value 2.00 m.

5.1.3 Selecting network settings for units

You can display the units for distance, speed and acceleration - either metric or imperial. The selected units are displayed by default in lists and windows. Many raw data are displayed by default with metric units.

1. From the **Base Data** menu, choose > **Network Settings**.
2. Select the **Units** tab.
3. Make the desired changes:

Element	Description
All Imperial	All parameters for lengths, speeds and accelerations are displayed in imperial units.
All Metric	All parameters for lengths, speeds and accelerations are displayed in metric units.
- **Length** - **Speed** - **Acceleration**	Select the desired unit for the parameter.

5.1.4 Selecting network settings for attribute concatenation

In list windows displaying indirect attributes, for the attribute selected, you can consecutively list the values of all network objects reached via a relation using the **Aggregate function Concatenate**. To do so, you can choose a separator and the maximum string length.

1. From the menu **Base Data** menu, choose > **Network Settings**.
2. Select the **Attributes** tab.
3. Make the desired changes:

Element	Description
Separator	Separator used between multiple attribute values specified in lists. Default: comma
Max. string length	Maximum number of characters listed in a row for the attribute values output. When the maximum number is reached, the output is cut off. If no value is specified, the output is not cut off. - Default: 255 - The box may remain empty. Then the string length is unlimited.

5.1.5 Selecting network settings for 3D signal heads

In the red and amber signal fields of 3D heads, you can select how you want to show arrows.

1. From the **Base Data** menu, choose > **Network Settings**.
2. Select the **Display** tab.
3. Make the desired changes:

Element	Description
Signal head arrow color red & amber (3D)	- **Black arrow on colored lens**: If this option is selected, a black arrow against a colored background is shown for red & amber. This is the setting according to the regulations in Germany. - **Colored arrow on black**: If this option is selected, a colored arrow against a black background is shown for red & amber. For Green, the setting is always a colored arrow displayed against a black background.

5.1.6 Network settings for standard types of elevators and elevator groups

The display of the cabin floor, the cabin ceiling, the shaft, and the door is based on a display type in each case. For each of these display types, you can select a default type.

Each pedestrian in the cabin and each pedestrian alighting from the cabin uses a walking behavior that is based on an area behavior type. You can select a default type for each of these area behavior types.

When you open an **.inpx* network file saved in a Vissim version that did not yet include standard types, Vissim will create the standard types automatically.

1. From the **Base Data** menu, choose **Network Settings**.
2. Select the **Standard types** tab.
3. In the **Elevators and elevator groups** section, in the list boxes, click the entries of your choice.

Element	Description
Display type cabin wall	Default display type of cabin walls
Display type cabin floor	Default display type of cabin floor
Display type cabin ceiling	Default display type of cabin ceiling
Display type shaft	Default display type of area between exterior of elevator shaft and cabin
Display type door	Default display type for the elevator door
Area behavior type in cabin	Default area behavior type of which the pedestrian uses the walking behavior in the cabin
Area behavior type alighting	Area behavior type of which the pedestrian uses the walking behavior when alighting from the cabin

The selected display type is used as a default for the display of the respective part of the elevator when defining the elevator and it is displayed in the list of elevator attributes (see "Elevator attributes" on page 993).

The selected area behavior type is used when defining an elevator group and displayed in the list of the elevator group attributes (see "Attributes of elevator groups" on page 996).

5.1.7 Network settings for standard type of direction change duration distribution

Time during which a vehicle remains in standstill, between backing out of a parking space and driving forward, after it has left the parking lot, based on the attribute **Direction change duration distribution** of the parking lot. You can select a time distribution for this attribute. The default time distribution is **5 s**.

When you open an **.inpx* network file saved in a Vissim version that did not yet include standard types, Vissim will create the standard types automatically.

1. From the **Base Data** menu, choose **Network Settings**.
2. Select the **Standard types** tab.
3. In the **Parking lots** section, in the list **Direction change duration distribution**, click the desired time distribution.

*The time distribution selected is used as a default value when you define the **Direction change duration distribution** used and is displayed in the list of attributes of the parking lot* *(see "Attributes of parking lots" on page 500).*

5.1.8 Showing reference points

You can define a point in the live map as a reference point and assign it to the corresponding coordinate of Vissim (see "Mapping Vissim network to background position" on page 396). The coordinates of both reference points are shown in the network settings.

1. From the **Base Data** menu, choose > **Network Settings**.
2. Select the **Display** tab.

   *The coordinates are shown in the **Background maps** area. The entry **No map assignment defined** is displayed, when in the Network editor, no network object has been inserted.*

Element	Description
Reference point in map	The coordinates specify the location of the Vissimnetwork geocoded on the background map. The **Reference point in network** is assigned to the **Reference point in map**. This means the two reference points overlap. The coordinates of the reference point in the map are Mercator coordinates. They refer to the point of intersection of the Equator and the international prime meridian. The intersection has the coordinates 0.000 (x, horizontal), 0.000 (y, vertical).
Reference point in network	The coordinates specify where the Vissimnetwork is anchored to the reference point on the background map. The coordinates 0.000, 0.000 indicate that the reference point in the network lies precisely on the reference point in the map. The Vissimnetwork is based on Cartesian coordinates, whereas the background map is based on Mercator coordinates. This why with increasing distance from the reference point in the network, deviations occur between the Vissimnetwork and the background map. However, there are no area distortions between the Vissimnetwork and the background map where the reference point in the network and the reference point in the map overlap.

Converting Vissimcoordinates into decimal degrees

You can convert Vissimcoordinates into decimal degrees.

Thereby the following applies:

π	*PI*: 3,14159265358979
r	6378137 m equatorial radius at reference object sphere
x	X coordinate of the point to be converted (Cartesian Vissimsystem)
y	Y coordinate of the point to be converted (Cartesian Vissimsystem)
x_m	X coordinate of the reference point in the network (Cartesian Vissim system)
y_m	Y coordinate of the reference point in the network (Cartesian Vissim system)
x_n	X coordinate of the reference point in the background map (Mercator)
y_n	Y coordinate of the reference point in the background map (Mercator)
lon	Longitude to P (by reference to sphere)
lat	Latitude to P (by reference to sphere)

$$l = 1.000930417(2atan(e^{(y_m/r)}) - \frac{\pi}{2})$$

$$s = \frac{1}{cos(l)}$$

$$m_x = (x - x_m)s + x_n$$

$$m_y = (y - y_m)s + y_n$$

$$lon = 1.001120232\frac{m_x}{r}\frac{180}{\pi}$$

$$lat = 1.000930417(2atan(e^{\frac{m_y}{r}}) - \frac{\pi}{2})\frac{180}{\pi}$$

You can find programs for the conversion of decimal degrees into other coordinate systems on the Internet. Check the results obtained for correctness prior to using the data.

5.1.9 Selecting angle towards north

If in your Vissim network, north is not at the top, in the Network Editor, you can turn the compass rose to have the red tip point north. When you turn the compass rose in the Network Editor, the background map cannot be shown.

1. Make sure that no background map is displayed (see "List of base graphic parameters for network editors" on page 171).
2. From the **Base Data** menu, choose > **Network Settings**.
3. Select the **Display** tab.
4. Enter the desired value.

Element	Description
North	**Angle to north direction:**: degrees, default 0.000 degrees

5.1.10 Network settings for the driving simulator

Notes:

- You must have a license for the add-on module.
- Verify that the connection to the driving simulator via the interface is configured correctly.
- By default, information for developers in English is saved to the installation directory of your Vissim installation: *..\API\DrivingSimulator_DLL\doc*: *Driving_Simulator_Interface.pdf*
- You can find sample files by default in the installation directory of your Vissiminstallation: *..\api\driving simulator_dll\example\driving simulator text client*

1. From the **Base Data** menu, choose > **Network Settings**.
2. Select the **Driving simulator** tab.
3. Make the desired changes:

Element	Description
Driving simulator active	Select this option to activate the interface to your external driving simulator. Up to 1000 vehicles and 1000 pedestrians can be added to the simulation using a driving simulator. You must select a vehicle type if you wish to start a simulation.
Vehicle type	Vehicle type which is not controlled by Vissim during the simulation, but by your external driving simulator.
Pedestrian type	Pedestrian type which is not controlled by Vissim during the simulation, but by your external driving simulator.

5.2 Using user-defined attributes

For most network objects and base data objects, you can define further attributes in addition to the input attributes, output attributes and user-defined attributes (UDA). User-defined attributes (UDA) may be edited and managed in lists in the same way as predefined attributes (see "Creating user-defined attributes" on page 211).

You may read in user-defined attributes additively from another Vissim network (see "Reading a network additionally" on page 361).

When you open a **.inpx* file in which a user-defined attribute is defined whose short or long name corresponds to the name of an attribute in Vissim, Vissim adds a suffix to the name of the user-defined attribute:

- In the event of a short name: *ShortName_UDA_<next higher number available>*
- In the event of a long name: *LongName (UDA <next higher number available>)*
- If the short or long name contains a number in the suffix, the other name either receives a suffix with the same number or no suffix.

Data attribute or formula attribute

For a user-defined attribute, select one of the following data source types:

- **Data**: The user-defined attribute is based on Vissim data. You can create a default value, minimum value and maximum value and specify the number of decimal places.
- **Formula**: You create the desired formula using operands, operators, and/or functions. Use parentheses to structure them. You can define TableLookup function as part of the formula. The TableLookup function allows you to access attributes and attribute values of other objects and use them in the formula.

Examples of use

- Continue to process result attributes: You define a user-defined attribute that contains a formula for the LOS calculation model of your choice. This formula for instance uses queue length data, derived from node evaluation and emission factors.
- You define user-defined attributes, for which you obtain values via the COM Interface or DLL interfaces.
- Support data calibration: You define user-defined attributes and fill them with real data. You can show this data in Vissim in lists together with the simulation data. You can then compare the data of your two sources.
- You define user-defined attributes for vehicles or pedestrians of the simulation, add data via the COM Interface and output this data together with the simulation results, e.g. in a vehicle record.
- You define user-defined attributes for vehicles or pedestrians of the simulation that contain a formula. This formula contains the LOS calculation model of your choice.

5.2.1 Creating user-defined attributes

You can create a user-defined attribute in the Base data menu or the Attribute selection window (see "Selecting attributes and subattributes for columns of a list" on page 112).

Creating a user-defined attribute in the Base data menu

1. From the **Base Data** menu, choose > **User-defined attributes**.

   *The **User-defined attributes** list opens. If no user-defined attribute is defined, only the column titles are displayed.*

2. In the list, on the toolbar, click the **Add** button ✚.

   *The **User-defined attribute** window opens.*

Tip: Alternatively, you can also open the **User-Defined Attribute** window via the following functions: When doing so, you adopt the network object type as the object type of the user-defined attribute.

- On the Network object toolbar, from the shortcut menu of the desired network object type, choose **Create User-Defined Attribute**.
- In the Attributes list of the desired network object type, right-click the row header of the desired network object. Then from the shortcut menu, choose **User-Defined Attribute**.
- In the Attributes list of the desired network object type, on the toolbar, click the **Create User-Defined Attribute** button .

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

3. Make the desired changes:

Element	Description
Object type	Object type of user-defined attribute (mandatory field) In addition to the network object types, you can select other object types, such as **Network settings** (**NETPARA**). For **NETPARA**, choose the data source type **Formula** to enter a formula value that you can use as a user-defined constant. Select **TableLookup** to access **NETPARA**.
Short name	Abbreviated name of user-define attribute (mandatory field)
LongName	Full name
Comment	Optional comment

Element	Description
Data type	A user-defined attribute must have a data type. Select the desired entry.
Data types allowed:	
Acceleration	Acceleration
Bool	For logical statements, true or false. A user-defined attribute that can only be set to 0 or 1.
Filename	Filename: Reference to a file, for example for a user-defined attribute, which lists where to find further information.
Duration (integer)	Time [s] without decimal separator
Duration (floating-point number)	Time [s] with decimal separator
Area (km2)	Square kilometers
Area (m2)	Square meters

Element	Description
Integer	Integer without decimal separators
Speed (km/h)	Speed (km/h)
Speed (m/s)	Speed (m/s)
Floating-point number	Floating-point number with decimal separators
Length (km)	Length (km)
Length (m)	Length (m)
Length (mm)	Length (mm)
Text	String
Currency	Currency
Time	Time in format `hh:mm:ss (12:23:12)` or `hh:mm (12:23)`

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Subattributes</td><td>

Simulation run: only for the following objects:

User-defined attributes of network objects of the data source type Data that have evaluation attributes and can have user-defined data attributes. The values of these attributes can only be changed during the simulation run and are saved to the evaluation database (*.db).
User-defined attributes of the data source type Formula

The values of the user-defined attribute are recorded for each simulation run. When you show the user-defined attribute in an attributes list, the attribute values are listed per column for each simulation run.
Time interval: For user-defined attribute consider time intervals of network object type. The network object type must be assigned time intervals (see "Defining time intervals for a network object type" on page 326). When you show the user-defined attribute in an attributes list, the attribute values are listed per column for each time interval.

</td></tr>
</table>

4. Select the desired data source type.

- **Data**: You can set the following attribute values and options. Depending on the data selected, different boxes and options are displayed in the section on the right.
    - **Default**: For optional entry of a default value
    - **Minimum value**: **Has Value**: If this option is selected, choose the smallest possible value as default.
    - **Maximum value**: **Has Value**: If this option is selected, choose the largest possible value as default.

- **Formula**: In the section on the right, you can create a formula using the following commands:

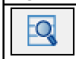
Element	Description
Decimals	Number of decimal places for a value with decimal separator

Symbol	Description
x	Open a window with the attributes of the network object type selected in the **Object type** box and select an attribute as term. The term is added in blue.

Symbol	Description
±	Select operator that combines one term with another. The operator is added in black.

Symbol	Description
f(x)	Open the **Insert function** window and select a function for the formula. The function is added in turquoise.
Possible functions:	
Floor(x)	Specifies the greatest integer that is smaller than or equal to `x`. At the cursor position `FLOOR()` is inserted.
Abs(x)	Absolute value of `x`
Truncate(x)	Truncates the decimal places of `x`. At the cursor position `TRUNCATE()` is inserted.
Ceil(x)	Specifies the smallest integer that is greater than or equal to `x`. At the cursor position `CEIL()` is inserted.
Exponential(x)	Forms the exponential of `x`. At the cursor position, `EXP()` is inserted.
GEH Statistic (x; y)	Calculates the GEH statistics for `x` and `y`. At the cursor position, `GEH(;)` is inserted. The GEH Statistic is a empirical formula widely used in traffic modeling for the analysis of values. The formula is as follows: $GEH = \sqrt{\frac{2(M-C)^2}{M+C}}$ where: M: modeled values C: counted values
Reciprocal(x)	Corresponds to `1/x`. At the cursor position `RECIPROCAL()` is inserted.

Symbol	Description
Logarithm(x)	Forms the natural logarithm of `x`. At the cursor position, `LN()` is inserted.
Max(x; y)	Forms the maximum of `x` and `y`
Min(x; y)	Forms the minimum of `x` and `y`
Power(x; y)	Corresponds to `x^y`. At the cursor position, `POW(;)` is inserted.
Percent(x; y)	Corresponds to `100*x/y`. At the cursor position `PERCENT()` is inserted.
Round(x; p)	Rounds `x` to `p` places after the decimal point. `p` is optional. At the cursor position `ROUND(;)` is inserted.
StringInNumberl(x)	Converts a string into a floating-point number. At the cursor position `STRTONUM()` is inserted.
If(b; w; f)	Conditional branching. Outputs `w` if `b` is true. Outputs `f` if `b` is false. At the cursor position `IF(;;)` is inserted.
Root(x)	Forms the square root of `x`. At the cursor position, `SQRT()` is inserted.
NumberInString(x)	Converts a floating-point number into a string. At the cursor position `NUMTOSTR()` is inserted.

Symbol	Description
	Open the **Insert TableLookup** window to insert a function of the type **TableLookup** into the formula. Select an attribute value of a network object type and continue to edit it in the formula. The TableLookup function is added in turquoise. - **Network object type**: In the list, click the object type you want to use as the basis of the function. All conditions of the function that follow refer to the object type selected. In addition to the network object types, you can select other object types, for instance **Network settings** (**NETPARA**) that contains a user-defined value. To select another object type, as condition of the TableLookup command, e.g. use the value **1** for **TRUE**. - **Variable name**: Enter a variable name that starts with a letter. This name is a so-called loop variable. In each loop, it represents another network object of the type iterated over. - **Insert explaining comments**: Inserts an example of a condition and result into the input field that you can overwrite.
	The **TableLookup** function entry box must correspond to the following scheme: `TableLookup(<object type> <variable name>;(<condition>);(<result>)` The elements consist of:

Symbol	Description
	TableLookup(<object type: English Name in capital letters> <entered variable name>; (<condition, for example <entered variable name>[<English attribute short name>]=<property>>); (<result, for example <identifier>[<attribute short name>]>)) For a detailed example, see below the tables.

Symbol	Description
( )	Insert opening and closing parentheses: - Around selected part of formula - None of the formula is selected: Insert at the position of the cursor

If the attributes selected for the formula are based on dynamic data that can change during a simulation run, the values of the user-defined attribute displayed in a results or attribute list can also change during the simulation run.

5. Confirm with **OK**.

Syntax errors are highlighted in red

Vissim highlights formula syntax errors in red. Below the entry box for the formula, a message is displayed with information on the possible cause of the syntax error.

Example of a user-defined attribute

The user-defined attribute **ConflGap** will calculate the gap in front for the two links of a conflict area, if **Link1** has the **Status2** and **Link2** has **Status1**.

For this example the following data are selected or entered:

- **Object type**: Links
- **Short name**: ConflGap
- **Long name**: ConflAreaFrontGap
- **Comment**: Front gap of a conflict area of this link, where vehicles need to observe the right of way
- **Data type**: Entry **floating point number**
- **Data source type**: **Formula** option

In the **Insert TableLookup** window:

- **Object type**: Conflict areas
- **Variable name**: ca

In the **Formula entry box**, change entry TableLookup(CONFLICTAREA ca; ;) to:

```
TableLookup(CONFLICTAREA ca; ca[LINK1\NO]=[NO] & ca[STATUS]=2 | ca
[LINK2\NO]=[NO] & ca[STATUS]=1;ca[FRONTGAPDEF])
```

The user-defined attribute **ConflGap** is then available as an attribute of the network object type **Link**. In the **Links** list, you can show **ConflGap** as a column. You can also show the **Front gap** value for each link in the list that has conflict areas with the two statuses defined.

5.2.2 Editing user-defined attribute values

In the attribute list of the network object type selected as **object type** for the user-defined attribute, you can show the attribute in a column. In the attribute list, you can edit the values of the user-defined attribute.

1. From the **Lists** menu, choose the network object type selected as **Object type** for the user-defined attribute.

   The Attribute list opens.

2. Click on the  **Select attributes** icon.

   *The window **<Name Network object type>: Select Attributes** opens. User-defined attributes are highlighted with a black circle.*

3. If desired, click the **Filter** button and filter the data (see " Setting a filter for selection of subattributes displayed" on page 117).
4. If you have filtered data, confirm **Preselection Filter** window with **OK**.
5. Repeat the following steps for all attributes that you want to show in the attribute list.
6. In the section on the left, click the user-defined attribute.
7. Click the icon .

   The attribute selected on the left is listed on the right in an additional row. You cannot edit hatched cells or the attribute name.

8. If desired, edit the value in a cell on the right (see "Selecting attributes and subattributes for columns of a list" on page 112).
9. Confirm with **OK**.

In the attribute list, a column with the values of the user-defined attribute is displayed. A column with attribute values is shown for each filtered sub-attribute.

10. Edit the desired entries.

5.3 Using aliases for attribute names

An alias is an alternative name for an attribute name. You can use aliases in particular to give long names of indirect attributes an alternative, shorter name that meets your requirements.

The alias is displayed in the following elements:

- In the attributes list: the alias replaces the standard name of the attribute. A column must be shown for the attribute, for which you have defined the alias..

- In the Attribute Selection window, the alias is displayed at the position specified by alphabetical order. The alias is highlighted with a blue symbol (see "Selecting attributes and subattributes for columns of a list" on page 112). The standard name of the attribute is still displayed.

The alias is output in the following files:

- Network file (*.*inpx*)
- Attribute file (*.*att*)
- Model transfer file (*.*trax*)
- Pedestrian record file (*.*pp*)
- Vehicle record file (*.*ftp*)

5.3.1 Defining aliases

1. From the **Base Data** menu, choose > **Aliases**.

   *The **Aliases** list opens.*

2. In the list, on the toolbar, click the **Add** button .

   *The **Alias** window opens.*

3. Make the desired changes:

Element	Description
Network object type	**ObjTypeName**: In the list, click the network object type with the attribute for which you want to enter an alternative name.
Attribute	**AttrName** : Opens the window **<Network object type>: Select attribute**. All attributes of the network object type are displayed. Click the desired network object. You can filter the entries (see " Setting a filter for selection of subattributes displayed" on page 117). Confirm with **OK**.
Name	Alias name of your choice

4. Confirm with **OK**.

*In the **Aliases** list, a new row is inserted. You can edit the alias in the **Name** column.*

The alias is displayed:

- If in the attributes list of the network objects of the network object type, a column is shown for the attribute, the alias replaces the standard attribute name.
- In the Attribute Selection window, the alias is displayed at the position specified by alphabetical order. The alias is highlighted with a blue symbol (see "Selecting attributes and subattributes for columns of a list" on page 112). The standard name of the attribute is still displayed.

Tip: Alternatively, define an alias in the opened list of network object attributes:

1. Right-click into the column header of the column with the desired attribute.
2. From the shortcut menu, choose **Add alias**.

Tip: Alternatively, you can define an alias in the attribute selection window (see "Selecting attributes and subattributes for columns of a list" on page 112).

5.3.2 Editing aliases in the Attribute selection list

You can define an alias for an attribute, change the name of the alias or delete the alias.

1. On the toolbar, click the **Attribute selection** button.

   *The attribute selection window **<Name Network object type>: Select Attributes** opens.*

2. Make the desired changes:

Symbol	Description
	The **Add alias...** symbol is displayed when you point the mouse pointer to the name of an attribute in the Explorer: Opens the **Alias** window (see "Defining aliases" on page 218).
	The **Edit alias** symbol is displayed when you point the mouse pointer to the name of an alias in the Explorer: Opens the **Alias** window. You can change the name of the alias.
	The **Delete alias** symbol is displayed when you point the mouse pointer to the name of an alias in the Explorer: Deletes the alias. The alias is no longer displayed in the Explorer or other elements.

5.4 Using 2D/3D models

A 2D/3D model defines the visualization of static and moving objects. In the attributes of 2D/3D models of vehicles and pedestrians, you can define their dimensions. A 2D/3D model may consist of one or multiple 2D/3D model segments. A 2D/3D-model segment can be based on a model file **.v3d*, **.skp*, **.3ds*, or **.dwf*.

This model file is also used for visualization in the 3D mode. For each 2D/3D model segment, you can specify the positions of axles, shaft length and joints, if your model segment includes these elements. In addition, you can define doors for vehicles whose position and size are relevant for the simulation of passengers boarding and alighting from PT vehicles.

You can assign colors to certain parts of the vehicle chassis of 2D/3D models of some buses and trains. This option is provided in addition to the color assigned to the PT vehicle based on the PT line. For the respective vehicle type, select the desired color distribution for the attributes **Color 2 Color 3**. If these two colors are not assigned a color distribution, the respective surfaces are assigned the same color as the PT line.

5.4.1 Defining 2D/3D models

You can define 2D/3D models for vehicles and pedestrians with or without 3D model files. With 3D model files, the following file formats are supported:

Format	Model file
.v3d	Vissim 3D

.skp	SketchUp
	Vissimoffers advanced functionality if certain naming conventions are contained in the **.skp* file before adding it to Vissim:
	- Functional colors: Use one of the names of the list below as a name of the material in SketchUp to assign the corresponding functionality to all objects or surfaces of that material:
	  - **Color group 1**, **Color group 2**, **Color group 3**, **Color group 4**: The color in Vissim is determined by the corresponding Vissimvehicle or pedestrian attribute **Color 1** … **Color 4**.
	  - **Indicator left**
	  - **Indicator right**
	  - **Brake lights**
	- Animated doors: Each door object in the SketchUp model must be grouped separately and the group named **Door**. Then Vissim will auto-generate corresponding door objects upon importing the 3D model.
	
	Note: This advanced functionality will work only for objects or surfaces that are not part of any SketchUp **Component**. To ensure this, use SketchUp's **Explode** command on all referenced objects. Keep in mind that objects may also be embedded.
	
	Vissim adds the 3D model of the vehicle, which is based on a SketchUp file, into the network in the correct travel direction. If necessary, the model is rotated. If the **.skp* file contains coordinates of the geolocation, you can position the 3D model in the Vissim network based on these coordinates or based on the position you clicked.
	For the following use cases, edit the 3D model in SketchUp, before you add it in Vissim:
	- Scale size
	- Colors for **Color group 1**, **Color group 2**, **Color group 3**, **Color group 4**, **Indicator left**, **Indicator right**, **Brake lights**
	- Define door object

Format	Model file
	Automatically generate the correct orientation of *.skp files: In the **Add 2D/3D model** window, in the **Orientation & Position** section, click the **Generate automatically** button. In most cases, a correct orientation is achieved. While in most cases this will result in the desired orientation, it may not always do so, e.g. it the width of a vehicle is larger than its length. In these cases, enter values for the **Yaw angle**, **Offset X**, **Offset Y**, and **Offset Z**. In addition, also a scale factor can be defined if the original model is out of scale. Then refresh the orientation and position.
.3ds	Autodesk 3ds Max
.dwf	Autodesk Design Web Format

You can also position 3D models of static 3D objects in the Network editor, e.g. to display buildings, plants or other static objects (see "Defining static 3D models" on page 674).

Note: When editing and saving a 3D-model file outside of Vissim that you have added in Vissim, close and re-open Vissim for the changes to take effect in Vissim.

5.4.1.1 Defining a 2D/3D model based on a 3D model file

1. Select from the menu **Base Data** > **2D/3D Models**.

   *The coupled list **2D/3D Model Segments** opens.*

2. In the list, on the toolbar, click the **Add** button .

   *The **Open** window opens. By default, 3D models are saved to the following directories and subdirectories:*

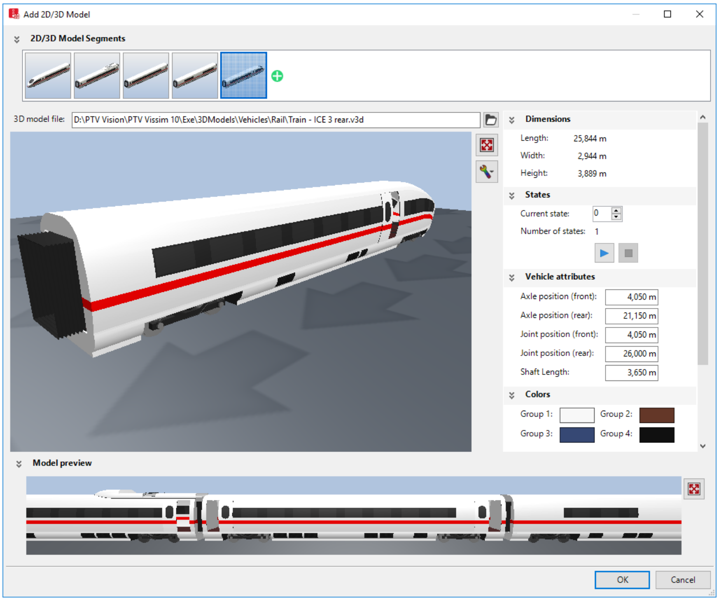
   - *..\Exe\3DModels\Pedestrians*: Pedestrian models of boys, girls, men, women, wheelchair users, women with child
   - *..\exe\3DModels\vehicles*: Models for vehicles:
     - *Rail*: Segments for trams and subways
     - *Road*: Bikes, motorbikes, scooters, cars, buses and segments for buses, trucks with trailers

   *All files saved to the selected path are listed in the section below them. The names of the 3D models are standardized. When importing a *.inp or *.inpx network file, references to old 3D model file names are replaced automatically by the new file names. If Vissim does not find a file, a message opens.*

3. Select the desired directory.

4. Double-click the desired file **.v3d*, **.skp*, **.3ds* or **.dwf*.

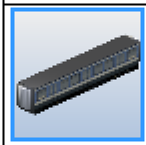
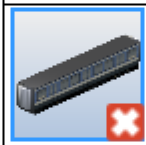
   *The **Add 2D/3D Model** window opens.*



The window consists of the following sections:

- Section **2D/3D model segments**: You can create a 2D/3D model from segments.
- Large Preview window: Displays the selected 2D/3D model.
- Attributes **Dimensions**, **States**, **Vehicle attributes**, **Colors**, **Orientation & Position**: You can adjust the values.
- Section **Model preview**: : Show the complete 2D/3D model created from segments. Rotating the scroll wheel changes the display:
  - Rotate down: enlarge (zoom in)
  - Rotate up: reduce (zoom out)

*If at the top, the **2D/3D model segments** section has been expanded using the symbol, the model is displayed in the narrow preview at the top. You can add further models to it. This allows you to create a model from segments, e.g. a train that consists of models for a power car and several railway cars. In the **2D/3D model segments** section, you can change the sequence of individual 2D/3D model segments and delete 2D/3D model segments:*

Element	Description
	Opens or closes the **2D/3D model segments** section
	Opens the **Open** window. You can select the file of a 2D/3D model and add it as a model segment behind the last model listed in the **2D/3D model segments** section. All model segments are displayed in the **2D/3D model segments** list. To show all model segments that belong to a 2D/3D model, In the **2D/3D model** list, select the 2D/3D model. Then, on the list toolbar, in the **Relations** list box, click **2D/3D model segments** (see "Assigning model segments to 2D/3D models" on page 225).
	Move segment: Click the image of the model segment, hold down the mouse button and drag the image to the desired position in the sequence of the model segments.
	Delete segment: Point the mouse pointer to the bottom right corner and click the symbol.

Below it, a large Preview window shows the selected 2D/3D model. If the 2D/3D model includes elements that move or change, e.g. doors or indicators, an animation is displayed in the Preview window.

5. Make the desired settings for the attributes.

The window also provides the following commands:

Element	Description
3D model file	Path and file name of the selected 2D/3D model file
Large Preview window	3D display of the selected 3D model. ▸ Zoom: Turn the mouse wheel. ▸ Rotate the model: Left-click and keep the mouse button pressed while moving the mouse pointer in the desired direction.
	Opens the **Open** window for selection of a 2D/3D model file
	Resets the Preview to default settings. Does not reset the attribute values.

Element	Description
	**Adjust visualization**: Show options that allow you to display additional elements in the preview. For example, **Show axles**, **Show joint and shaft length** or **Show ground plate**: If the option is selected, the element selected is displayed in the preview. For example, in the case of **Show ground plate**, a gray, transparent ground plate is displayed below the 3D model. This view supports the orientation during rotation and tilting of the 3D model in the preview. If this option is not selected, the element selected will not be displayed in the preview. This view corresponds to the view provided by the Network editor.
	If various model states are available, for example for moving pedestrians or cyclists, all states are displayed automatically one after the other.
	Stops the animation of the states.
Vehicle attributes	The default values of vehicle attributes depend on the model.
Colors	Colors for the different areas of the model. These are based on the colors selected for the **vehicle type** (see "Editing static data of a vehicle type" on page 270): - **Group 1**: Color 1 - **Group 2**: Color 2 - **Group 3**: Color 3 - **Group 4**: Color 4
Orientation & position	- **Scale**: Factor used to scale the 2D/3D model. - **Yaw angle**: Angle of rotation around z axis - **Offset X**: X coordinate of the position in the network - **Offset Y**: Y coordinate of the position in the network - **Offset Z**: Base height of 3D model above the level surface - **Generate automatically**: Position the 2D/3D model automatically

6. Confirm with **OK**.

   The model is saved. If you have grouped the elements, the vehicle length is calculated as the sum of elements and displayed in the corresponding window of each vehicle type (see "Using vehicle types" on page 267).

- In 2D mode, the vehicle is always displayed with the data from the **2D/3D Model Segments** list (see "Attributes of 2D/3D model segments" on page 227).
- In 3D mode, the 3D model of the selected file is used. Changes to the data in the **2D/3D Model Segments** list result in the geometric data such as length or the axis positions of the preselected 3D model file in the simulation being ignored. This may result in that in the 3D visualization, vehicles overlap or seemingly hold very large distances. If the geometric data are not suitable for the model file when loading the network file **.inpx*, a warning appears.

- Selection of a new 3D model overwrites all geometric data.
- If there is no reference between the 2D model and 3D model for a vehicle or pedestrian type, vehicles and pedestrians of that type are displayed in 3D mode as a colored cuboid.
- Since 3D elements have a static length, a length distribution can be defined in which you select various models with different lengths for a distribution.
- The color of a distribution, a class or a PT line is used to assign a color to the selected surfaces of the 3D model. Surfaces of Vissim which are to be displayed by color can be defined in the add-on **V3DM** module if the corresponding base models are available.
- During the simulation, the tractrix curves of the vehicles are used for vehicle display. Therefore, the turning behavior, in particular of the multi-part vehicles, seems more realistic; the higher simulation resolution is selected.
- 2D/3D model distributions are predefined for each vehicle type. The distribution for cars contains 7 different car models with different percentages (24 %, 16 %, 16 %, 16 %, 14 %, 20 %, 10 %). These vehicle models have been assigned as a relation **2D/3D model distribution elements** of the 2D/3D model distribution **Car**. The other 2D/3D model distributions are also assigned as a relation **2D/3D model distribution elements**.
- Changes to the model file of a standard vehicle model only affect the simulation result when the **Select 3D Model** window is closed with **OK**.

5.4.1.2 Defining 2D/3D models without a 3D model file

1. Select from the menu **Base Data** > **2D/3D Models**.

   *The **2D/3D Models** coupled list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. Right-click the row header.
3. From the context menu, choose **Add Without File**.

   A new row with default data is inserted.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

4. Into the list, enter the desired attribute values.

In the list on the right, you can show 2D/3D model segments, assign them to a 2D/3D model, and edit attributes (see "Assigning model segments to 2D/3D models" on page 225).

5.4.2 Assigning model segments to 2D/3D models

1. Select from the menu **Base Data** > **2D/3D Models**.

   *The **2D/3D Models** list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. Select the desired entry.
3. On the list toolbar, select **2D/3D model segments** in the **Relations** list box.

   If in the list on the left, the selected 2D/3D model has already been assigned model segments, their attributes are displayed on the right, in the ***2D/3D model segments*** *list. The 2D/3D-model can be assigned only one model segment, for example a single vehicle, or multiple model segments, for example, for a train that consists of individual model files for the traction head and several goods wagons.*

   For each model selected in the list, you can add rows in the list on the right that each contain a model segment.
4. To add a model segment to the list on the right, on the toolbar of the list, click the **Add** button .

   The ***Edit 2D/3D Model*** *window opens.*
5. At the top of the **2D/3D model segments** section, click the button.

   The ***Open*** *window opens.*
6. Open the desired folder and select the file of your choice.
7. Click the **Open** button.

   If at the top, the ***2D/3D model segments*** *section has been expanded using the symbol, the model is displayed in the narrow preview at the top. You can add further models to it. This allows you to create a model from segments, e.g. a train that consists of models for a power car and several railway cars. In the* ***2D/3D model segments*** *section, you can change the sequence of individual 2D/3D model segments and delete 2D/3D model segments (see "Defining 2D/3D models" on page 220).*

   Below it, a large Preview window shows the selected 2D/3D model. If the 2D/3D model includes elements that move or change, e.g. doors or indicators, an animation is displayed in the Preview window.
8. Confirm with **OK**.

The attributes of the model segment are displayed in the ***2D/3D model segments*** *list, in a new row.*

9. To edit model segments in the list on the right, right-click into the row header.
10. Select the desired entry.

11. Enter the desired data.

The data is allocated.

Tip: You can also assign model segments to models in the **Select 3D model** window (see "Defining 2D/3D models" on page 220).

5.4.3 Attributes of 2D/3D model segments

If your vehicles should be equipped with axles, shafts or clutches, you can define these attributes in model segments.

1. From the **Base Data** menu, choose > **2D/3D Model Segments**.

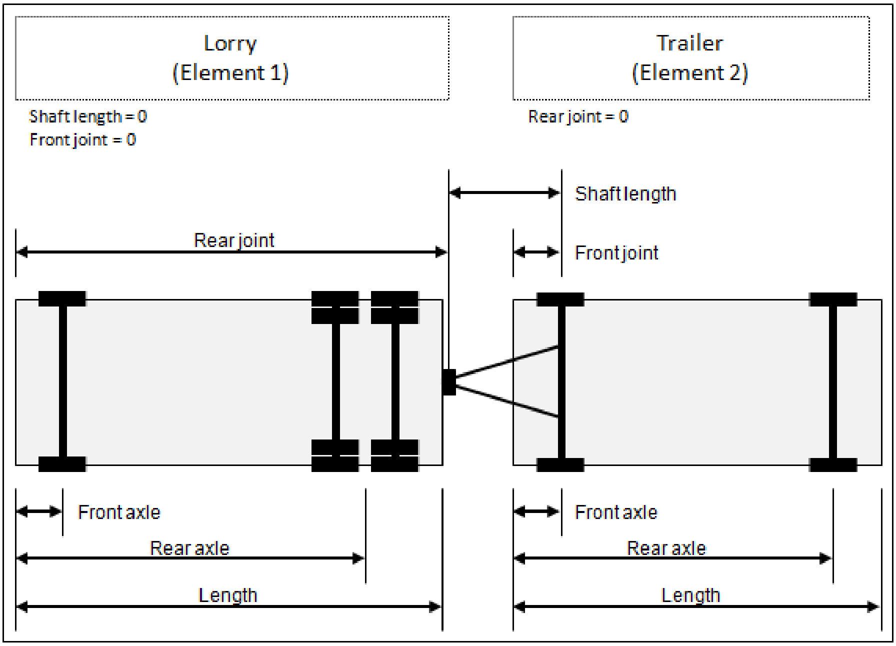
   *The **2D/3D Model Segments** list opens.*

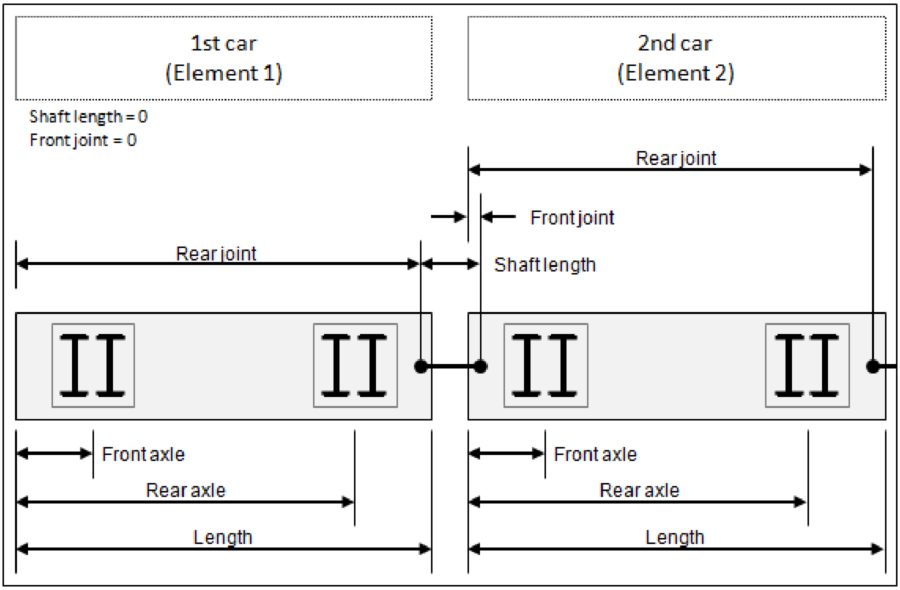
The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Click on the desired entry.
3. Into the list, enter the desired attribute values.

Element	Description
Index	Consecutive number of model segment
File3D	**3D model file**: Name of the file **.v3d *.3ds*, **.dwf* or **.skp*, by default saved to the directory *..\exe\3DModels* or to one of the subdirectories in your Vissim installation. Click on the file name to show the path to the directory.
Length	Length
Width	Width
Height	Height
ShaftLen	Shaft length
JointFront	Joint position (front)
JointRear	Joint position (rear)
AxleFront	Axle position (front)
AxleRear	Axle position (rear)
2D/3D model	2D/3D model





4. On the list toolbar, in the **Relations** list, click the desired entry.

   - **2D/3D model**: Edit 2D/3D model assigned (see "Defining 2D/3D models" on page 220)

- **Doors**: Display list of assigned doors and edit attribute values (see "Defining doors for public transport vehicles" on page 229)

The data is allocated.

5.4.4 Defining doors for public transport vehicles

You can define doors for the 2D/3D model segments of the 2D/3D models of public transport vehicles

Note: Make sure the length of PT vehicles matches the length of the PT stops. For the Viswalk simulation, all doors need to be located in full width within the stop and within the corresponding pedestrian area of the **Platform edge** type.

1. From the **Base Data** menu, select **2D/3D Model Segments**.

   *The **2D/3D Model Segments** list opens.*

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

2. Select the desired 2D/3D model segment of a PT vehicle.
3. On the list toolbar, in the **Relations** list box, click > **Doors**.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

Element	Meaning
Pos	Position: distance between door and front of the vehicle element
Width	Door width
yOffset	y-offset: Distance between the door and the longitudinal axis through the vehicle center
ZOffset	ZOffset: height above the level
Side	• **Both**: The element has doors on both sides • **Right**: Direction of traffic right • **Left**: Direction of traffic left
Usage	• **Boarding**: only for boarding • **Alighting**: only for lighting • **Both**: for alighting and boarding • **None**

4. Right-click on the row header in the right-hand list.
5. From the shortcut menu, choose **Add**.
6. Enter the desired data.

You can define further doors.

5.4.5 Editing doors of public transport vehicles

1. From the **Base Data** menu, select **2D/3D Model Segments**.

   *The **2D/3D Model Segments** list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. Right-click the desired 2D/3D model segment of a public transport vehicle.
3. Select the **Edit Doors** entry in the context menu.

   *The **Doors** list opens on the right-hand side. The door attributes are displayed.*

4. Edit the desired entries.

5.5 Defining acceleration and deceleration behavior

To account for differences in the driving behavior of several drivers and different vehicle properties during acceleration and deceleration, Vissim uses functions instead of individual acceleration or deceleration data.

Acceleration and deceleration are functions of the current speed. Thereby it is taken into account that combustion engines reach their maximum acceleration at lower speeds, and AC motors of trams and trains constantly accelerate over a large speed range.

In Vissim there are four types of functions: two acceleration functions and two deceleration functions. These are illustrated as curves:

- **Maximum acceleration**: max. acceleration technically possible. It is used to keep a certain speed on slopes, i.e. when stronger acceleration is required. The maximum acceleration is automatically adjusted for up and down gradients of links (see "Stochastic distribution of values for maximum acceleration and deceleration" on page 232):
  - by -0.1 m/s² per gradient percent incline
  - by 0.1 m/s² per gradient percent downgrade
- **Desired acceleration**: used in all situations, in which maximum acceleration is not required.
- **Maximum deceleration**: max. deceleration technically possible. As deceleration values have a negative algebraic sign, the maximum deceleration is the smallest acceleration value. Not even the desired deceleration can fall below it. Example: If the maximum deceleration is -5 m/s², the desired deceleration cannot be - 6m/s². The maximum deceleration is automatically adjusted for up and down gradients of links and connectors:
  - by -0.1 m/s² per gradient percent incline
  - by 0.1 m/s² per gradient percent downgrade
- **Desired deceleration**: Is used as the upper bound of deceleration in the following cases. Thereby maximum deceleration is not exceeded.
  - based on a desired speed decision
  - when approaching a red light

- when closing up to a preceding vehicle, e.g. during stop-and-go traffic
- in case of insufficient side clearance when overtaking on the same lane
- when approaching an emergency stop on connectors of routes
- for co-operative braking. Thereby 50% of the vehicle´s desired deceleration are used as the max. reasonable deceleration to decide whether an indicating vehicle may change from the neighboring lane to the vehicle´s lane.

You can assign acceleration and deceleration functions to the vehicle types of your choice. In all other situations, the parameters of the car-following model are relevant.

Desired acceleration, maximum acceleration, desired deceleration and maximum deceleration of a vehicle, driving at a certain speed, lie within a certain range between a maximum and a minimum value. For each of these four functions, you can show the maximum-minimum range in a graph for the median and limiting graphs for the upper and lower threshold values (see "Defining acceleration and deceleration functions" on page 233). The limiting graphs define the bandwidth. The median graph shows intermediate points as red circles that allow you to edit the median course. The limiting graphs show the intermediate points in green.

Modifying data points during a simulation run is possible only via the COM method **ReplaceAll**, which replaces all existing data points of the function with those included in the command call of the method. Individual data points cannot be changed during a simulation run.

Note: Vissim provides default acceleration and deceleration functions for vehicle types typically used in Western Europe.

5.5.1 Default curves for maximum acceleration and deceleration

The functions for maximum acceleration, provided in Vissim for passenger cars, correspond approximately to those established in the traffic flow model Wiedemann 74 (see "Driving states in the traffic flow model according to Wiedemann" on page 285).

- For cars, these measurements which were performed in Germany before 1974 have been slightly adapted for shorter time steps with jerk limitation and for the user-definable range (minimum-maximum).

  Jerk is the derivative of acceleration; that is, the change of acceleration with respect to time. With more than two time steps per second, it is limited by the share that corresponds with twice the duration of time step.

  Example: With ten time steps per second (time step = 0.1 s), the limit is 20% (0.2) of the intended change in acceleration.

  The data for the acceleration from a standstill have been validated against the test vehicle data gathered in the 2004 European research project RoTraNoMo.

- For HGV, the acceleration/deceleration curves have been adapted to data from the European research project CHAUFFEUR 2 in 1999.

- For trams and buses, the acceleration/deceleration curves have been set according to information from the Karlsruhe Transport Authority (VBK), 1995.

Note: All functions should be adapted to local conditions. This applies especially to your vehicle fleet data, if these are substantially different from Western European data.

5.5.2 Stochastic distribution of values for maximum acceleration and deceleration

For all vehicles, maximum acceleration is affected by gradients:

- Maximum acceleration is reduced by 0.1 m/s² per 1% upward gradient.
- Maximum acceleration is increased by 0.1 m/s² per 1% downward gradient.

For HGV vehicles as well, the actual acceleration is limited by the desired acceleration function. This is why for HGV vehicles high values for maximum acceleration are only relevant at very low speeds and with steep gradients.

A vehicle's maximum acceleration at a certain speed lies within a maximum and a minimum value. You can show the maximum-minimum range in a graph for the median and limiting graphs for the upper and lower threshold values (see "Defining acceleration and deceleration functions" on page 233). The limiting graphs define the bandwidth. The median graph shows intermediate points as red circles that allow you to edit the median course. The limiting graphs show the intermediate points in green. The exact position within this range depends on the following parameters:

  - For the maximum acceleration of vehicles of a vehicle type of the category HGV, on power and weight (edit functions and distributions of a vehicle type).
  - For maximum acceleration of all other vehicles, on a random value The random value is normally distributed with an average value of 0.5 and a standard deviation of 0.15, but is limited to [0..1]. So the distance between the median and the min/max curves is 3.333 times the standard deviation (SD).
  - For desired acceleration and deceleration, the gradient is not relevant.

  As a result:

- Approx. 70% of the vehicles are within the inner third (-1 SD to + 1 SD) of their random value.
- 95% are within the inner two thirds (-2 SD to + 2 SD).

Linear interpolation in Vissim

- For random values under 0.5, Vissim interpolates between the minimum value (0.0) and the median (0.5).
- For random values above 0.5, Vissim interpolates between the median and the maximum value (1.0).

Random values are not used for HGV vehicles. Instead, the power/weight ratio is taken into account (see "Editing functions and distributions of a vehicle type" on page 271). In metric units, the minimum value is 7 kW/ton and the maximum is 30 kW/ton. This means the average is 18.5 kW/ton. Accordingly, the following applies:

- For all HGV with a power/weight ratio of 7 or less, the minimum curve is used.
- For all HGV with a power/weight ratio of 30 or more, the maximum curve is used.
- For all HGV with a power/weight ratio of 18.5, the median is used.
- For HGV with other values, linear interpolation is performed.

Example linear interpolation for maximum acceleration

Speed	**40 km/h**
smallest value	1m/s^2
greatest value	3.5 m/s^2
Median	2.2m/s^2
Random value	0.6

Linear interpolation between 0.5 and 1.0:

$((3.5-2.2) / (1.0-0.5)) \bullet (0.6-0.5) + 2.2 = 2.46$

After interpolation, the maximum acceleration is adapted depending on the gradient, as described further above.

Note: If the actual power/weight ratios lie outside this range for your vehicles, you need to use maximum acceleration curves (small spread) and separate vehicles for these values.

5.5.3 Defining acceleration and deceleration functions

You can insert, select and edit acceleration and deceleration functions.

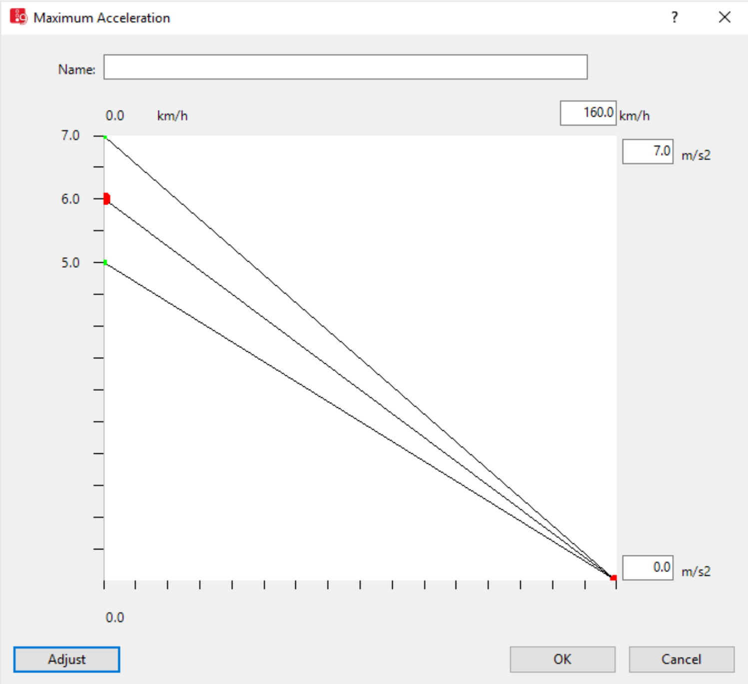
1. Select from the menu **Base Data** > **Functions**.
2. Select the desired entry:
   - **Maximum Acceleration**
   - **Desired Acceleration**
   - **Maximum Deceleration**
   - **Desired Deceleration**

   The list of defined objects for the base data type opens.

   By default, you can edit the list (see "Using lists" on page 93).

3. In the list, on the toolbar, click the **Add** button ✚.

   The window for the chosen function opens.



4. Make the desired changes:

Element	Description
Horizontal abscissa	Speed
Vertical ordinate	Acceleration value
Three curves	describe the stochastic distribution ➤ Median (red disks) ➤ Two boundary curves define the bandwidth (green disks)

Vissim uses the units which are selected by default (see "Selecting network settings for units" on page 205).

5. Make the desired changes:

Element	Description
Name	Name of function
km/h	Max. speed of desired speed range
m/s^2	➤ top box: maximum acceleration ➤ bottom box: minimum acceleration
Adjust	Show curve progression for entire value range

6. You can change the settings if you wish.

Element	Description
Insert intermediate point	Right-click on the desired position
Move intermediate point	Click intermediate point and drag with the mouse. Progression of the three curves changes in the value range between the limiting intermediate points.
Delete intermediate point	Click the intermediate point and move it to the adjacent start or end point.

7. Confirm with **OK**.

You can show the attributes of the acceleration and deceleration functions in the respective attributes list (see "Attributes of acceleration and deceleration functions" on page 235).

5.5.4 Attributes of acceleration and deceleration functions

1. Select from the menu **Base Data** > **Functions**.
2. Select the desired entry:
   - Maximum Acceleration
   - Desired Acceleration
   - Maximum Deceleration
   - Desired Deceleration

   The list of distributions for the type selected opens.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Element	Description
No	Number of acceleration or deceleration function
Name	Name of acceleration or deceleration function

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Acceleration function data points**
- **Deceleration function data points**

  Individual data point values of the functions are:

  - **x**-coordinate and **y**-coordinate for the median of distribution. Data points are shown in red in the window of the respective function.
  - **yMin**: Minimum speed of function at data point of **x**-coordinate and **y**-coordinate
  - **yMax**: Maximum speed of function at data point of **x**-coordinate and **y**-coordinate

    The data points of the minimum and maximum values are shown in green in the window of the respective function.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

5.5.5 Deleting the acceleration/deceleration function

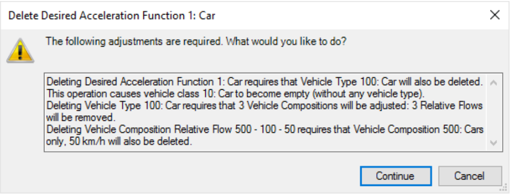
1. Select from the menu **Base Data** > **Functions**.
2. Select the desired acceleration or deceleration function.

   The list of defined network objects for the network object type opens.

3. Right-click on the row number of the desired row.

4. From the shortcut menu, choose **Delete**.

   A warning is issued.

5. Confirm with **Yes**.

Use this function to delete every vehicle type assigned to this function. Thus the related vehicle categories cannot include any vehicle type anymore. The query window closes. The deleted curve is no longer displayed.

5.6 Using distributions

There are various use cases for stochastic distributions in Vissim, for example the desired speed of vehicles on certain network objects or the weight and power of HGVs. Using Vissim, you can model any type of stochastic distribution.

You can also define general distributions in which the value range and usage are user-defined (see "Using general distributions" on page 257).

5.6.1 Using desired speed distributions

The distribution function of desired speeds is a particularly important parameter, as it has an impact on link capacity and achievable travel times. If not hindered by other vehicles or network objects, e.g. signal controls, a driver will travel at his desired speed. This applies accordingly to pedestrians.

A driver, whose desired speed is higher than his current speed, will check whether he can overtake other vehicles without endangering anyone. The more drivers' desired speed differs, the more platoons are created.

Desired speed distributions are defined independently of vehicle type or pedestrian type.

You may use desired speed distributions for vehicle compositions, pedestrian compositions, reduced speed zones, desired speed decisions, PT lines and parking lots.

For pedestrian simulation with Viswalk you can use desired speed distributions described in the specialized literature (see "Using desired speed distributions for pedestrians" on page 873).

Note: Vissim provides typical default values for desired speed distributions.

5.6.1.1 Defining desired speed distributions

You can define new desired speed distributions and add intermediate points as spline points to the course of the curve. In general, two nodes are sufficient to achieve more or less an S-shaped distribution, and thus a concentration around the mean value.

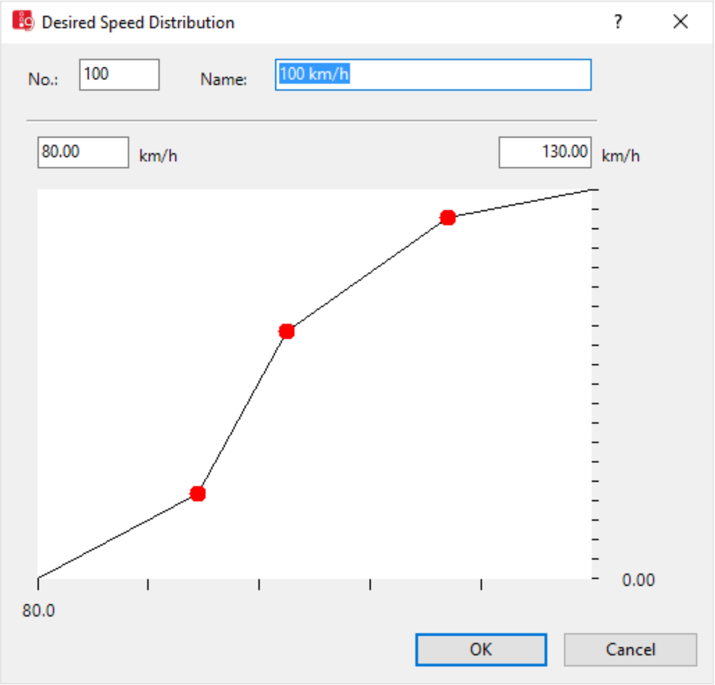
1. Choose from the menu **Base Data** > **Distributions** > **Desired Speed**.

   *The **Desired Speed Distributions** list opens.*

2. In the list, on the toolbar, click the **Add** button ✚.

   A new row with default data is inserted.

   *The **Desired Speed Distribution** window opens.*



3. Make the desired changes:

Element	Description
Name	Description
Left field with speed unit	Minimum desired speed
Right field with speed unit	Maximum desired speed

4. Right-click in the line.

   A node is entered.

5. Click the intermediate point and keep the mouse button pressed.
6. With the mouse button pressed, drag the intermediate point to the desired position.
7. Release the mouse button.

The horizontal axis depicts the desired speed. The vertical axis depicts the proportional value in the value range from 0.0-1.0. The course of the curve is adjusted. The labeling for the y and x axes is adjusted. The figure shows an example in which 22 % of the vehicles drive between 80.0 and 94.50 km/h and 78% of the vehicles between 94.50 km/h and 130.00 km/h (all equally distributed across the speed range). Due to the S-shaped distribution of the three intermediate points at 94.50 km/h, approx. 104 km/h and 116 km/h, a concentration around the mean value 105 km/h is achieved.

8. Confirm with **OK**.

*The desired speed distribution is displayed in the **Desired Speed Distributions** list (see "Attributes of desired speed distributions" on page 239).*

5.6.1.2 Attributes of desired speed distributions

1. Choose from the menu **Base Data** > **Distributions** > **Desired Speed**.

   *The **Desired Speed Distributions** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Element	Description
No	Number of desired speed distribution
Name	Name of desired speed distribution
Lower bound	Minimum desired speed
Upper bound	Maximum desired speed

By default, the **Units** currently set under **Network settings** are used (see "Selecting network settings for units" on page 205).

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values of speed distribution. For predefined distributions, at least two points are defined by default: Data point 1 for minimum desired speed and the last data point for maximum desired speed. You can insert additional data points between these data points. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).
  - **x**: Desired speed at data point in curve.
    - At data point 1: Minimum desired speed of selected desired speed distribution
    - Desired speed of additional data points, if defined for this curve

    - At last data point: Maximum desired speed of selected desired speed distribution
- **FX** (**f(x)**): Probability of desired speed **x** at data point
    - At data point 1: **0**
    - If additional data points are defined for the curve: Probability of desired speed **x** at data point
    - At last data point: **1**

  Speed distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

3. Enter the desired data.

The data is allocated.

5.6.1.3 Deleting the desired speed distribution

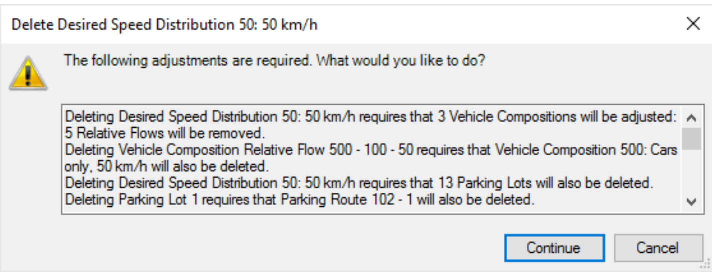
1. Choose from the menu **Base Data** > **Distributions** > **Desired Speed**.

   *The **Desired Speed Distributions** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

   *If the desired speed distributions are assigned to the network objects, the window **Delete desired speed distribution** opens.*



4. Select from the list box an appropriate desired speed distribution for the network objects you want to delete, to which the desired speed distribution was previously assigned.

5. Confirm with **OK**.

The window closes. The affected network objects are assigned the selected desired speed distribution.

5.6.2 Using power distributions

Power refers exclusively to vehicles of vehicle category **HGV** and is specified via power distribution. Power distribution is irrelevant for other vehicle categories. Power distribution is specified independently of the vehicle type. Vissim uses power distributions and weight distributions for **HGVs** (see "Using weight distributions" on page 243).

The probability increases up to the maximum power defined and then reaches the value 1.

Note: Vissim provides typical default power distributions.

5.6.2.1 Defining power distributions

You define the power through power distributions only for vehicles of **HGV** category. Power distribution is irrelevant for other vehicle categories. Vissim uses power distributions together with weight distributions (see "Using weight distributions" on page 243). Some typical power distributions are predefined.

1. From the **Base Data** menu, choose **Distributions** > **Power**.

   *The **Power Distributions** list opens.*

2. In the list, on the toolbar, click the **Add** button ✚.

   A new row with default data is inserted.

   *The **Power Distribution** window opens.*

3. Make the desired changes:

Element	**Description**
Name	Description
Left field with power unit	Minimum power (kW)
Right field with power unit	Maximum power (kW)

4. Right-click in the line.

   A node is entered.

5. Click the intermediate point and keep the mouse button pressed.
6. With the mouse button pressed, drag the intermediate point to the desired position.
7. Release the mouse button.

The labeling for the y and x axes is adjusted.

8. Confirm with **OK**.

*The power distribution will be shown in the **Power Distributions** list.*

5.6.2.2 Attributes of power distributions

1. From the **Base Data** menu, choose **Distributions** > **Power**.

   *The **Power Distributions** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Element	Description
No	Number of power distribution
Name	Name of the power distribution
Lower bound	Minimum power (kW)
Upper bound	Maximum power (kW)

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values for power distribution. For predefined distributions, two points are defined by default: Data point 1 for minimum power and data point 2 for maximum power. You can insert additional data points between these data points. These data points are then displayed as intermediate points in the curve(see "Editing the graph of a function or distribution" on page 265).
  - **x**: Power (kW) at data point in curve
    - At data point 1: Minimum power of selected power distribution
    - Power of additional data points, if defined for the curve
    - At last data point: Maximum power of selected power distribution
  - **FX** (**f(x)**): Probability of power **x** at data point
    - At data point 1: **0**
    - If additional data points are defined for the curve: Probability of power **x** at data point
    - At last data point: **1**

  Power distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.
3. Enter the desired data.

The data is allocated.

5.6.2.3 Deleting the power distribution

1. From the **Base Data** menu, choose **Distributions** > **Power**.

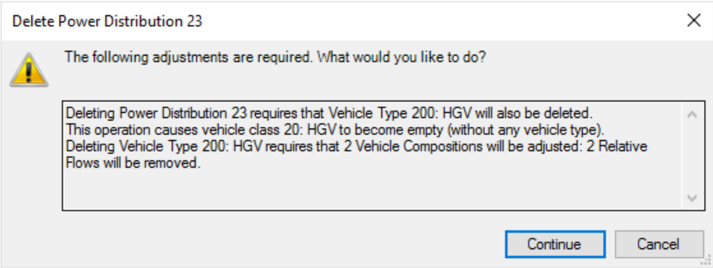
   *The **Power Distributions** list opens.*

2. Right-click the entry of your choice.

Warning: When you delete a power distribution, all vehicles types assigned to it are deleted!

3. From the shortcut menu, choose **Delete**.

   If the power distribution has been assigned vehicle types, a window opens.



4. Click **Delete**.

5.6.3 Using weight distributions

The weight of vehicles of category **HGV** is defined via weight distributions. Weight distributions are irrelevant for other vehicle categories.

You need to assign each vehicle type to a vehicle category. From the weight and power distribution data assigned, Vissim randomly selects a value for each vehicle with a vehicle type under vehicle category **HGV**. The weight and power distribution values are independent from each other. This means Vissim may assign high power to a low-weight vehicle. Using the weight and power data, Vissim calculates the specific power (in kW/t). The specific power is limited to a range between 7 and 30 kW/t, so that no unrealistic weight/power combinations are created. If a value < 7 kW/t is calculated, the specific power is set to 7 kW/t. For values exceeding 30 kW/t, the specific power is set to 30 kW/t.

The specific power has an impact on acceleration and deceleration behavior (see " Defining acceleration and deceleration behavior" on page 230). This is particularly important for links with gradients. Using the specific power, Vissim calculates the percentile used to select the relevant acceleration curve from the distribution of acceleration functions.

Weight distributions are defined independently from the vehicle type.

The probability increases up to the maximum weight defined, reaching the value 1.

Note: Vissim provides typical default values for desired distributions.

5.6.3.1 Defining weight distributions

1. Select from the menu **Base Data** > **Distributions** > **Weight**.

   *The **Weight Distributions** list opens.*

By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button ✚ .
3. Right-click the row header.

4. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

   The ***Weight Distribution*** *window opens.*
5. Make the desired changes:

Element	Description
Name	Description
Left field with weight unit	Minimum weight
Right field with weight unit	Maximum weight

6. Right-click in the line.

   A node is entered.
7. Click the intermediate point and keep the mouse button pressed.
8. With the mouse button pressed, drag the intermediate point to the desired position.
9. Release the mouse button.

The labeling for the y and x axes is adjusted.

10. Confirm with **OK**.

The weight distribution will be shown in the ***Weight Distributions*** *list (see " Attributes of weight distributions" on page 244).*

5.6.3.2 Attributes of weight distributions

1. From the **Base Data** menu, choose > **Distributions** > **Weight**.

   The ***Weight Distributions*** *list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Element	Description
No	Number of weight distribution
Name	Name of the weight distribution

Element	Description
Lower bound	Minimum weight (kg)
Upper bound	Maximum weight (kg)

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values for weight distribution. By default, two points are defined. The curve of weight distribution **Dreyfuss F** also contains an intermediate point.

  Data point 1 for minimum weight and the last data point for maximum weight. You can insert additional data points between these data points. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).

  - **x**: Weight (kg) at data point in curve.
    - At data point 1: Minimum weight of minimum weight distribution selected
    - Weight of additional data points, if defined for the curve
    - At last data point: Maximum weight of weight distribution selected
  - **FX** (**f(x)**): Value for probable weight **x** at data point
    - At data point 1: **0**
    - If additional data points are defined for the curve: Probability of weight at data point **x**
    - At last data point: **1**

    Weight distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

3. Enter the desired data.

The data is allocated.

5.6.3.3 Deleting the weight distribution

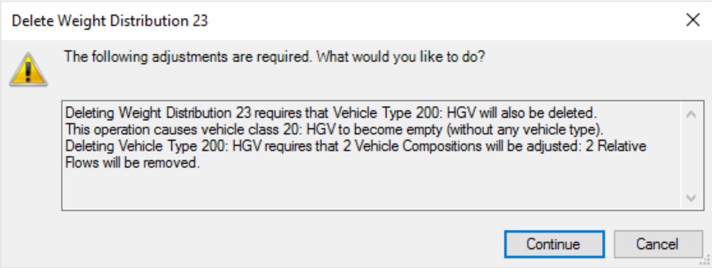
1. Select from the menu **Base Data** > **Distributions** > **Weight**.

   *The **Weight Distributions** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

   *If the weight distribution is assigned to the network objects, the **Delete weight distribution** window opens.*

4. Select from the list box an appropriate weight distribution for the objects you want to delete, to which the weight distribution was previously assigned.
5. Confirm with **OK**.

The window closes. The affected network objects are assigned the selected weight distribution.

5.6.4 Using time distributions

You can use dwell time distributions for:

- Standstill times on parking lots, which you specify for routing decisions of the type **Parking Lot** per time interval (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459)
- Waiting times at toll counters through stop signs, such as managed lanes and border crossings
- PT stops: For PT vehicles, such as bus or tram, you thereby give in this way the time required to allow passengers to board and alight. If the method for calculating the board- ing and alighting times is not used, the dwell time distribution in Vissim must be assigned to every PT stop or railway station.
- Set the time a vehicle remains in standstill, between backing out of a parking space and driving forward, after it has left the parking lot. This time period is defined by the attribute **Direction change duration distribution** of the parking lot (see "Attributes of parking lots" on page 500). Default value: time distribution **5 s** with lower bound 0.00, upper bound 15.00, standard deviation 1.00, average value 5.00.

The probability increases up to the maximum dwell time defined and then reaches the value 1.

5.6.4.1 Defining time distributions

1. Choose from the menu **Base Data** > **Distributions** > **Time**.

   *The **Time Distributions** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the row header.

3. From the shortcut menu, choose **Add**.

4. Choose the desired entry from the context menu.
   - Empirical Distribution
   - Normal Distribution

   A new row with default data is inserted.

5. Enter the desired values.

   You can edit the curve of an empirical distribution.

6. If you would like to edit the empirical distribution of the time distribution in a graph, double-click the entry.

   The ***Time Distribution*** *window opens.*

7. Make the desired changes:

Element	Description
Name	Description
Left field with time unit s	Minimum duration in seconds
Right field with time unit s	Maximum duration in seconds

If you are using public transport dwell time to model public transport, a standard deviation = 0 s will result in a constant public transport dwell time (see "Calculating the public transport dwell time for PT lines and partial PT routes" on page 531).

8. Right-click in the line.

   A node is entered.

9. Click the intermediate point and keep the mouse button pressed.

10. With the mouse button pressed, drag the intermediate point to the desired position.

11. Release the mouse button.

The labeling for the y and x axes is adjusted.

12. Confirm with **OK**.

The time distribution will be shown in the ***Time Distributions*** *list* *(see "Attributes of time distributions" on page 247)**.*

5.6.4.2 Attributes of time distributions

1. From the **Base Data** menu, choose > **Distributions** > **Time**.

   The ***Time Distributions*** *list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Long name	Short name	Description
Number	**No**	Number of time distribution
Name	**Name**	Name of the time distribution
Type	**Type**	Type of distribution function: • **Empirical**: Cumulative frequency function. Definition between lower limit and upper limit, with intermediate points in the curve • **Normal**: Normal distribution. Definition between lower limit and upper limit, with **mean value** and **standard deviation**
Lower bound	**Lower bound**	Minimum duration in seconds
Upper bound	**Upper bound**	Maximum duration in seconds
Standard deviation	**StdDev**	Can only be changed for normal distribution: Dispersion of values between lower bound and upper bound
Mean	**Mean**	Can only be changed for normal distribution: Mean of values between lower bound and upper bound

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values of time distribution. You can insert data points for an empirical distribution. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).
  - **x**: Dwell time (s) at data point in curve.
    - At data point 1: Minimum dwell time of selected time distribution
    - Dwell time of additional data points, if defined for the curve of empirical distribution
    - At last data point: Maximum dwell time of selected time distribution
  - **FX** (**f(x)**): Probability of dwell time **x** at data point
    - At data point 1: **0**

- If, for an empirical distribution, additional data points are defined for the curve: Values for probability of dwell time at data point **x**
- At last data point: **1**

Dwell time distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

3. Enter the desired data.

The data is allocated.

5.6.4.3 Deleting the time distribution

1. Choose from the menu **Base Data** > **Distributions** > **Time**.

   *The **Time Distributions** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

4. Confirm with **OK**.

5.6.5 Using location distributions for boarding and alighting passengers in PT

A location distribution allows you to define how the total number of boarding/alighting passengers is distributed over the entire length of the PT vehicle. For each door of a PT vehicle that is meant for boarding and alighting passengers, the share of the vehicle length on both its sides is calculated:

- half the distance to the next door and/or
- the entire distance to the start or end of the vehicle

For each share of the total vehicle length, an increase in y direction is shown on the x-axis as a percentage of passengers for the respective door. The probability increases from NULL at the very front of the vehicle to 1 at the very back.

5.6.5.1 Defining location distributions for boarding and alighting passengers in PT

Some typical location distributions for boarding and alighting passengers in PT vehicles are predefined:

Element	Description
Uniform	Linear distribution over the full length
Center	More boarding and alighting passengers in the middle
Front	More boarding and alighting passengers at the front
Rear	More boarding and alighting passengers at the back
Front and rear	Less boarding and alighting passengers in the middle

1. Choose from the menu **Base Data** > **Distributions** > **Location**.

*The **Location Distributions** list opens.*

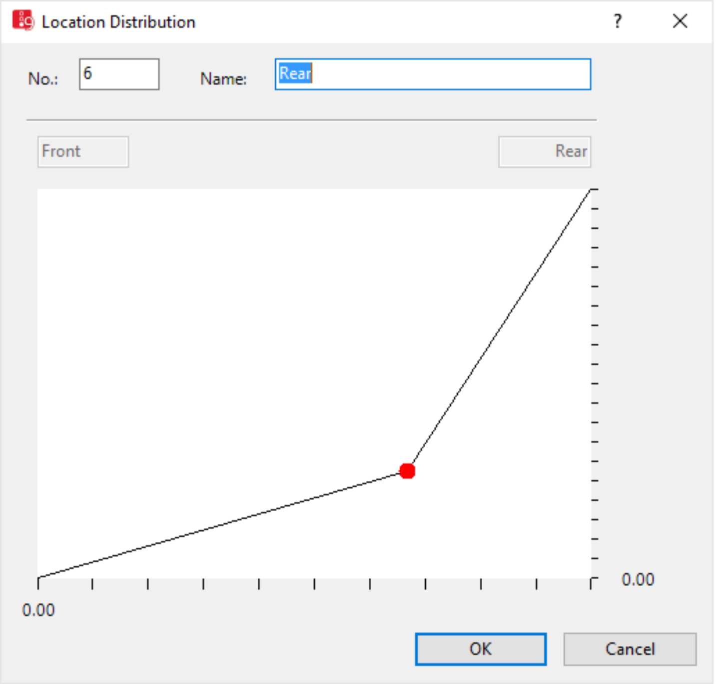
By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button  .

   A new row with default data is inserted.

   *The **Location Distribution** window opens. The x-axis defines the position:*

   - 0.00: maximum towards the front
   - 1.00: maximum towards the rear



3. Enter the desired name.
4. Right-click in the line.

   A node is entered.

5. Click the intermediate point and keep the mouse button pressed.
6. With the mouse button pressed, drag the intermediate point to the desired position.
7. Release the mouse button.

The labeling for the y and x axes is adjusted.

8. Confirm with **OK**.

*The location distribution will be shown in the **Location Distributions** list (see "Attributes of location distributions" on page 251).*

5.6.5.2 Attributes of location distributions

1. From the **Base Data**, choose > **Distributions** > **Location**.

   *The **Location Distributions** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Long name	Short name	Description
Number	**No**	Number of location distribution
Name	**Name**	Name of the location distribution
Lower bound	**Lower bound**	Fixed value 0.00: maximum distance front
Upper bound	**Upper bound**	Fixed value 0.00: maximum distance rear

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values of location distribution. Except for the predefined location distribution **Equally distributed**, predefined distributions, by default, contain several defined data points in order to steer boarding and alighting passengers towards the front or rear of the PT vehicle. You can insert additional data points between these data points. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).
  - **X**:Focal point of boarding and alighting passengers at data point in the curve (value range 0.00 to 1.00)
    - At data point 1: for passengers boarding and alighting at the very front of the PT vehicle
    - Values of additional data points, if defined for this curve
    - At the last data point: for passengers boarding and alighting at the very end of the PT vehicle
  - **FX** (**f(x)**): Probability of passengers boarding and alighting **x** at data point

- At data point 1: **0**
- If additional data points are defined for the curve: Probability of passengers boarding and alighting **x** at data point
- At last data point: **1**

Location distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

3. Enter the desired data.

The data is allocated.

5.6.5.3 Selecting alighting location distribution and boarding location distribution

Selecting alighting location distribution

A PT line with a PT stop has to be defined. When a PT line is selected, by default, the PT stop is displayed in red.

1. Double-click the PT stop.

   *The **PT Line Stop** window opens. In the **Alighting location** list, the default value is **No distribution**: Alighting passengers are distributed equally to all doors regardless of their position.*

2. Select the desired entry.

Selecting boarding location distribution

You can choose the boarding location distribution for every pedestrian area with PT usage through the option **Boarding location** (see "Modeling construction elements" on page 880).

The standard value is the **Nearest door**: A boarding passenger selects the door that is accessible on the shortest path from his location.

5.6.6 Using distance distributions

Using distance distributions, you can define the distribution between a point and a maximum distance. To do so, you enter a maximum distance (default value 100 m). The minimum limit 0 m cannot be edited.

If you are using external software to simulate the communication between vehicles or the communication between vehicles and suitable roadside infrastructure, you can exchange data with Vissim via the COM interface. This type of data includes the probability of possible data loss when the sending vehicle is at a certain distance. If the distance to the sending vehicle (world coordinates) is NULL, the probability is NULL. The probability increases up to the maximum distance and then reaches the value 1.

5.6.6.1 Defining distance distributions

1. From the **Base Data** menu, choose **Distributions** > **Distance**.

   *The **Distance Distributions** list opens.*

By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button ✚ .

   A new row with default data is inserted.

   *The **Distance Distribution** window opens.*

3. Make the desired changes:

Element	Description
Name	Description
Left field with distance unit	Minimum distance 0.0: No change possible
Right field with distance unit	Maximum distance, default value 100 m

4. Right-click in the line.

   A node is entered.

5. Click the intermediate point and keep the mouse button pressed.
6. With the mouse button pressed, drag the intermediate point to the desired position.
7. Release the mouse button.

The labeling for the y and x axes is adjusted.

8. Confirm with **OK**.

*The distance distribution is shown in the **Distance Distributions** list (see "Attributes of distance distributions" on page 253).*

5.6.6.2 Attributes of distance distributions

1. From the **Base Data** menu, choose **Distributions** > **Distance**.

   *The **Distance Distributions** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Long name	Short name	Description
Number	**No**	Number of distance distribution
Name	**Name**	Name of distance distribution
Lower bound	**Lower bound**	Minimum distance to assigned object
Upper bound	**Upper bound**	Maximum distance to assigned object

By default, the **Units** currently set under **Network settings** are used (see "Selecting network settings for units" on page 205).

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Data points**.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values for distance distribution. Distance distribution is not defined by default. When you define a distance distribution, Vissim by default defines the lower limit = 0.0 m and the upper limit = 100.0 m. You can insert additional data points between these two data points. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).
  - **x**: Distance at data point in curve.
    - At data point 1: Minimum distance of selected distance distribution
    - Distance of additional data points, if defined for the curve
    - At last data point: Maximum distance of selected distance distribution
  - **FX** (**f(x)**): Probability of distance **x** at data point
    - At data point 1: **0**
    - If additional data points are defined for the curve: Probability of distance **x** at data point
    - At last data point: **1**

    Distance distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

3. Enter the desired data.

The data is allocated.

5.6.6.3 Deleting the distance distribution

1. From the **Base Data** menu, choose > **Distributions** > **Distance**.

   *The **Distance Distributions** list opens.*
2. Right-click the entry of your choice.
3. From the shortcut menu, choose **Delete**.
4. Confirm with **OK**.

5.6.7 Using occupation distributions

By defining an occupancy distribution, you specify how the total number of occupants of vehicles are distributed across the vehicles of a certain vehicle type. You can assign an occupancy distribution to each vehicle type.

5.6.7.1 Defining occupancy distributions

1. From the **Base Data** menu, choose **Distributions** > **Occupancy**.

   *The **Occupancy Distributions** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the row header.

3. From the shortcut menu, choose **Add**.

4. Choose the desired entry from the context menu.

   - Empirical distribution: Can generate the value NULL, for example for empty runs of autonomous vehicles
   - Normal Distribution

   A new row with default data is inserted.

5. Enter the desired values.

   You can edit the curve of an empirical distribution.

6. If you would like to edit the empirical distribution of the occupancy distribution, double-click on the entry.

   *The **Occupancy Distribution** window opens.*

7. Make the desired changes:

Element	Description
Name	Designation
Left field	Minimum number of occupants
Right field	Maximum number of occupants

8. Right-click in the line.

   A node is entered.

9. Click the intermediate point and keep the mouse button pressed.

10. With the mouse button pressed, drag the intermediate point to the desired position.

11. Release the mouse button.

The labeling for the y and x axes is adjusted.

12. Confirm with **OK**.

The occupancy distribution is shown in the ***Occupancy Distributions*** *list (see "Attributes of occupancy distributions" on page 256).*

5.6.7.2 Attributes of occupancy distributions

1. From the **Base Data** menu, choose **Distributions** > **Occupancy**.

   The ***Occupancy Distributions*** *list opens.*

> Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Long name	Short name	Description
Number	**No**	Number of occupancy distribution
Name	**Name**	Name of the occupancy distribution
Type	**Type**	Type of distribution function: • **Empirical**: Cumulative frequency function. Definition between lower limit and upper limit, with intermediate points in the curve • **Normal**: Normal distribution. Definition between lower limit and upper limit, with **mean value** and **standard deviation**
Lower bound	**Lower bound**	Minimum occupation of vehicle
Upper bound	**Upper bound**	Maximum occupation of vehicle
Standard deviation	**StdDev**	Can only be changed for normal distribution: Dispersion of values between lower bound and upper bound
Mean	**Mean**	Can only be changed for normal distribution: Mean of values between lower bound and upper bound

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values for occupancy distribution. By default, no point is defined for the predefined distribution **Individual**, of the type **Normal**. You can insert data points for an empirical distribution. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).

- **x**: Occupancy at data point in curve
  - At data point 1: Minimum occupancy of selected occupancy distribution
  - Occupancy of additional data points, if defined for the curve
  - At last data point: Maximum occupancy of selected occupancy distribution
- **FX** (**f(x)**): Value for probable occupancy **x** at data point
  - At data point 1: **0**
  - If, for an empirical distribution, additional data points are defined for the curve: Probability of occupancy at data point **x**
  - At last data point: **1**

  Occupancy distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

2. On the list toolbar, in the **Relations** list, click the desired entry.
3. Enter the desired data.

The data is allocated.

5.6.7.3 Deleting the occupancy distribution

1. From the **Base Data** menu, choose **Distributions** > **Occupancy**.

   *The **Occupancy Distributions** list opens.*
2. Right-click the entry of your choice.
3. From the shortcut menu, choose **Delete**.
4. Confirm with **OK**.

5.6.8 Using general distributions

A general distribution allows you to define an empirical distribution or a normal distribution, with values >= 0. For example, you can access a value of a user-defined general distribution via a vehicle attribute decision or a pedestrian attribute decision, if for the network object of these network object types, in the **Decision type** attribute, **Distribution** has been selected (see "Defining vehicle attribute decisions" on page 506), (see "Defining Pedestrian Attribute Decisions" on page 966).

5.6.8.1 Defining general distributions

1. From the **Base Data** menu, choose > **Distributions** > **General**.

   *The **General Distributions** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the row header.

3. From the shortcut menu, choose **Add**.

4. Choose the desired entry from the context menu.

   - Empirical Distribution
   - Normal Distribution

   A new row with default data is inserted.

5. Enter the desired values.

   You can edit the curve of an empirical distribution.

6. To edit the empirical distribution of the general distribution, double-click the entry.

   *The **General Distribution** window opens.*

7. Make the desired changes:

Element	Description
Name	Designation
Left field	Minimum number
Right field	Maximum number

8. Right-click in the line.

   A node is entered.

9. Click the intermediate point and keep the mouse button pressed.

10. With the mouse button pressed, drag the intermediate point to the desired position.

11. Release the mouse button.

The labeling for the y and x axes is adjusted.

12. Confirm with **OK**.

*The general distribution is displayed in the **General Distributions** list (see "Attributes of general distributions" on page 258).*

5.6.8.2 Attributes of general distributions

1. From the **Base Data** menu, choose > **Distributions** > **General**.

   *The **General Distributions** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Long name	Short name	Description
Number	**No.**	Number of general distribution
Name	**Name**	Name of the general distribution
Type	**Type**	Type of general distribution: - **Empirical**: Cumulative frequency function. Definition between lower limit and upper limit, with intermediate points in the curve - **Normal**: Normal distribution. Definition between lower limit and upper limit, with **mean value** and **standard deviation**
Lower bound	**Lower bound**	Minimum value
Upper bound	**Upper bound**	Maximum value
Standard deviation	**StdDev**	Can only be changed for normal distribution: Dispersion of values between lower bound and upper bound
Mean	**Mean**	Can only be changed for normal distribution: Mean of values between lower bound and upper bound

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **Data points**: Individual data point values of general distribution. You can insert data points for an empirical distribution. These data points are then displayed as intermediate points in the curve (see "Editing the graph of a function or distribution" on page 265).
  - **x**: Value at data point in curve
    - At first data point: minimum value of general distribution selected
    - Values of additional data points, if defined for this curve
    - At last data point: maximum value of general distribution selected
  - **FX** (**f(x)**): Value for probability of value **x** at data point

    A general distribution is monotonically increasing. This is why each **FX** value must be greater than or equal to its preceding value.

If the type of distribution is **empirical**, you can edit data points in the list toolbar, in the list box **Relation data points**:

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

5.6.8.3 Deleting general distribution

1. From the **Base Data** menu, choose > **Distributions** > **General**.

   *The **General Distributions** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

4. Confirm with **OK**.

5.6.9 Using 2D/3D model distributions

You can use a 2D/3D model distribution instead of a single model for vehicles or pedestrians of a given type.

If you want to use only one model for all objects of a type, you must still define a 2D/3D model distribution. You assign only one model to this 2D/3D model distribution.

In a 2D/3D model distribution you can use non-zero proportions to assign the desired 2D/3D models. The absolute share of Vissim is calculated automatically as a ratio of the individual relative share to the sum of all shares.

You can define a vehicle or pedestrian model by selecting a 3D model file. This automatically defines all geometry data, such as the length and width or the positions of axles and clutches. If you do not select a 3D model file, you can enter the geometry data manually. Models of this sort are shown as blocks in the network.

Some typical 2D/3D model distributions are predefined.

5.6.9.1 Defining 2D/3D model distributions for 2D/3D models

1. From the **Base Data** menu, choose **Distributions** > **2D/3D Model**.

   *The **2D/3D Model Distributions** list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Into the list on the left, enter the desired number and name.
4. On the list toolbar, in the **Relations** list, click **2D/3D model distribution elements**.

   In the next steps you select a new model distribution element from the right list and assign it to the selected object in the left list. For each object in the left list, you can insert rows with model distribution elements in the right list and so assign it.

   A new row with default data is inserted.

5. Make the desired changes:

Element	Meaning
Share	2D/3D model distribution share, by default 0.1
2D/3D model	➤ Select a predefined model ➤ In the list box of the cell, click the **Add** button to open the **Select 3D Model** window (see "Defining 2D/3D models" on page 220).

The data is allocated.

6. If you want to assign other 2D/3D model distribution elements, right-click the row header in the right list.
7. From the shortcut menu, choose **Add**.
8. Select the desired entry.
9. Confirm with **OK**.

Editing an assigned 2D/3D model

1. If you want to edit the attributes of an assigned 2D/3D model, click the model distribution element entry in the right list.
2. From the context menu, choose **Edit 2D/3D Models**.

   The ***2D/3D Models*** *list opens. The selected* ***2D/3D model*** *is automatically highlighted.*

3. Enter the desired values.

5.6.9.2 Attributes of 2D/3D model distributions

1. From the **Base Data** menu, choose **Distributions** > **2D/3D Models**.

   The ***2D/3D Model Distributions*** *list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Long name	Short name	Description
Number	**No**	Number of 2D/3D model distribution
Name	**Name**	Name of 2D/3D model distribution

Showing and editing dependent objects as relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, select **2D/3D model segments** in the **Relations** list box.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

- **2D/3D model distribution elements**: Proportion and name of 2D/3D model assigned to selected model distribution element
  - **Proportion**: Relative proportion of 2D/3D model (or model segment) in total value 1 of 2D/3D model distribution.
  - **2D/3D model** (**Model2D3D**): Name of 2D/3D model (see "Defining 2D/3D models" on page 220). This may be a model without a file or a model based on a **.v3d* file.

3. Enter the desired data.

The data is allocated.

5.6.9.3 Deleting the 2D/3D model distribution

Warning: When you delete a 2D/3D model distribution, all vehicles types assigned to it are deleted!

1. From the **Base Data** menu, choose **Distributions** > **2D/3D Model**.

   *The **2D/3D Model Distributions** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

   If the 2D/3D model distribution has been assigned vehicle types, a window opens.

4. Click **Delete**.

5.6.10 Using color distributions

Color distributions are relevant only for the graphical display and have no influence on the simulation results.

The color distribution is used instead of a single color for visualization of a type of vehicle or pedestrian. You can define a maximum of 10 colors for color distribution. By specifying a relative share, you can specify the frequency of occurrence of each color. The absolute share of Vissim is calculated automatically as a ratio of the individual relative share to the sum of all shares.

If you want to use only one color for a type, you must still define a color distribution. You assign only the desired color to this color distribution.

Some typical color distributions are predefined.

5.6.10.1 Defining color distributions

1. Choose from the menu **Base Data** > **Distributions** > **Color**.

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button ✚.

A new row with default data is inserted.

3. Into the list on the left, enter the desired number and name.
4. On the list toolbar, select the **Color distribution elements** entry in the **Relations** list box.

In the next steps, set up new colors and assign them to the object selected in the left list. For each object in the left list, you can add rows with colors to the list on the right and define share and color distribution.

5. Right-click on the row header in the right-hand list.

A new row with default data is inserted.

6. Make the desired changes:

Element	Meaning
Color	Color and hexadecimal color code.
Share	Color distribution share, by default 0.1

7. If you would like to edit a color, double-click in the **Color** box.

A color selection window opens. There are various ways to select a color.

8. Select the desired color:

Element	Meaning
Color definition	Click on the desired color
RGB	To select the color, use the sliders to change the values for red, yellow and blue or enter values between 0 and 255
HSL	To select the color, use the sliders to change the color value (0 to 359), saturation (0 to 100) and relative brightness (0 to 100) or enter values
Hex	Enter the color as a hexadecimal value
Alpha	Use the slider to select the transparency or enter a value: 0 = transparent, 255 = opaque
Predefined colors	Click on the desired color. - **+** button: Adds the selected color to a user-defined pick list below the predefined colors. - **x** button: Removes the selected color from the user-defined pick list.
Screen color picker	Select color with the pipette: click in the area around the pipette, keep the mouse button pressed and drag the mouse arrow to the point on the screen whose color you would like to copy. Release the mouse button.
New	Preview of the selected color
Current	Currently assigned color

9. Click next to the window when you want to close it.

The data is allocated.

5.6.10.2 Deleting the color distribution

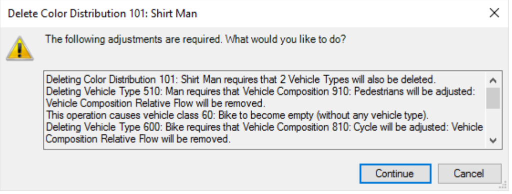
1. Choose from the menu **Base Data** > **Distributions** > **Color**.

   The ***Color Distributions*** *list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Delete**.

   A message is displayed if the color distribution is assigned to network objects.



4. Select from the list box an appropriate color distribution for the network objects you want to delete, to which the color distribution was previously assigned.

5. Confirm with **OK**.

The window closes. The affected network objects are assigned the selected color distribution.

5.6.11 Editing the graph of a function or distribution

You can edit the values of a distribution in the list of the distribution type concerned. For the following distributions, you can also open a window where you can define or move intermediate points:

- Desired speed
- Power
- Weight
- Location
- Distance
- Time: Only for empirical distributions
- Occupancy: Only for empirical distributions

1. Select the desired distribution from the menu **Base Data** > **Distributions**.

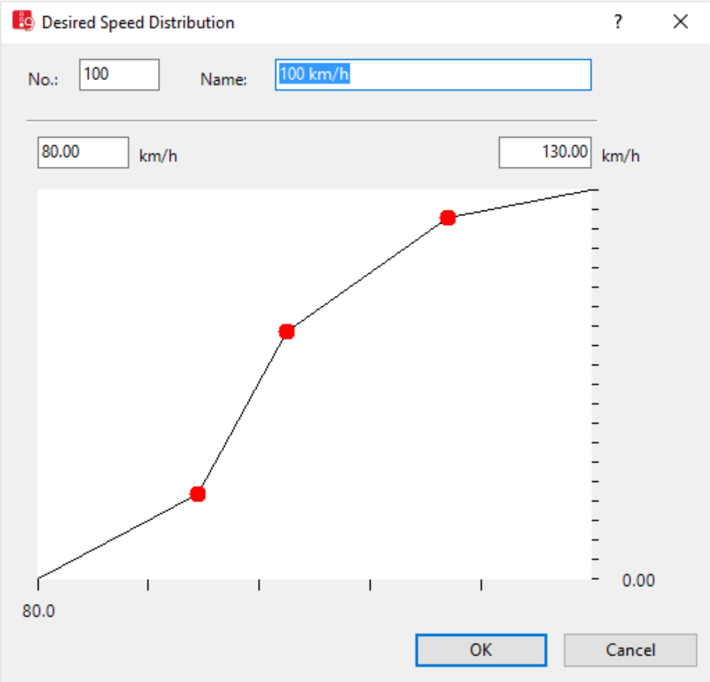
   The list of distributions of the selected distribution type opens.

2. Select the desired entry.

3. Edit the desired entries.

4. To edit the intermediate points in the distribution curve, double-click the desired distribution.

   *The **<Name> Distribution** window opens. Example desired speed distribution with three intermediate points in the curve:*



5. Make the desired changes:

Element	Description
Name	Name of distribution
Left-hand field with unit	Minimum value
Right-hand field with unit	Maximum value

6. To add an intermediate point, on the line, right-click the position of your choice.
7. To change the shape of the curve, click the desired intermediate point, hold the mouse button down and drag the intermediate point to the desired position.
8. Release the mouse button.

5.6.12 Deleting intermediate point of a graph

You may delete intermediate points for the following distributions:

- Desired speed distributions
- Power distributions
- Weight distributions
- Location distributions
- Distance distributions

1. Choose the desired distribution type from the menu **Base Data** > **Distributions**.

The list of distributions for the distribution type opens.

2. Double-click the desired entry.

   *The **<Name> of distribution** window opens.*

3. Click the intermediate point and keep the mouse button pressed.
4. Drag the intermediate point to another intermediate point.
5. If there is only one intermediate point and you want to delete it, drag the intermediate point out of the bottom left or top right corner.
6. Release the mouse button.

7. Confirm with **OK**.

5.7 Managing vehicle types, vehicle classes and vehicle categories

Using Vissim, you can group vehicles with similar technical driving properties into vehicle types and then classify vehicle types into vehicle classes (see "Using vehicle classes" on page 280). You must assign a vehicle type a vehicle category. The **Vehicle category** attribute specifies the basic behavior in traffic for a vehicle type (see "Using vehicle categories" on page 279).

5.7.1 Using vehicle types

A vehicle type allows you to form a group of vehicles with the same technical driving characteristics. The vehicle type data is included in the emission calculation. Vissim provides the following default vehicle types:

- Car
- HGV
- Bus
- Tram
- Man
- Woman
- Bike

Based on these vehicle types, you can define your own vehicle types, for example, trailer truck, articulated truck, standard bus, articulated bus.

If vehicles in a vehicle category have different speed or acceleration behavior, you define each vehicle type separately.

If vehicles of one type only differ in their shape, length or width, you may distinguish them by 2D/3D model distribution or color distribution and still manage them under the same vehicle type.

- Example 1: The models Car1 to Car6 represent vehicle models that differ in length, but have a similar driving behavior. This is why they can be defined under a single vehicle type, using 2D/3D model distribution for these 6 vehicles.
- Example 2: Standard and articulated buses only differ in length. This is why you can define them under a single vehicle type, using 2D/3D model distribution for the two vehicle models. To distinguish between standard and articulated buses for PT lines, you need to define standard buses and articulated buses as two separate vehicle types.

5.7.1.1 Defining vehicle types

1. From the **Base Data** menu, choose **Vehicle Types**.

   The list of defined network objects for the network object type opens.

   By default, you can edit the list (see "Using lists" on page 93).

   You can define a new vehicle type in the list.

2. Right-click in the list.
3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

   *The **Vehicle type** window opens.*

4. Enter the desired data.

Element	Description
No.	Unique identification number of the vehicle type
Name	Vehicle type label
Tab	- **Static**: (see "Editing static data of a vehicle type" on page 270) - **Functions and distributions**: (see "Editing functions and distributions of a vehicle type" on page 271) - **Special** > Section **Dynamic Assignment**: (see "Editing vehicle type data for the dynamic assignment" on page 272) - **Special** > **Others** > **External emission model**: (see "Activating emission calculation and emission model for a vehicle type" on page 274) - **Special** > Section **Other** > **Vehicle Type: PT Parameters**: (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275) - **External Driver Model**: (see "Activating the external driver model for a vehicle type" on page 277)

The attributes are saved in the **Vehicle Types** list (see "Attributes of vehicle types" on page 269).

5.7.1.2 Attributes of vehicle types

- From the **Base Data** menu, choose **Vehicle Types**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Unique identification number of the vehicle type
Name	Vehicle type label
Category	**Vehicle category** (see "Editing static data of a vehicle type" on page 270)
Model2D3DDistr	**2D/3D model distribution** (see "Using 2D/3D model distributions" on page 260), (see "Editing static data of a vehicle type" on page 270)
ColorDistr1	**Color distribution 1** of **Color 1** (see "Editing static data of a vehicle type" on page 270)
OccupDistr	**Occupancy distribution**: (see "Defining occupancy distributions" on page 255) Default value 1.
Capacity	**Capacity**: PT Parameters: Maximum number of passengers permitted per vehicle (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275)

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Parking lot selection parameters (see "Defining the destination parking lot selection" on page 756)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

5.7.1.3 Editing static data of a vehicle type

1. From the **Base Data** menu, choose **Vehicle Types**.

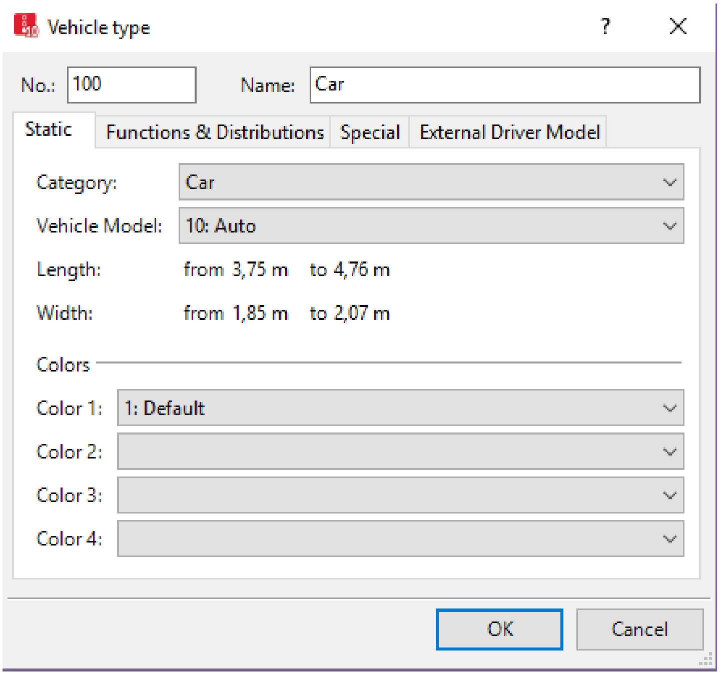
   The list of defined objects for the base data type opens.

2. Click on the desired vehicle type.
3. Right-click in the list.

4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*

5. Select the **Static** tab.



6. Make the desired changes:

Element	Description
No.	Unique number
Name	Vehicle type label
Category	One of the standard vehicle categories (see "Using vehicle categories" on page 279)
Vehicle Model	Vehicle shape and length for the vehicle type of the selected model distribution. You need to define new vehicle models in the 2D/3D model distribution.

Element	Description
Length	Minimum and maximum vehicle length, depending on 2D/3D model distribution (see "Using 2D/3D model distributions" on page 260)
Width	Minimum and maximum vehicle width, depending on 2D/3D model distribution (see "Using 2D/3D model distributions" on page 260) The width is relevant for overtaking within the lane (see "Applications and driving behavior parameters of lane changing" on page 300).
Colors	Color distributions define the colors for the 3D representation of four vehicle model parts of the vehicle type selected (see "Using color distributions" on page 262). This applies for all objects of a vehicle type. When you select a vehicle type, whose 2D/3D model color distributions have been assigned, these are displayed in the list boxes. You can already assign your 2D/3D models color distributions in V3DM. You may also define additional color distributions (see "Defining color distributions" on page 263). You can choose color distributions for each of the four colors. - Color 1: Attribute **ColorDistr1** - Color 2: Attribute **ColorDistr2** - Color 3: Attribute **ColorDistr3** - Color 4: Attribute **ColorDistr4** The setting is ignored for PT lines in the following cases: - when a different color is selected for the vehicle class to which the PT vehicle belongs - if a color is selected for the PT line itself

7. Confirm with **OK**.

5.7.1.4 Editing functions and distributions of a vehicle type

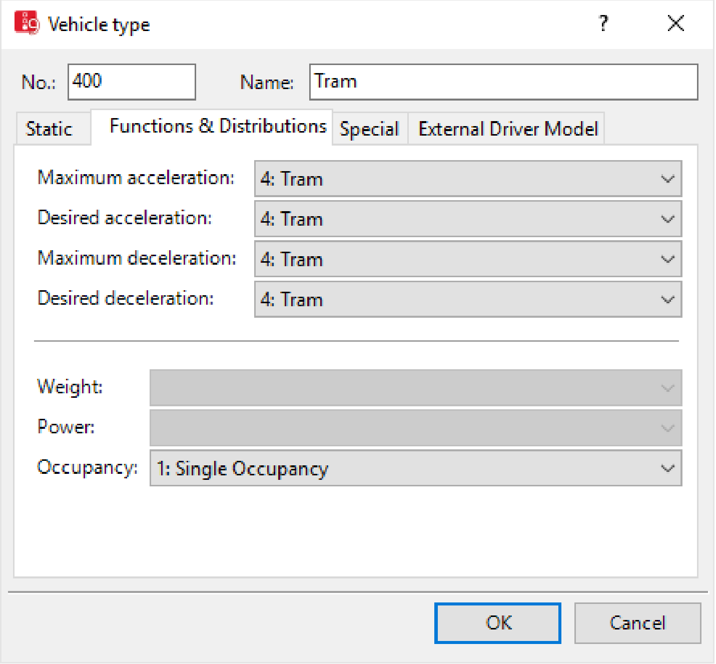
1. From the **Base Data** menu, choose **Vehicle Types**.

   The list of defined network objects for the network object type opens.

2. Click on the desired vehicle type.
3. Right-click in the list.
4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*

5. Select the **Functions & Distributions** tab.



6. Make the desired changes:

Element	Description
Maximum Acceleration **Desired Acceleration** **Maximum Deceleration** **Desired Deceleration**	Define the accelerations and decelerations of that vehicle type (see "Defining acceleration and deceleration functions" on page 233).
Weight	The weight distributions are active only for vehicle types of Category **HGV** and also, if an external model is selected (see "Using weight distributions" on page 243)
Power	The power distributions are active only for vehicle types of category **HGV** and also, if an external model is selected (see "Using power distributions" on page 240).
Occupancy	Defines the number of persons (including the driver) in a vehicle

7. Confirm with **OK**.

5.7.1.5 Editing vehicle type data for the dynamic assignment

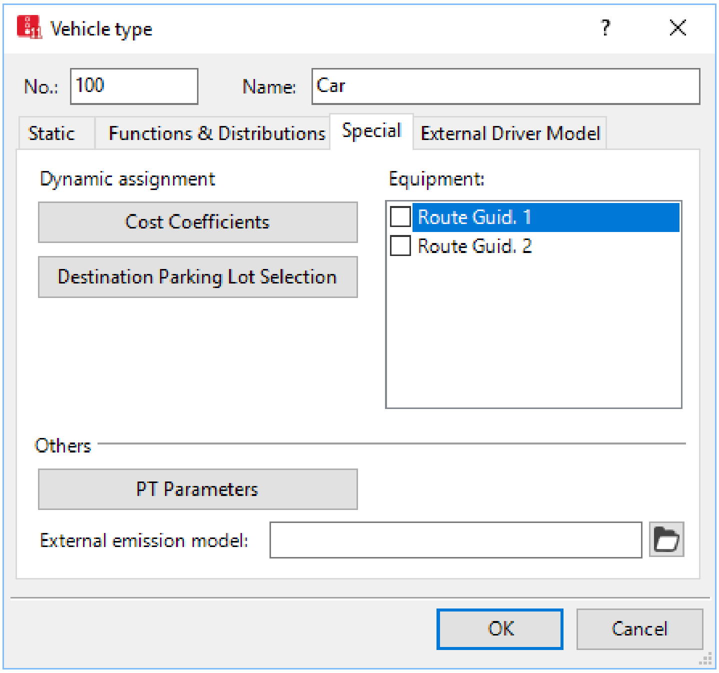
1. From the **Base Data** menu, choose **Vehicle Types**.

The list of defined network objects for the network object type opens.

2. Click on the desired vehicle type.
3. Right-click in the list.

4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*
5. Select the **Special** tab.



6. Make the desired changes:

Section	Element	Description
Dynamic assignment	**Cost Coef-ficients**	Defines the proportion of various factors for the path evaluation for that type (see "Defining simulated travel times" on page 735)
	Destination Parking Lot Selection	The parameters are used as a decision criterion for parking choice in a corresponding **decision situation** (see "Defining the destination parking lot selection" on page 756).
Equipment		**RouteGuidance1**, **RouteGuidance 2**: If this option is selected, use a route guidance system, for example a navigation system. It may interfere with your route selection.

7. Confirm with **OK**.

5.7.1.6 Activating emission calculation and emission model for a vehicle type

Note: You must have a license for the add-on module.

If you have purchased the add-on module **API**, you can access the *Emission Model.dll* file for emission calculation via an interface. You must provide the *EmissionModel.dll*. It is not part of Vissim.

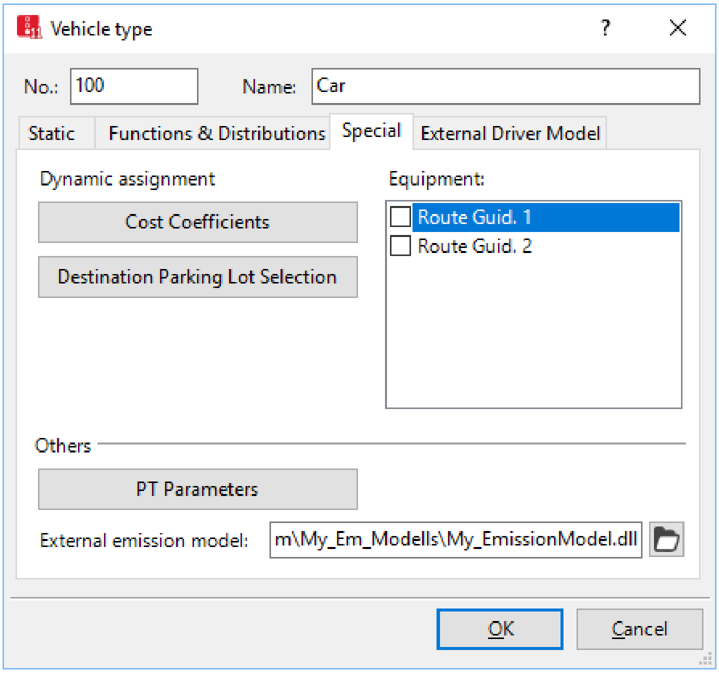
1. From the **Base Data** menu, choose **Vehicle Types**.

   The list of defined network objects for the network object type opens.

2. Click on the desired vehicle type.
3. Right-click in the list.
4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*

5. Select the **Special** tab.



6. Make the desired changes:

Section	Element	Description
Others	**External emission model**	Selection of an external emission model in the *Emission Model.dll* file. You must provide the *EmissionModel.dll*. It is not part of Vissim.

Emission values are displayed in:

- **Vehicle Network Performance Evaluation** (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085)
- **Vehicle record** (see "Saving vehicle record to a file or database" on page 1031)
- Evaluation **Links per segment** (see "Showing data from links in lists" on page 1103)
- **Vehicles in Network** list (see "Displaying vehicles in the network in a list" on page 847)

5.7.1.7 Changing attributes for a vehicle type for the duration of boarding and alighting

You can choose to calculate the duration of boarding and alighting based on the actual number of passengers The PT parameter definition is only applicable for PT vehicles of PT lines.

The duration of the passenger can also change based on a stochastic distribution of the stop time are calculated (see "Defining dwell time according to dwell time distribution" on page 532).

1. From the **Base Data** menu, choose **Vehicle Types**.

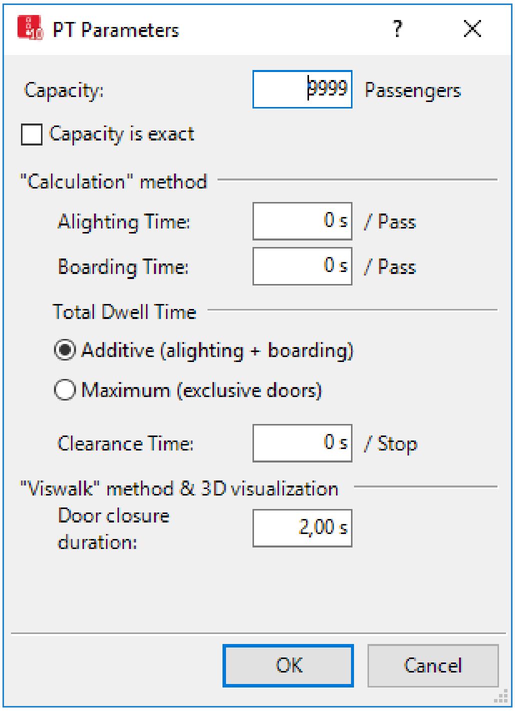
   The list of defined network objects for the network object type opens.

2. Click on the desired vehicle type.
3. Right-click in the list.
4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*

5. Select the **Special** tab.
6. Click the **PT Parameters** button.

   *The **PT Parameters** window opens.*



7. Make the desired changes:

Element	Description
Capacity	Maximum permitted number of passengers per vehicle. If the maximum capacity is reached and in the attribute **Enforce capacity limit exactly** you have not defined a larger number of passengers, no further passengers will be able to board the vehicle.
Enforce capacity limit exactly	If this option is selected, no more passengers than defined in the **Capacity** box will board the PT vehicle. If this option is not selected, within one time step more passengers could board the PT vehicle than specified in the **Capacity** box, in particular if the PT-vehicle has several doors.
"Calculation" method Alighting Time Boarding Time	Time required for a passenger to alight and board in seconds. Consider the number of doors: For example, if the time required is **6 s / Pass** and the vehicle has 3 doors, enter the value 2 seconds.
Total Dwell Time	**Sum**: Sum of the boarding and alighting times **Maximum**: Special doors will only be considered in the calculation for boarding or alighting by way of using the specified maximum time for

Element	Description
	each door.
Clearance Time	The time needed for a vehicle to stop, open/close doors, and other possible delays. Do not consider boarding and alighting times.
"Viswalk" method & 3D visualization	**Door closure duration** (**DoorClosDur**): Time required for the doors to close. Default value 2 s. You cannot edit the time required for the doors to open. It is 1.5 s. During this time the doors move parallel to the vehicle for 1.2 s and inwards for 0.3.s.

8. Make sure that the following data are defined:
   - Occupancy rate of the PT vehicles via **Departure times** in the **PT Line** window (see "Attributes of PT lines" on page 520)
   - Volume by PT stop (see "Attributes of PT stops" on page 513)
   - Alighting percentage and **Skipping possible** option in **PT Line Stop** window (see "Defining dwell time according to dwell time distribution" on page 532).
9. Confirm with **OK**.

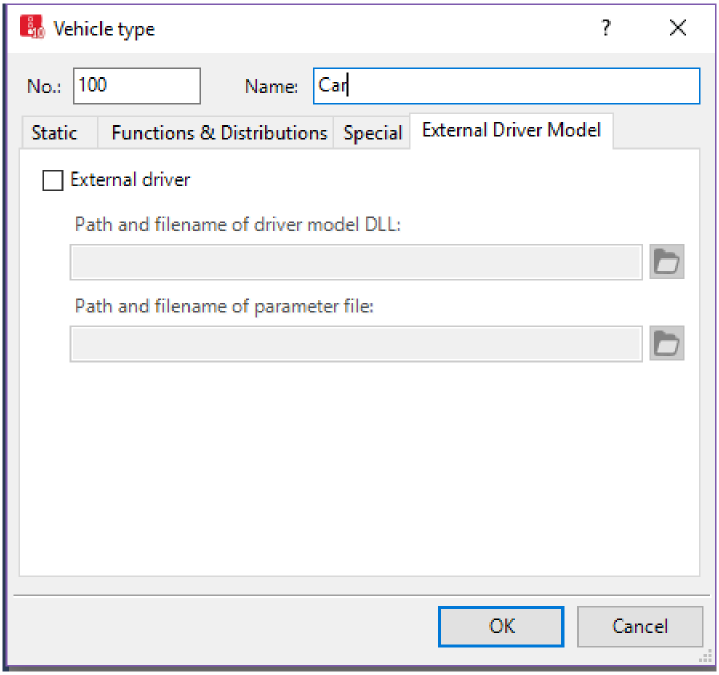
5.7.1.8 Activating the external driver model for a vehicle type

Note: You will need the **External Driver Model** add-on module.

1. From the **Base Data** menu, choose **Vehicle Types**.

   The list of defined network objects for the network object type opens.
2. Click on the desired vehicle type.
3. Right-click in the list.
4. From the shortcut menu, choose **Edit**.

   *The **Vehicle type** window opens.*
5. Select the **External Driver Model** tab.



6. Make the desired changes:

Element	Description
External driver	Only for the **External Driver Model** add-on module: If this option is selected, a vehicle type is not subject to the driving behavior models of Vissim, but is ruled by an external set of driving behavior parameters.
Path and file-name of driver model DLL	Enter the path and filename of the DriverModel.DLL containing the external driving behavior parameter set.
Path and file-name of para-meter file	Enter the path and filename of the parameter file.

7. Confirm with **OK**.

Vissim sends e.g. the following data to the DLL file even if 0 is returned by **DriverModelGetValue** (DRIVER_DATA_SETS_XY_COORDINATES, ...):

- DRIVER_DATA_VEH_REAR_X_COORDINATE
- DRIVER_DATA_VEH_REAR_Y_COORDINATE

This means that global coordinates for vehicle rear ends are available in usual driver model DLLs for vehicles on Vissim links.

For detailed information on all types that are managed in the files *DriverModel.cpp* and *DriverModel.h* , please refer to the file *Interface_ Description.pdf* in the folder *..\API\DriverModel_DLL* of your Vissim installation.

5.7.2 Using vehicle categories

The **Vehicle category** attribute of a vehicle type specifies its basic behavior in traffic (see "Operating principles of the car following model" on page 32). The latter varies between individual vehicle categories. For example, the vehicle category **Tram** does not allow for lane changes and the speed of vehicles of this category is not based on a desired speed. You need to assign each vehicle type a vehicle category (see "Editing static data of a vehicle type" on page 270). Vissim provides the following default vehicle categories:

- Car
- HGV
- Bus
- Tram
- Pedestrian
- Bike

The table shows the properties of vehicle categories that differ from the properties of the vehicle category **Car**:

Vehicle category	Differing property
HGV	• The weight distribution and the power distribution are only relevant for this vehicle category and only for the spread in acceleration curves (see "Using weight distributions" on page 243), (see "Using power distributions" on page 240). • In Wiedemann 99, in the **Free** and **Follow** interaction states, accelerates with only half the acceleration calculated (see "Driving states in the traffic flow model according to Wiedemann" on page 285), (see "Value of the Interaction state attribute" on page 852) • For the **right-side rule** general behavior when changing lanes, has different values for some non-user defined parameters of free lane changing. • Doesn't have an occupancy distribution, just one driver (see "Defining occupancy distributions" on page 255)
Bus	The properties correspond to the properties of the car vehicle category.
Tram	• Lange changes not allowed • In Wiedemann 74, in the **Free** interaction state and in the **Free driving** state, does not oscillate around the desired speed, but rather drives exactly at the desired speed (see "Driving states in the traffic flow model according to Wiedemann" on page 285), (see "Value of the Interaction

Vehicle category	Differing property
	state attribute" on page 852)

Pedestrian	- Always brakes at amber and red
	- Calculates a safety distance of 0.1 m when changing lanes (see "Editing the driving behavior parameter Following behavior" on page 286), (see "Defining the Wiedemann 74 model parameters" on page 294)
	- Doesn't have a stochastic threshold that defines the speed below the desired velocity at which a vehicle would be overtaken. Therefore a pedestrian overtakes immediately when he cannot continue at the desired velocity.
	- Doesn't have an occupancy distribution, just one person (see "Defining occupancy distributions" on page 255)
	- Has 21 motion states:
	  - Has a state for standing pedestrians
	  - 20 states for motion sequence of two steps

Bike	- Doesn't have an occupancy distribution, just one person (see "Defining occupancy distributions" on page 255)
	- Has 21 motion states:
	  - Has a state for cyclists
	  - 20 states for motion sequence of one crank revolution

5.7.3 Using vehicle classes

You can group vehicle types into vehicle classes. A vehicle class may contain any number of vehicle types. Vehicle classes provide the basis for speed data, evaluations, path selection behavior and other network objects. Per default, a vehicle class contains a vehicle type of the same name. You may assign a vehicle type to several vehicle classes. A vehicle class is, for example, used to obtain data for specific vehicle types or to recognize and distinguish them based on their color during simulation.

Vehicles with different technical driving properties must belong to different vehicle types. Group vehicle types to a vehicle class in the following cases:

- If for these vehicles you still want to define the same properties, for example route choice behavior.
- If you wish to collect aggregated data.

If vehicles with the same technical driving properties only differ in shape or color, they vehicles can still be assigned to the same vehicle type. To be able to distinguish between individual vehicles, for this vehicle type, select a suitable 2D/3D model distribution and color distribution.

5.7.3.1 Defining the vehicle class

You can define vehicle classes and assign vehicle types.

1. Select from the menu **Base Data** > **Vehicle Classes**.

The list of defined objects for the base data type opens.

By default, you can edit the list (see "Using lists" on page 93).

You can define a new vehicle class in the list.

2. In the list, on the toolbar, click the **Add** button ✚ .

A new row with default data is inserted.

Enter the desired data.

Element	Description
No	Unique identification number of the vehicle class
Name	Label of the vehicle class
Color	Default color of vehicle class during simulation (see "Static colors of vehicles and pedestrians" on page 175). Is not used in the following cases: ➤ When for a vehicle class the attribute **Use vehicle type color** (**UseVe-hTypeColor**) is enabled. ➤ When for the display of vehicles in the network, from the Graphic Para-meters menu, **DrawingMode** > **Use color scheme** is chosen, and for the **Color scheme configuration** attribute, a color scheme is specified that is to be used for classification.
Vehicle types	**VehTypes**: List box with options for selecting the vehicle types you wish to assign. Numbers and names of vehicle types.
Use vehicle type color	**UseVehTypeColor**: ☑ If this option is selected, the vehicle color is determined by simulation of each vehicle type (or public transport line respectively). The **Use vehicle type color** attribute is not used, when for the display of pedestrians in the network, from the Graphic Parameters menu, **DrawingMode** > **Use color scheme** is chosen, and for the **Color scheme configuration** attribute, a color scheme is specified that is to be used for classification.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of base data objects allocated to the base data object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click **Vehicle types**.
3. Enter the desired data.

The data is allocated.

5.8 Defining driving behavior parameter sets

Warning: Driving behavior parameters control the driving behavior and can therefore lead to a considerable change in the simulation results! Change the driving behavior parameters only if you are a very experienced user!

In a driving behavior parameter set, you can define the driving behavior properties of a link behavior type. In the driving behavior parameter set, select the desired values and options for the following parameters:

- Following
- Car following model without interaction or according to Wiedemann 74 or Wiedemann 99
- Lateral behavior
- Lane change behavior
- Behavior at signal controls
- Parameters for mesoscopic simulation

You may define several driving behaviors. The driving behavior parameter sets are predefined by default. You assign a link the desired driving behavior via the **Behavior type** attribute (see "Attributes of links" on page 409).

You can change the driving behavior parameters during the simulation. However, you cannot change **Safety distance reduction factor start (signals)** or **Safety distance reduction factor end (signals)**.

For each vehicle class, you can assign a driving behavior parameter set to a link behavior type. You assign links the desired link behavior type via the **Link behavior type** attribute (see "Defining link behavior types for links and connectors" on page 318).

1. From the **Base Data** menu, choose > **Driving Behaviors**.

   *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined. The Attributes list is displayed as the left of two coupled lists.*

2. Right-click in the list.

3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

   *The **Driving Behavior** window opens.*

4. Enter the desired data.

Element	Description
No.	Unique number of the driving behavior parameter set
Name	Name of the link behavior type assigned

Element	Description
Tab	- Following (see "Editing the driving behavior parameter Following behavior" on page 286) - Car following model (see "Editing the driving behavior parameter car following model" on page 293) - Lane change (see "Applications and driving behavior parameters of lane changing" on page 300) - Lateral behavior (see "Editing the driving behavior parameter Lateral behavior" on page 308) - Signal controllers (see "Editing the driving behavior parameter Signal Control" on page 315) - Meso (see "Editing the driving behavior parameter Meso" on page 317)

5. Confirm with **OK**.

5.8.1 Editing driving behavior parameters

Warning: Driving behavior parameters control the driving behavior and can therefore lead to a considerable change in the simulation results! Change the driving behavior parameters only if you are a very experienced user!

- From the **Base Data** menu, choose > **Driving Behavior**.

  *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined. The **Driving behavior** list is displayed as the left of two coupled lists.*

  *You can edit the driving behavior parameters for each driving behavior in the list or in the **Driving Behavior** window, in tabs.*

5.8.1.1 Editing driving behavior parameters in the list

1. Ensure that the columns in which you want to edit attribute values are shown.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. In the line of the relevant driving behavior, edit the attribute value of your choice.

The driving behavior has additional attributes that you can show in the Attributes list. Among them are the following attributes for example:

Attributes influencing the driving behavior during recovery from a traffic collapse

In real networks, after a traffic collapse, vehicles can recover more slowly and further downstream than is modeled with the default driving behaviors defined. The following attributes allow you to calibrate driving behavior, so that it matches real data that shows a slow recovery. To do so, select the **Recovery slow** attribute. The other attributes listed in the table below then have an impact within the distance defined in the **Recovery distance** attribute, even if **Recovery slow** has been selected for the driving behavior within this distance.

- The **Recovery slow** attribute has been selected for the vehicle for a time step of the simulation.
- During this time step, the vehicle stays below the speed defined in the attribute **Recovery threshold speed**.

Long name	Short name	Description
Recovery acceleration	**RecovAcc**	Percentage of normal acceleration used during slow recovery. Default value of 40 %.
Recovery distance	**RecovDist**	Maximum distance of impact of slow recovery from last location of a traffic collapse. Default 2000 m
Recovery threshold speed	**RecovSpeed**	Speed limit below which Vissim detects a traffic collapse. The default value is 60% of the desired speed. As long as the vehicle follows a driving behavior for which **Recovery slow** has been selected or the it reaches the distance defined via the **Recovery distance** attribute, the following applies: - In the interaction state **Free**, the vehicle accelerates to **Recovery threshold speed**. - The speed-dependent part of the desired safety distance is increased to **Recovery safety distance**.
Recovery slow	**RecovSlow**	☑ If this option is selected, how quickly vehicles recover after a traffic collapse depends on the attributes listed in this table.
Recovery safety distance	**RecovSafDist**	Percentage of normal safety distance used during slow recovery. Default value of 110 %.

5.8.1.2 Editing driving behavior parameters in the Driving Behavior window

1. Double-click the driving behavior of your choice.

   *The **Driving Behavior** window opens.*

2. Enter the desired data.

Element	Description
No.	Unique number of the driving behavior parameter set
Name	Name of the link behavior type assigned

Element	Description
Tab	- Following (see "Editing the driving behavior parameter Following behavior" on page 286) - Car following model (see "Editing the driving behavior parameter car following model" on page 293) - Lane change (see "Applications and driving behavior parameters of lane changing" on page 300) - Lateral behavior (see "Editing the driving behavior parameter Lateral behavior" on page 308) - Signal controllers (see "Editing the driving behavior parameter Signal Control" on page 315) - Meso (see "Editing the driving behavior parameter Meso" on page 317)

3. Confirm with **OK**.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Lateral behavior (see "Editing the driving behavior parameter Lateral behavior" on page 308)
- Vehicle class following (see "Editing the driving behavior parameter car following model" on page 293)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

5.8.2 Driving states in the traffic flow model according to Wiedemann

Vissim's traffic flow model is a stochastic, time step based, microscopic model that treats driver-vehicle units as basic entities.

The traffic flow model contains a psycho-physical car following model for longitudinal vehicle movement and a rule-based algorithm for lateral vehicle movement. The models deployed are based on Wiedemann's extensive research work.

- Wiedemann, R. (1974). Simulation des Straßenverkehrsflusses. Schriftenreihe des Instituts für Verkehrswesen der Universität Karlsruhe (seit 2009 KIT – Karlsruher Institut für Technologie), Heft 8
- Wiedemann, R. (1991). Modeling of RTI-Elements on multi-lane roads. In: Advanced Telematics in Road Transport edited by the Commission of the European Community, DG XIII, Brussels

Wiedemann's traffic flow model is based on the assumption that there are basically four different driving states for a driver (see " Traffic flow model and light signal control" on page 31):

- **Free driving**: No influence of preceding vehicles can be observed. In this state, the driver seeks to reach and maintain his desired speed. In reality, the speed in free driving will vary due to imperfect throttle control. It will always oscillate around the desired speed.
- **Approaching**: Process of the driver adapting his speed to the lower speed of a preceding vehicle. While approaching, the driver decelerates, so that there is no difference in speed once he reaches the desired safety distance.
- **Following**: The driver follows the preceding car without consciously decelerating or accelerating. He keeps the safety distance more or less constant. However, again due to imperfect throttle control, the difference in speed oscillates around zero.
- **Braking**: Driver applies medium to high deceleration rates if distance to the preceding vehicle falls below the desired safety distance. This can happen if the driver of the preceding vehicle abruptly changes his speed or the driver of a third vehicle changes lanes to squeeze in between two vehicles.

For each of the four driving states, acceleration is described as a result of current speed, speed difference, distance to the preceding vehicle as well as of individual driver and vehicle characteristics.

Drivers switch from one state to another as soon as they reach a certain threshold that can be described as a function of speed difference and distance. For instance, small differences in speed can only be perceived at short distances. Whereas large differences in speed already force drivers to react at large distances.

The perception of speed differences as well as the desired speed and safety distance kept vary across the driver population.

As the model accounts for psychological aspects as well as for physiological restrictions of drivers' perception , it is called psycho-physical car-following model.

5.8.3 Editing the driving behavior parameter Following behavior

1. From the **Base Data** menu, choose **Driving Behaviors**.

*The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined.*

By default, you can edit the list (see "Using lists" on page 93).

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

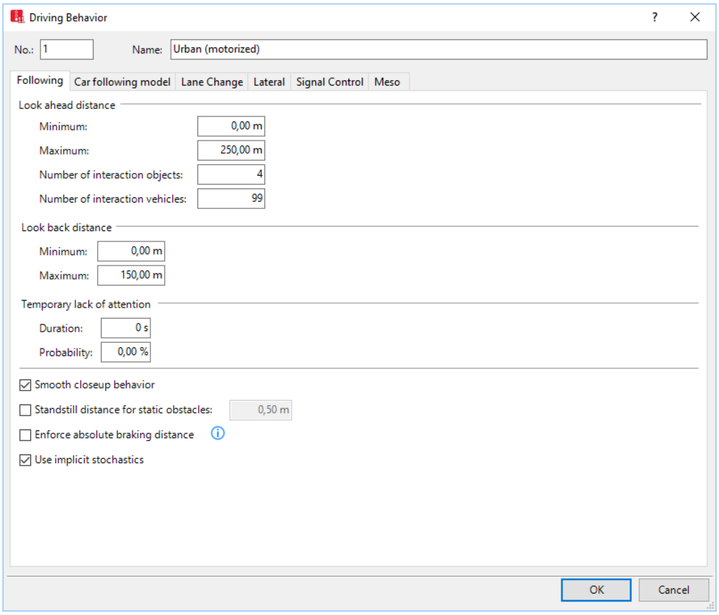
---

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Following** tab.



5. Enter the desired data.

Element	Description
Look ahead distance	**Mininum**, **Maximum**: Minimum and maximum distance that a vehicle can see ahead in order to react to other vehicles in front of or next to it on the same link. Vehicles take into account the minimum and maximum look-ahead distance in addition to the entered number of preceding vehicles. The **Minimum look-ahead distance** is important when modeling the lateral behavior of vehicles. • If the minimum look ahead distance is 0.00, only the number of **Observed vehicles** applies, which is specified in the attribute of **Observed vehicles**. • If several vehicles can overtake within a lane, this value needs to be greater than 0.00, e.g. in urban areas, depending on the speed, the look ahead distance might be approx. 20-30m, with correspondingly larger values for outside of the city. This way you avoid that during the simulation the impression is created that one vehicle is passing through another vehicle. This may happen when there are more vehicles than specified in the **Observed vehicles** attribute that want to position themselves in front of a stop line on the same link. This applies in particular to bicycles. • If several vehicles can overtake within a lane, you can enter a greater look ahead distance to prevent any vehicles from running a red light. When doing so, do not change the number of **Observed vehicles**. This can lead to an unrealistic simulation. Only the **Maximum look-ahead distance** needs to be extended, e.g. to model rail traffic with block signals (see "Modeling railroad block signals" on page 673).
	Number of interaction objects : The number of observed vehicles and/or the number of certain network objects affect/s how well vehicles in the link can predict other vehicles' movements and react accordingly. In addition to the number of observed vehicles entered, vehicles take the minimum and maximum **Look ahead distance** into account. In Vissim, the following network objects are modeled as vehicles. Vehicles treat these network objects as a preceding vehicle. These network objects act like a red signal head. If there are several of these network objects within a very short distance, enter a larger number for the **number of interaction objects**. However, this can lead to slightly longer simulation computation times. • Red signal heads • Reduced Speed Areas • Priority rules for cases in which the minimum time gap or minimum headway is not kept. Vehicles also treat the following network objects as a preceding vehicle, when they have to stop there:

Element	Description
	- Stop Signs - Public transport stops - Parking Lots Conflict areas behavior: A vehicle takes into consideration all conflict areas up to the preceding vehicle, indicative of the number of **interaction objects**. The default value for predefined driving behavior in **urban areas (motorized)**: 4 vehicles The default value for all other predefined driving behaviors: 2 vehicles In the **Number of interaction vehicles** attribute, under **Number of interaction objects**, you can specify the number of vehicles that Vissim takes into account in addition to the above-mentioned network objects. The vehicle is maximally aware of this number of interaction objects.
Number of interaction vehicles	Number of vehicles ahead that the vehicle further downstream perceives. The vehicle can thus react to vehicles in front of it or on the same link next to it. - The **Number of Interaction vehicles** is included in the **Number of interaction objects**. - The **Number of interaction vehicles** does not include the network objects listed in the description of the attribute **Number of interaction objects** and those that the vehicle perceives as a vehicle. - The vehicle is maximally aware of the **Number of interaction objects**. Default value for all predefined driving behaviors: 99 vehicles

Element	Description
Look back distance	**Minimum**, **Maximum**: Minimum and maximum distance a vehicle can see behind it in order to react to other vehicles behind it on the same link. The minimum look-back distance is important when modeling lateral vehicle behavior. If several vehicles can overtake within a lane, this value needs to be greater than 0.00, e.g. in urban areas it could be 20-30m, with correspondingly larger values in other places. This way you make sure the cars drive in an orderly fashion when two or more vehicles, than specified in the **Observed vehicles** attribute, on the same route want to position themselves at a stop line. This applies in particular to bicycles. You can reduce the maximum look-back distance in close-meshed networks, e.g. with many connectors over a short distance. This may positively affect the simulation speed.

Element	Description
Temporary lack of attention	**Duration**: The period of time when vehicles may not react to a preceding vehicle. They do react however to emergency braking. **Probability**: Frequency of the lack of attention With increasing values, the capacity of the affected links decreases.

Element	Description
Smooth closeup behavior	☑ If this option is checked, vehicles slow down more evenly when approaching a stationary obstacle. At the **maximum look-ahead** distance from the stationary obstacle, a following vehicle can plan to stop there as well, because the preceding vehicle will stop there too. ☐ If this option is not selected, the following vehicle uses the normal following behavior until the speed of the preceding vehicle drops to < 1 m/s and it comes almost to a halt. Only then, the following vehicle determines the final approach behavior. This approach behavior can include a temporary acceleration.

Element	Description
Standstill distance for static obstacles	Standstill distance (ax) upstream of static obstacles such as signal heads, stop signs, PT stops, priority rules, conflict areas. Not valid for stop signs in parking lots. The attribute **Smooth closeup behavior** must be selected. ☐ If this option is not selected, the vehicles use a normally distributed random value [0.5; 0.15]. ☑ If this option is selected, the vehicles will use the given value. The default value is 0.5 m. Activate this option for PT vehicles at PT stops with platform screen doors and queues at fixed positions on the platform. Enter the desired distance. Note: From Vissim 6, the optimized modeling of driving behavior can lead to different results than in the previous versions.

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Maintain absolute braking distance</td><td>EnforcAbsBrakDist: For braking, the vehicle accounts for the distance necessary to stop without causing a collision should the vehicle in front it stop immediately without a braking distance. The absolute braking distance applies to vehicles in the following cases:

Following behavior: The desired safety distance corresponds to at least the absolute braking distance.
Lane change: When deciding to change the lane, the vehicle takes into account the absolute braking distance to both the new vehicle preceding it and to the new vehicle following it. If the absolute braking distance is not sufficient, the vehicle does not change the lane.
For following behavior and lane changes the following applies: If for the vehicle the car following model Wiedemann 74 is selected, the standstill distance ax is added to calculate the absolute braking distance for other vehicles. If the car following model Wiedemann 99 is selected, instead the standstill distance CC0 is added.
In a crossing conflict, the vehicle travels the conflict area in the subordinate flow, if the vehicle in the main flow can maintain the absolute braking distance.
Vehicle inputs: The vehicle is inserted into the Vissim network with at least the absolute braking distance.
Conflict areas:The absolute braking distance applies to vehicles that are involved in the conflict and have priority: The vehicle is inserted into the Vissim network with at least the absolute braking distance. The gap time the vehicle with the right of way needs to come to a stop before the conflict, is calculated by Vissim from the current speed of the vehicle and the maximum deceleration possible at this speed. The time it takes the other vehicle to cover a distance of 1 m, driving at its current speed, is added to the gap time.

The absolute braking distance is not effective where priority rules are used.</td></tr>
</table>

Element	Description
Use implicit stochastics	**UseImplicStoch**: ☑ If this option is selected, for the following attributes and estimation uncertainty that takes human perception fluctuations into account, Vissim uses a variable stochastic value based on a) the time distribution of the speed-dependent part of the desired safety distance **CC1** and b) on distributions that apply to the desired acceleration and deceleration: - Safety distance - Desired Acceleration - Desired Deceleration ☐ If the option is deselected, Vissim uses non-variable deterministic values that are based on the time distribution meridian of the speed-dependent part of the desired safety distance **CC1** and of distributions that apply to the desired acceleration and deceleration.

In the **Driving behavior** list, you can select additional attributes. Among them are the following for example:

Element	Description
Increased Accel-eration	**IncrsAccel**: Allows you to increase the acceleration with which the vehicle follows a preceding vehicle that accelerates. Default value 100 %, value range 100 % to 999 %. With the default value 100%, the vehicle falls back in distance from the preceding vehicle when the latter accelerates. Only when the preceding vehicle stops accelerating does the following vehicle approach it. If the value > 100 %, the vehicle accelerates when the preceding vehicle accelerates and unimpeded acceleration is possible. **Increased Acceleration** affects the car following models **Wiedemann 74** and **Wiedemann 99**. **Increased Acceleration** has an impact on the following types of acceleration: - Desired Acceleration - **CC8**: **Desired acceleration from a standstill** - **CC9**: **Desired acceleration at 80 km/h** Jerk limitation in the **Wiedemann 99** car following model: If the vehicle is in the interaction state **Free**, acceleration is limited in the first time step of the vehicle via the model parameter **CC7 Oscillation during acceleration**. Jerk limitation is not performed if a value > 100 % is selected for **Increased Acceleration**, the distance dx > safety distance dsx and the preceding vehicle is accelerating.

5.8.4 Editing the driving behavior parameter car following model

1. From the **Base Data** menu, choose > **Driving Behaviors**.

   *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined.*

   By default, you can edit the list (see "Using lists" on page 93).

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

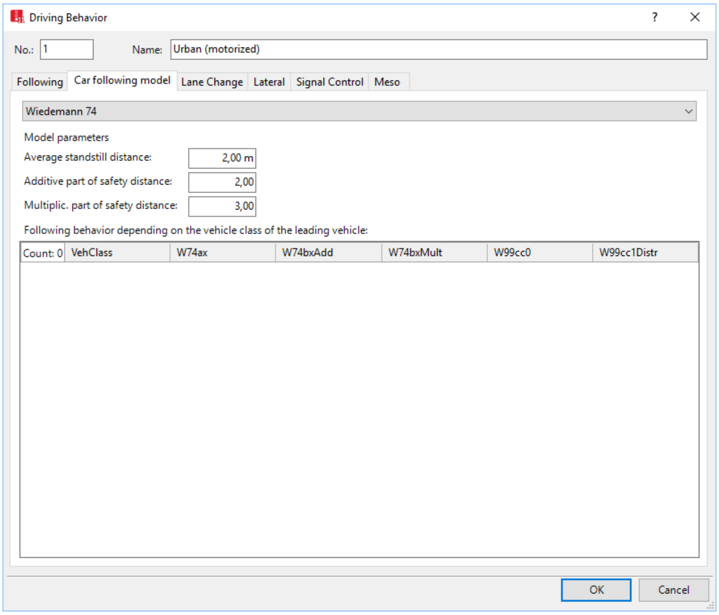
---

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Car following model** tab.



5. Enter the desired data.

Element	Description
List box	Car following model for the car-following behavior. Depending on the selected car following model the **Model parameters** change. - **No interaction**: Vehicles do not recognize any other vehicles. Use this entry to model pedestrian flows in an easy way. - **Wiedemann 74**: Model suitable for urban traffic and merging areas - **Wiedemann 99**: Model for freeway traffic with no merging areas
Model parameters	Displays different parameters depending on the car following model selected (see "Defining the Wiedemann 74 model parameters" on page 294), (see "Defining the Wiedemann 99 model parameters" on page 296). These model parameters affect the saturation flow.

Defining following behavior based on the vehicle class of the preceding vehicle

You can enter model parameters according to **Wiedemann 74** and **Wiedemann 99**. They define the following behavior of a vehicle based on the vehicle class of its preceding vehicle. This can be done for every vehicle class defined. The following behavior takes effect from the moment the vehicle enters the vehicle input.

1. To change the model parameters of a vehicle’s following behavior based on several vehicle classes, carry out the following steps in sequence for each desired vehicle class.
2. Right-click the section **Following based on vehicle class of preceding vehicle**.
3. From the shortcut menu, choose **Add**.

   *A new row is inserted. The row contains the model parameters for the vehicle class with the lowest number. The latter is not displayed yet. All **Wiedemann 74** and **Wiedemann 99** model parameters that are relevant for the following behavior are displayed (see "Defining the Wiedemann 74 model parameters" on page 294),(see "Defining the Wiedemann 99 model parameters" on page 296). The model parameters of the vehicle following model that is not selected are marked as hatched, i.e. as not relevant.*

4. Into the boxes that are not hatched, enter the desired attribute values for the following behavior of the vehicle, if they differ from the default values.

5.8.4.1 Defining the Wiedemann 74 model parameters

This model is an improved version of Wiedemann’s 1974 car following model.

The following parameters are available:

Parameters	Description
Average standstill distance (w74ax)	*(ax)*: Defines the average desired distance between two cars. The tolerance lies from –1.0 m to +1.0 m which is normally distributed at around 0.0 m, with a standard deviation of 0.3 m. Default value 2.0.
Additive part of safety distance (w74bxAdd)	(bx_{add}): Value used for the computation of the desired safety distance d. Allows to adjust the time requirement values. Default 2.0
Multiplicative part of safety distance (w74bxMult)	(bx_{mult}): Value used for the computation of the desired safety distance d. Allows to adjust the time requirement values. Greater value **=** greater distribution (standard deviation) of safety distance Default 3.0

The desired distance d is calculated from:

$$d = ax + bx$$

where:

ax: Standstill distance

$$bx = (\mathrm{bx}_{add} + bx_{mult} * z) * \sqrt{v}$$

v: vehicle speed [m/s]

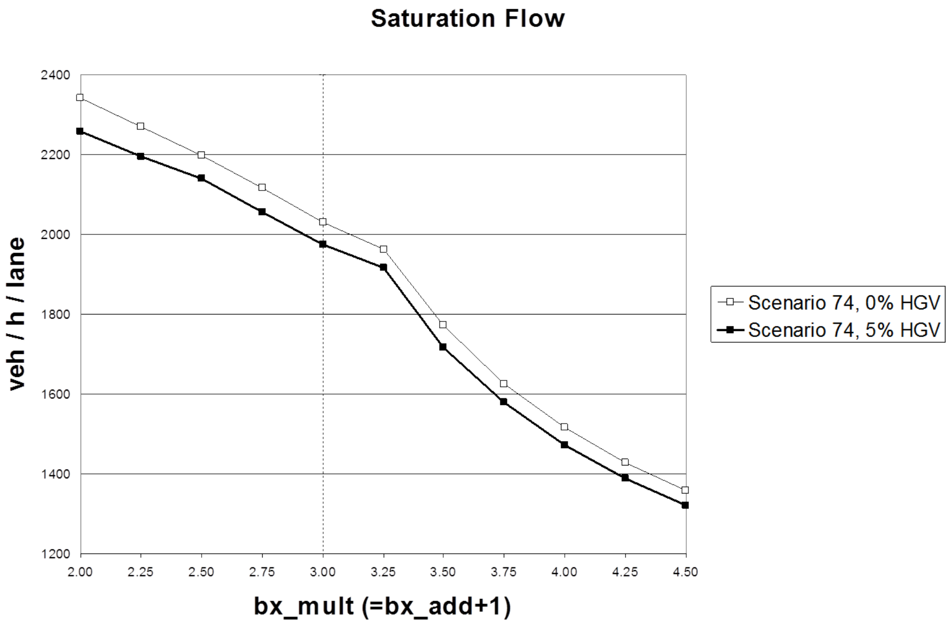
z: is a value of range [0.1], which is normally distributed around 0.5 with a standard deviation of 0.15

Defining the saturation flow rate with the Wiedemann 74 modeling parameters

The saturation flow rate defines the number of vehicles that can flow freely on a link for an hour. Impacts created through signal controls or queues are not accounted for. The saturation flow rate also depends on additional parameters, e.g. speed, share of HGV, or number of lanes.

You define the saturation flow by combining the parameters **Additive part of safety distance** and **Multiplicative part of safety distance**. Experienced users may want to use these parameters to adapt their model to observation data.

Note: The graphs show the saturation flow rates calculated for examples used in Vissim. When using a different network, you receive graphs depicting different values.



Scenario 74 was created with the following parameters:

- single lane link
- speed distribution between 48 and 58 km/h
- Default driving behavior, with the exception of parameters bx_{add} (**Additive part of safety distance**) and bx_{mult} (**Multiplicative part of safety distance**) that vary along the x-axis. In this example the following applies: $bx_{add} = bx_{mult}-1$
- one time step per simulation second

5.8.4.2 Defining the Wiedemann 99 model parameters

This model is based on Wiedemann's 1999 car following model.

The following parameters are available:

Parameters	Unit	Description
CC0	m	**Standstill distance**: The desired standstill distance between two vehicles. It has no variation. You can define the behavior upstream of static obstacles via the attribute **Standstill distance for static obstacles** (see "Editing the driving behavior parameter Following behavior" on page 286).
CC1	s	**Following distance**:Time distribution of speed-dependent part of desired safety distance. Shows number and name of time distribution Each time distribution may be empirical or normal. Each vehicle has an individual, random **safety** variable. Vissim uses this random variable as a fractile for the selected time distribution **CC1**. Based on the time distribution, the following distance for a vehicle is calculated. This is the distance in seconds which a driver wants to maintain at a certain speed. The higher the value, the more cautious the driver is. The safety distance is defined in the car following model as the minimum distance a driver will maintain while following another vehicle. In case of high volumes this distance becomes the value which has a determining influence on capacity.
CC2	m	**Longitudinal oscillation**. Restricts the distance difference a driver allows for before he intentionally moves closer to the car preceding him. If this value is set to e.g. 10 m, the following behavior results in distances between dx_{safe} and $dx_{safe} + 10m$. The default value is 4.0m which results in a quite stable following behavior.
CC3	s	**Perception threshold for following**: Defines the beginning of the deceleration process, i.e. the number of seconds before the safety distance is reached. At this stage the driver recognizes a preceding slower vehicle.
CC4	m/s	**Neg. speed difference**: Defines negative speed difference during the following process. Low values result in a more sensitive driver reaction to the acceleration or deceleration of the preceding vehicle.
CC5	m/s	**Pos. speed difference**: Defines positive speed difference during the following process. Enter a positive value for **CC5** which corresponds to the negative value of **CC4**. Low values result in a more sensitive driver reaction to the acceleration or deceleration of the preceding vehicle.
CC6	1/(m • s)	**Influence speed on oscillation**: Influence of distance on speed oscillation during the following process: - Value 0: The speed oscillation is independent of the distance - Larger values: Lead to a greater speed oscillation with increasing distance
CC7	m/s^2	**Oscillation during acceleration**: Oscillation during the acceleration phase

Parameters	Unit	Description
CC8	m/s^2	**Acceleration starting from standstill**: Desired acceleration when starting from standstill (limited by maximum acceleration defined within acceleration curves).
CC9	m/s^2	**Acceleration at 80 km/h**: Desired acceleration at 80 km/h (limited by maximum acceleration defined within acceleration curves).

Note: The units of Wiedemann 99 model parameters cannot be edited. These units are independent of the network settings for units in the base data.

Defining the saturation flow rate with the Wiedemann 99 modeling parameters

The saturation flow rate defines the number of vehicles that can flow freely on a link for an hour. Impacts created through signal controls or queues are not accounted for. The saturation flow rate also depends on additional parameters, e.g. speed, share of HGV, or number of lanes.

In the car-following model Wiedemann 99, parameter **CC1** has a major impact on the safety distance and saturation flow rate. The scenarios shown below are based on the following assumptions:

- car-following model Wiedemann 99, containing default parameters with the exception of **CC1** that varies across the x-axis
- one time step per simulation second

The main properties of the following graphs are:

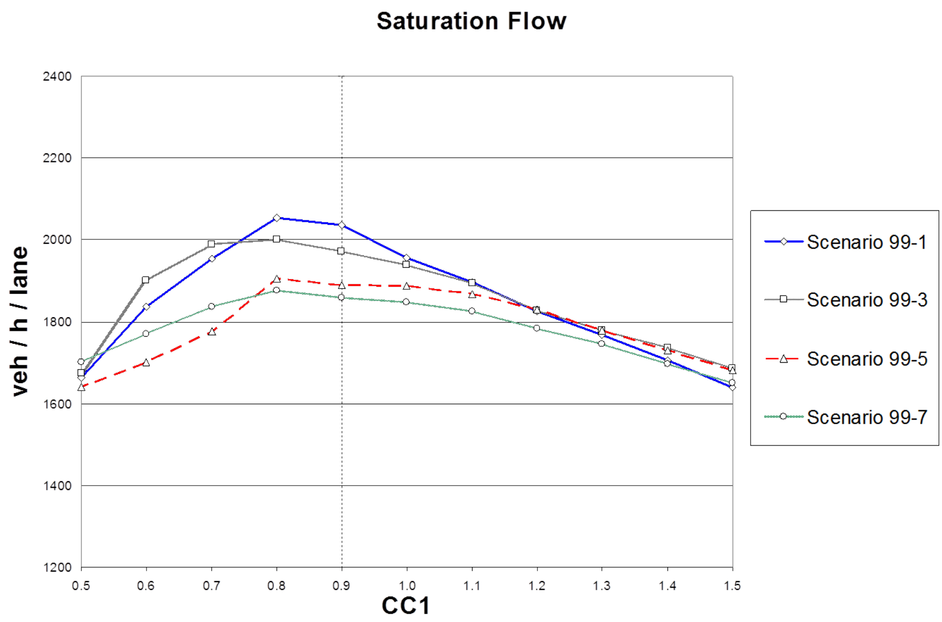
Scenario	Right-side rule	Lane	Speed cars*	Speed HGV*	% HGV
99-1	no	2	80	n/a	0%
99-2	no	2	80	85	15%
99-3	yes	2	80	n/a	0%
99-4	yes	2	80	85	15%
99-5	yes	2**	120	n/a	0%
99-6	yes	2	120	85	15%
99-7	yes	3***	120	n/a	0%
99-8	yes	3	120	85	15%

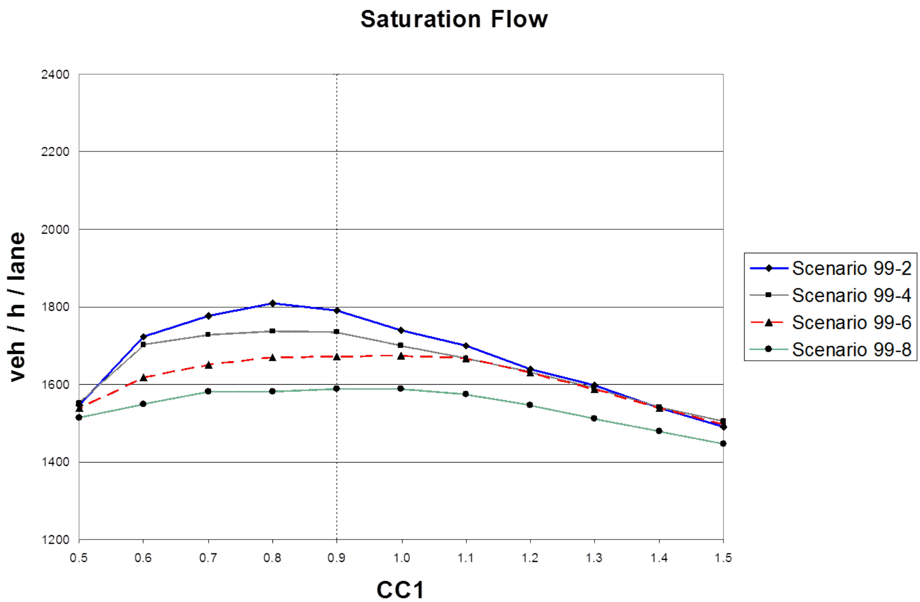
* Vissim default setting

** Lane 2 closed for HGV traffic

*** Lane 3 closed for HGV traffic

Note: The graphs show the saturation flow rates calculated for examples used in Vissim. When using a different network, you receive graphs depicting different values.


Saturation Flow
veh / h / lane
2400
2200
2000
1800
1600
1400
1200
0.5
0.6
0.7
0.8
0.9
1.0
1.1
1.2
1.3
1.4
1.5
CC1
Scenario 99-1
Scenario 99-3
Scenario 99-5
Scenario 99-7


Saturation Flow
veh / h / lane
2400
2200
2000
1800
1600
1400
1200
0.5
0.6
0.7
0.8
0.9
1.0
1.1
1.2
1.3
1.4
1.5
CC1
Scenario 99-2
Scenario 99-4
Scenario 99-6
Scenario 99-8

5.8.5 Applications and driving behavior parameters of lane changing

Vissim distinguishes between the following lane changes:

- Necessary lane change in order to reach the next connector of a route

  For a necessary lane change, the driving behavior parameters contain the maximum acceptable deceleration for a vehicle and its trailing vehicle on the new lane. The deceleration depends on the distance to the emergency stop position of the next route connector.

- Free lane change if there is more space and a higher speed is required

  For a free lane change, Vissim checks the desired safety distance to the trailing vehicle on the new lane. The desired safety distance depends on the speed of the vehicle that wants to change the lane and on the speed of the vehicle preceding it. You cannot change the degree of "aggressiveness" for free lane changes. You can, however, influence free lane change by changing the safety distance. Safety distances are used to specify car-following behavior (see "Editing the driving behavior parameter Following behavior" on page 286).

For both types of lane change, you first need to find a suitable gap in the direction of travel. The gap size depends on two speeds:

- speed of the vehicle changing the lane
- speed of the vehicle approaching from behind on the lane to be switched to For necessary lane changes, the time gap also depends on drivers' "aggressiveness". Here too the maximum delay of the driving behavior parameters is included in the calculation of the time gaps.

In 2D animation, a current change of lanes, as well as the desire to change lanes is visualized via a small red line to the right or left of the vehicle (representing the indicator), from the defined **Lane change distance** on. This is also the case for lane changes on connectors. In 3D animation, a current lane change and the desire to change lanes is shown via an indicator, if this is defined for the 3D model of the vehicle. The desire to change lanes is triggered by:

- the vehicle route
- in the context of dynamic assignment by the path
- when a desired lane is set via the COM interface

5.8.5.1 Editing the driving behavior parameter Lane change behavior

1. From the **Base Data** menu, choose **Driving Behaviors**.

   *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined.*

   By default, you can edit the list (see "Using lists" on page 93).

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

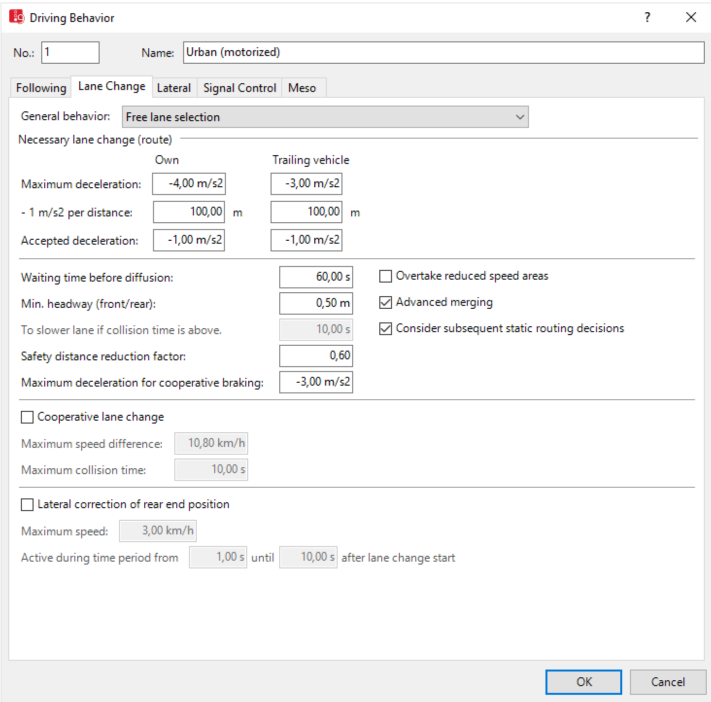
Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Lane Change** tab.



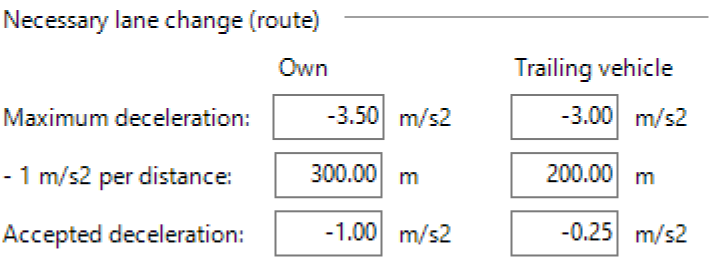
You can edit the already defined network objects in the **Driving Behaviors** list or via the menu **Base Data** > **Driving Behaviors**.

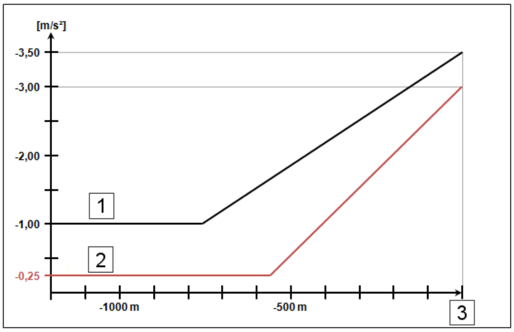
5. Make the desired changes:

Element	Description
General behavior	(**Lane change rule**) (**LnChgRule**): - **Free lane selection**: Vehicles may overtake on each lane. - **Slow lane rule, Fast lane rule**: Allows overtaking on freeways or similar links according to StVO (German Traffic Code) and to the rules in road traffic of other countries. Regardless of the option selected, you can model the general behavior more realistically using the settings under **Cooperative lane change**.

Element	Description
Necessary lane change (route)	- Columns **Own** and **Trailing vehicle**: Delay for changing lanes based on the specified routes for their own overtaking vehicle and the trailing vehicle is accepted by the driver - **Maximum deceleration**: Enter the maximum deceleration for changing lanes based on the specified routes for **own vehicle overtaking** (**MaxDecelOwn**) and the **trailing vehicle** (**MaxDecelTrail**) - **Maximum deceleration:**: Upper bound of deceleration for own vehicle and tailing vehicle for a lane change - **Accepted deceleration** (**AccDecelTrail** and **AccDecelOwn**): Lower bound of deceleration for own vehicle and trailing vehicle for a lane change - **-1m/s2 per distance** (**DecelRedDistTrail** and **DecelRedDistDown**): In addition, the change of the deceleration is specified (in meters per -1 m/s^2). This reduces the **Maximum deceleration** with increasing distance from the emergency stop distance linearly by this value down to the **Accepted deceleration**.

For example, the following parameters yield the course of the curve shown below:





Legend:

1 black line: lane changer (own)

2 red line: trailing vehicle

3: emergency stop distance

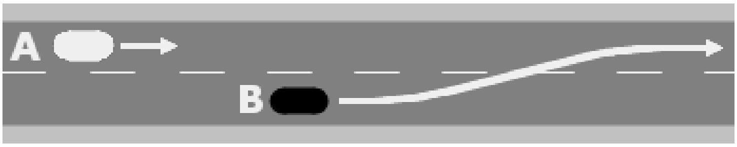
Element	Description
Diffusion time	**DiffusTm**: The maximum amount of time a vehicle can wait at the emergency stop distance for a necessary change of lanes. When this time is reached the vehicle is removed the network, at the same time a warning is written to the **.err* file and displayed in the **Messages** window.

Element	Description
Min. headway (front/rear):	**Minimum headway** (**MinHdwy**):The minimum distance between two vehicles that must be available after a lane change, so that the change can take place (default value 0.5 m). A lane change during normal traffic flow might require a greater minimum distance between vehicles in order to maintain the speed-dependent safety distance.

Element	Description
To slower lane if collision time is above	**Free driving time** (**FreeDrivTm**): only for **Slow lane rule** or **Fast lane rule**: defines the minimum distance to a vehicle in front, in seconds, which must be present on the slower lane, so that an overtaking vehicle switches to the slower lane.

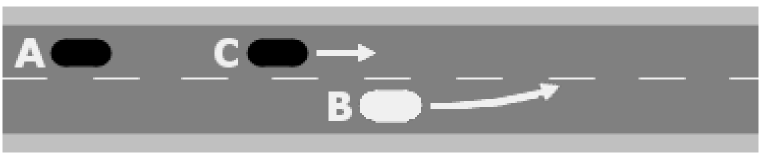
Element	Description
Safety distance reduction factor:	**Safety distance reduction factor (lane change)**, (**SafeDistRedFact**): is taken into account for each lane change. It concerns the following parameters: - The safety distance of the trailing vehicle on the new lane for determ-

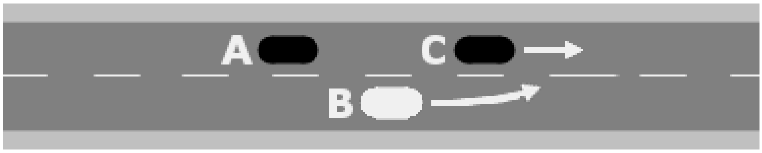
	ining whether a lane change will be carried out - The safety distance of the lane changer itself - The distance to the preceding, slower lane changer During the lane change Vissim reduces the safety distance to the value that results from the following multiplication: *Original safety distance • safety distance reduction factor* The default value of 0.6 reduces the safety distance by 40%. Once a lane change is completed, the original safety distance is taken into account again.

Element	Description
Maximum deceleration for cooperative braking	**Maximum cooperative deceleration** (**CoopDecel**): Specifies to what extent the trailing vehicle **A** is braking cooperatively, so as to allow a preceding vehicle **B** to change lanes into its own lane. When the trailing vehicle **A** detects that it would have to brake more heavily than what this value indicates if the preceding vehicle **B** is set for lane changing, the cooperative braking stops or is not initiated. The higher the value, the stronger the braking and greater the probability of changing lanes. While changing lanes, the preceding vehicle considers the factor for the reduced safety distance and the parameters of the car-following model. Default -3 m/s^2.  During cooperative braking, a vehicle decelerates with the following values: - 0% to a maximum of 50 % of the desired deceleration, until the vehicle in front begins to change lanes (see " Defining acceleration and deceleration behavior" on page 230) - Between 50% of the desired deceleration and the maximum deceleration (100 %) specified in the **Maximum deceleration** field. Typically, the deceleration during the lane change will be considerably less than the maximum deceleration, because the preceding vehicle, which changes lanes, does not expect such a high deceleration from the trailing vehicle.

Element	Description
Overtake reduced	**Overtake reduced speed areas** (**OvtRedSpeedAreas**): The option is not selected by default.

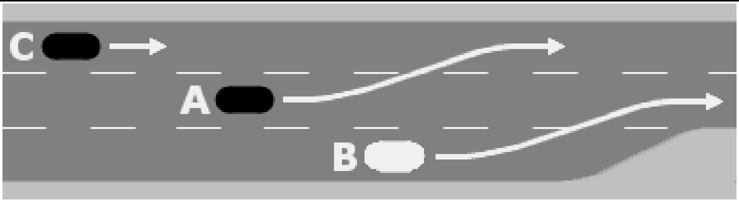
speed areas	☑ If this option is selected, vehicles immediately upstream of a reduced speed area may perform a free lane change (see "Applications and driving behavior parameters of lane changing" on page 300). If there is also a reduced speed area on the lane the vehicle changes to, it is accounted for. ☐ If the option is not selected, vehicles never start a free lane change directly upstream of a reduced speed area. They also completely ignore the reduced speed areas on the new lane.

Element	Description
Advanced merging	**Advanced merging** (**AdvMerg**): This option is selected by default in the driving behaviour parameter sets by newly created networks. The option is considered for any necessary lane change towards the next connector along the route. ☑ If this option is selected, more vehicles can change lanes earlier. Thus, the capacity increases and the probability, that vehicles come to a stop to wait for a gap, is reduced. Select the option accordingly to achieve the desired lane change behavior: If vehicle **A** has to change lanes and recognizes that the neighboring vehicle in front **B** on the target lane has approximately the same speed or is only slightly faster (-1.0 m/s < dv < 0.1 m/s), **A** slows down slightly (by 0.5 m/s²) to move into the gap behind **B**, if the option is selected.  ☐ If the option is not selected, the vehicle **A** slows down only when it approaches the emergency stop distance. If the vehicle **A** with vehicle in front **C** detects that a neighboring vehicle in front **B** wants to change to the lane of **A**, this option can be used so that cooperative braking of **A** also take place when **A** is downstream from **C**.  ☐ If this option is not selected, vehicle **A** leaves the cooperation to its

	preceding vehicle **C**. In this case, **C** might already be too close to **B**, so that **C** overtakes **B**, whereby **A** eventually gets too close to **B** for cooperative braking:  Let us assume vehicle **B** is a neighboring vehicle in front of vehicle **A**. A plans to let B merge, who is meanwhile driving downstream of **C** (in front of vehicle **A**), on its own lane. In this case, vehicle **A** forgets that **B** should have been permitted to merge. Thus, vehicle **A** can immediately permit other vehicles to change into its lane. ☐ If this option is not selected, then within the next 50 m vehicle **A** will not brake for any other neighboring vehicle in front, also if vehicle **B** is downstream of the vehicle in front, **C** (on vehicle **A**’s lane). If vehicle **A** wanted to allow the vehicle ahead of it on the other lane, **B**, to merge, but **B** did not change lanes and **A**, in the meantime, has overtaken **B**, this option allows vehicle **A** to forget that it should have let **B** merge. Thus, vehicle **A** can immediately permit other vehicles to change into his lane. ☐ If this option is not selected, then within the next 50 m vehicle **A** will not brake for any other neighboring vehicle in front, also if **A** has meanwhile overtaken B.

Element	Description
Consider subsequent static rout-ing decisions	**Vehicle routing decisions look ahead** (**VehRoutDecLookAhead**): ☑ If this option is selected, vehicles leaving the route identify new routing decisions on the same link in advance and take them into account when choosing the lane. For routing decisions further downstream that vehicles should identify in advance, the option **Combine static routing decisions** must be selected (see "Attributes of static vehicle routing decisions" on page 468).

Element	Description
Cooperative lane change	**Cooperative lane change** (**CoopLnChg**): If vehicle **A** observes that a leading vehicle **B** on the adjacent lane wants to change to his lane **A**, then vehicle **A** will try to change lanes itself to the next lane in order to facilitate lane changing for vehicle **B**. For example, vehicle **A** would switch from the right to the left lane when vehicle **B** would like to switch to the left from a merging lane to the right lane.



Vehicle **A** behaves during this lane change as if it would have to change lanes due to a connector at a long distance. It accepts its own **Maximum deceleration** and the deceleration of the trailing vehicle **C** on the new lane, in accordance with the parameters for the necessary lane change.

Vehicle **A** does not make a cooperative lane change, when the following conditions are true:

- the new lane is less appropriate for continuing its route
- if vehicle **B** is faster than the maximum speed difference (in the example 10.80 km/h (=3 m/s)
- if the collision time exceeded the maximum collision time (in the example 10 seconds), and the speed of vehicle **A** increased by the maximum speed difference (in the example 10.80 km/h).

- ☑ When you select **Cooperative lane change**, the user-defined cooperative lane change rule is activated for the respective driving behavior parameter set. For **Maximum speed difference** and **Maximum collision time** the user-defined settings are used.
- ☐ If this option is not selected, the user-defined cooperative lane changing behavior is not active for the particular driving behavior parameter set.
- **Maximum speed difference**: If option ☑ **Cooperative lane change** has been selected, the user-defined value for the maximum possible speed difference is taken into account.
- **Maximum collision time**: If option ☑ **Cooperative lane change** has been selected, the user-defined value for the maximum collision time is taken into account.

Element	Description
Cross-correction of the back end	**Rear correction of lateral position** (**RearCorr**): If a lane change takes place at a lower speed than specified in the **Maximum speed** box, the vehicle's rear end moves laterally. The rear correction compensates for this movement. This causes the vehicle to be aligned to the middle of the lane at the end of the lane change, instead of at angle in the original lane. The rear correction is performed completely, even when the vehicle comes to a standstill. A rear correction affects the capacity. Rear correction is only performed if the **Keep lateral distance to vehicles on next lane(s)** option is selected for the driving behavior parameter **Lateral behavior** (see "Editing the driving behavior parameter Lateral behavior" on page 308). - **Maximum speed**: Speed up to which the correction of the rear end position should take place. Default value 3 km/h. Lateral correction of the rear end position is not performed for faster vehicles. - **Active during time period from**: Time after the start of the lane change at which the lateral movement of the rear end position should start, default value 1.0 s. - **until**: Time after the start of the lane change at which the lateral movement of the rear end position should end. The value includes 3 s for the lane change of the front end, default value 10.0. The attributes **Active during time period from** and **to** also determine the speed at which a rear correction is performed.

5.8.6 Editing the driving behavior parameter Lateral behavior

By default, in Vissim a vehicle uses the entire width of the lane. You can define in the driving behavior and parameters-lateral behavior, whether the vehicles in a lane can drive on the left, on the right or in the middle without specifying a lateral orientation. If the lane is wide enough and the attributes in the section **Default behavior when overtaking vehicles on the same lane or on adjacent lanes** allow for overtaking on the same lane, overtaking maneuvers on a single lane are also possible. If the maximum deceleration prevents the overtaking vehicle from braking in time, it overtakes the other vehicle if possible, even if this is not allowed by the driving behavior parameters **Consider next turning direction** and **Minimum lateral distance**. Improper overtaking therefore takes precedence over a collision.

In addition, these settings are used when the option **Observe adjacent lane(s)** is selected.

1. From the **Base Data** menu, choose **Driving Behaviors**.

   *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined.*

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

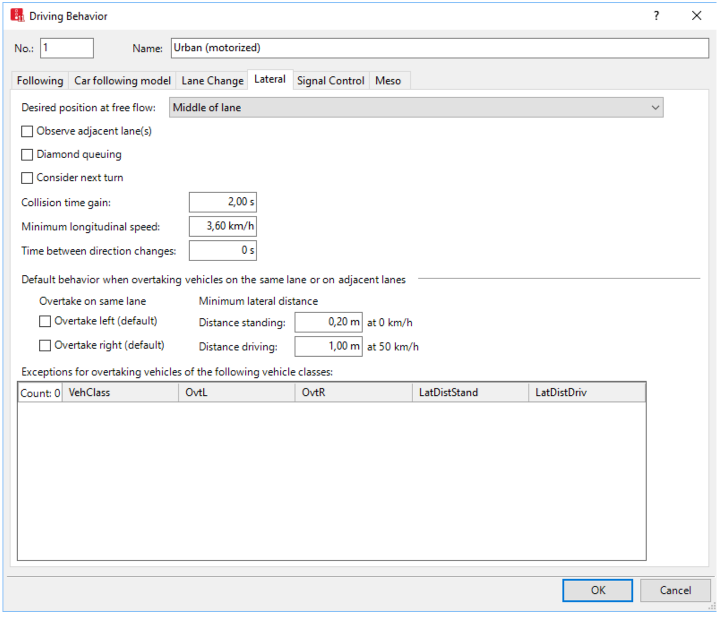
By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Lateral** tab.



5. Make the desired changes:

Element	Description
Desired position at free flow	**Desired position at free flow** (**DesLatPos**): Lateral orientation of a vehicle within its lane while it is in free traffic flow
Observe adjacent lanes	**ObsAdjLns**: ☑ If this option is selected, vehicles account for the position and lateral orientation of other vehicles on adjacent lanes and keep the **Minimum lateral distance**. For this purpose, vehicles even adjust their lateral orientation on their own lane and swerve out of the way. The simulation also regards the actual positions of the back ends of vehicles, which change a lane to an adjacent lane or have already changed. ☐ If this option is not selected, vehicles on adjacent lanes are ignored even if they are wider than their lanes, except when they perform a lane change.

Element	Description
	Note: Using this option can reduce the simulation speed significantly!
Diamond shaped queuing	**DiamQueu**: ☑ If this option is selected, queues take into account a realistic shape of the vehicles with vehicles positioned offset, such as bikes. Vehicles are internally represented not as a rectangle, but as a rhombus.
Consider next turning direction	**ConsNextTurn**: In case of non-lane-bound traffic enables a lateral behavior that accounts for another vehicle as follows: ☑ If the option has been selected, a vehicle with this driving behavior does not pass another vehicle on the same lane if this could cause a collision at the next turning connector. To achieve this, attributes that enable passing on the same lane must be selected in the section **Default behavior when overtaking vehicles on the same lane or on adjacent lanes** and/or vehicle classes that may be overtaken must be selected in the section **Exceptions for overtaking vehicles of the following vehicle classes**. Option **Consider next turning direction** also considers the attribute **Desired Direction** of the next connector of the route of the vehicle (see "Attributes of connectors" on page 422). If, for example, **left** has been selected for this attribute, the vehicle only passes another vehicle on the right if that vehicle turns at the same connector at the latest. If the vehicle is within the lane change distance defined in the **Lane change** attribute (**Lane change distance**), it moves laterally on its lane to the respective side if there is sufficient space. Simultaneously, vehicles which do not want to turn in the same direction on the same connector or some time before it, do not try to overtake the vehicle on that side, since this would cause a collision at the next intersection. The vehicle flashes if the option **Consider next turning direction** is selected and the vehicle changes lanes within the **Lane change distance** defined in the **Lane change** attribute and the attribute **Left** or **Right** of the connector is selected. Note: The option **Consider next turning direction** has precedence over option **Desired position at free flow**.
Collision time gain	**Minimum collision time gain** (**MinKCollTimeGain**): Minimum value of the collision time gain for the next vehicle or signal head, which must be reached so that a change of the lateral position on the lane is worthwhile and will be performed. The collision time is calculated based on the desired speed of the vehicle. The default value for collision time gain is two seconds. Smaller values lead to a livelier lateral behavior, since vehicles also have to dodge sideways for minor improvements.

Element	Description
Minimum longitudinal speed	**Minimum longitudinal speed for lateral movement** (**MinSpeedForLat**): Minimum longitudinal speed which still allows for lateral movements. The default value of 1 km/h ensures that vehicles can also move laterally if they have almost come to a halt already.
Time between direction changes	**Lateral direction change - minimum time (Lateral behavior) (LatDirChgMinTm**): Standard 0.0 s: defines the minimum simulation time which must pass between the start of a lateral movement in one direction and the start of a lateral movement in the reverse direction. The higher this value, the smaller are the lateral movements of vehicles. These lateral movements only take place if overtaking on the same lane is permitted. Lateral movement for a lane change is not affected by this parameter.

Element	Description

Default behavior when overtaking vehicles on the same lane or on adjacent lanes	This applies for all vehicle classes, with the exception of the vehicles classes listed under **Exceptions for overtaking vehicles of the following vehicle classes**.
	- **Overtake on same lane**: When modeling traffic that is not lane-bound, you can allow vehicles to overtake within a lane.
	  - **Left**: Vehicles are allowed to overtake on a lane to the left
	  - **Right**: Vehicles are allowed to overtake on a lane to the right
	- **Minimum lateral distance**: Minimum distance between vehicles when overtaking within the lane and keeping the distance to vehicles in the adjacent lanes, default value 1 m.
	  - **Distance standing at 0 km/h** (**LatDistStandDef**): lateral distance of the passing vehicle in meters. Default value: 1 m
	  - **Distance driving at 50 km/h** (**LatDistDrivDef**): lateral distance of the passing vehicle in meters. Default value: 1 m
	The minimum distance is linearly interpolated for other speeds than at 0 km/h and 50 km/h.
	If the option **Keep lateral distance to vehicles on next lane(s)** is not selected, vehicles on adjacent lanes are ignored, even if they are wider than their lanes, except when they change lanes.

Exceptions for overtaking vehicles of the following vehicle classes	Behavior for specific vehicle classes that deviates from the default behavior when overtaking vehicles on the same lane or on adjacent lanes. When modeling traffic that is not lane-bound, you can select vehicle classes which may be overtaken within a lane by vehicles of this driving behavior set.
	1. Right-click in the list.
	2. From the shortcut menu, choose **Add**.
	A new row with default data is inserted.
	3. Make the desired changes:
	  - **VehClass**: Vehicle class whose vehicles may be overtaken by vehicles of this driving behavior parameter set within the lane.
	  - **OvtL** (**Overtake left**): Vehicles are allowed to overtake on the left lane
	  - **OvtR** (**Overtake right**): Vehicles are allowed to overtake on the right lane
	  - **LatDistStand**: Minimum distance at 0 km/h
	  - **LatDistDriv**: Minimum distance at 50 km/h

5.8.6.1 Example of modeling lateral behavior

The example takes into account the following guidelines:

- Bikes and cars travel on the same one-lane link.
- Bikes must drive on the right side.
- Bikes may be overtaken by cars only on the left.
- Bikes may overtake cars only on the right.
- Bikes may overtake other bikes only on the left.

For this, you define three driving behavior parameter sets:

Defining the driving behavior parameter set Urban lateral behavior

1. From the **Base Data** menu, choose **Driving Behaviors**.

   *The **Driving Behaviors** list opens. Some driving behavior parameter sets can be predefined.*

2. Right-click **Urban (motorized)**.
3. From the shortcut menu, choose **Duplicate**.
4. For the new driving behavior parameter set, in the **Name** box, enter: `Urban lateral behavior`
5. Right-click the entry.
6. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior Parameter Set** window opens.*

7. Make the desired changes:

Element	Description
Following tab	**Look ahead distance**: **min.**: 0 **max.**: 30 m
Lateral tab	Section **Exceptions for overtaking vehicles of the following vehicle classes**: 1. From the shortcut menu, choose **Add**. *A new row is inserted.* 2. Select the vehicle class **Bike**. 3. Select the option **OvtL**.

4. Confirm with **OK**.

Defining the driving behavior parameter set Urban Bike

1. In the **Driving behavior** list, right-click **Cycle-Track (free overtaking)**.
2. From the shortcut menu, choose **Duplicate**.
3. For the new driving behavior parameter set, in the **Name** box, enter: `Urban Bike`

4. Right-click the entry.
5. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior Parameter Set** window opens.*

6. Make the desired changes:

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Lateral tab</td><td>Desired position at free flow:Right
Section Default behavior when overtaking vehicles on the same lane or on adjacent lanes:
1. Under Overtake on same lane, deactivate the options On left and On right.
2. From the shortcut menu, choose Add.
A new row is inserted.
3. Select the vehicle class Car.
4. Select the attribute OvtR.
5. Right-click the entry.
6. From the shortcut menu, choose Add.
A new row is inserted.
7. Select the vehicle class Bike.
8. Select the attribute OvtL.
9. In the LatDistStand section, select: 0.3 m</td></tr>
</table>

10. Confirm with **OK**.

Defining the link behavior type Urban lateral behavior Bike

1. Select from the menu **Base Data** > **Link Behavior Types**.

   *The list **Link Behavior Types** opens. Some link behavior types can be predefined.*

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. For the new link behavior type, in the **Name** column, enter: `Urban lateral behavior Bike`.
4. In the column **DrivBehavDef**, select the driving behavior parameter set **Urban Bike**.
5. Assign the applicable links in the **LinkBehaviorType** attribute to the new **Urban lateral behavior Bike** link behavior type.

5.8.7 Editing the driving behavior parameter Signal Control

For the driving behavior at signal controls, specify the following:

- how vehicles respond to amber signal
- how vehicles respond to red-amber signal
- a reduced safety distance before stop lines
- a time distribution for the response time

1. From the **Base Data** menu, choose **Driving Behaviors**.

   The list of defined network objects for the network object type opens.

   The list shows driving behavior parameter sets. Some driving behavior parameter sets can be predefined.

   By default, you can edit the list (see "Using lists" on page 93).

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Signal Control** tab.
5. Make the desired changes:

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Reaction to amber signal</td><td>
Decision model: Defines the behavior of vehicles when they approach an amber light.

Continuous check: Driver of vehicle continuously decides whether to continue driving or whether to stop. Vehicles assume that the amber light will only be visible for another two seconds. They then decide continuously, with each time step, whether they will continue to drive or stop.

A vehicle will not brake, if its maximum deceleration does not allow it to stop at the stop line, or if it would have to brake for longer than 4.6 m/s².
The vehicle will brake, if at its current speed, it cannot drive past the signal head within two seconds.
Both braking and stopping are possible for cases that lie in between these two scenarios. Using a normally distributed random variable, Vissim decides whether or not the driver will brake.

One decision: The decision made is maintained until the vehicle crosses the stop line. To calculate the probability p, i.e. whether a driver stops at an amber light or not, the program uses a logistic regression function, with the following parameters Alpha, Beta1, Beta2, vehicle speed v and distance to stop line dx:

$$p = \frac{1}{1+e^{-\alpha-\beta_1 v-\beta_2 dx}}$$

The default values of the Probability factors Alpha, Beta1, Beta2 are based on empirical data:

Alpha: default 1.59
Beta1: default -0.26
Beta2: default 0.27

The decision made is maintained until the vehicle crosses the stop line.
To produce the most accurate results, select the One decision option. To do so, adjust the number of Observed vehicles accordingly for the look ahead distance (see "Editing the driving behavior parameter Following behavior" on page 286). As signal heads (and some other network objects as well) are modeled internally as vehicles, they are only recognized if the number of vehicles or network objects between the vehicle in question and the signal head does not exceed the number of Observed vehicles minus 1.
The following settings make a vehicle continue driving for longer when there is an amber liight and occasionally even make it run a red light:

The One decision option is selected
Alpha is greater than the default value 1.59
Beta2 is greater than the default value -0.26 but less than 0.00.
Beta1 is greater than the default value 0.27

</td></tr>
</table>

6. Make the desired changes:

Element	Description
Behavior at red/amber signal	Modeling country-specific or regional behavior at red/amber signal. - **Stop (same as red)** - **Go (same as green)**
Reduced safety distance close to a stop line	Defining the behavior of vehicles close to a stop line. - If a vehicle is located in an area between **Start upstream of stop line** and **End downstream of stop line**, the factor is multiplied by the safety distance of the vehicle. The safety distance used is based on the car following model. The safety distance may be reduced via the **Safety distance reduction factor** attribute (see "Editing the driving behavior parameter Lane change behavior" on page 300). For lane changes in front of a stop line, the two values calculated are compared. Vissim will use the shorter of the two distances. - **Start upstream of stop line**: Distance upstream of the signal head - **End downstream of stop line**: Distance downstream of signal head
Reaction time distribution	Reaction time of a vehicle to the **Go** signal. It causes a time delay between the time step when the signal switches to **Go** and the time step when the first vehicle upstream of the corresponding stop line starts to move. The **Go** signal is defined by the **Behavior at red/amber signal** attribute: - **Stop (same as red)**: The **Go** signal is green. The response time is effective from the time step the signal changes to green. - **Go (same as green)**: The **Go** signal is red-amber. The response time is effective from the time step the signal changes to red-amber. If no time distribution is selected, the default time is 0 s.

5.8.8 Editing the driving behavior parameter Meso

Mesoscopic simulation uses a simplified vehicle following model (see "Car following model for mesoscopic simulation" on page 803) for modeling vehicle behavior.

1. From the **Base Data** menu, choose **Driving Behaviors**.

   The list of defined network objects for the network object type opens.

   The list shows driving behavior parameter sets. Some driving behavior parameter sets can be predefined.

   By default, you can edit the list (see "Using lists" on page 93).

You can edit all driving behavior parameters for lane change, lateral behavior and following behavior in the list or in tabs with the following steps.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Driving Behavior** window opens.*

4. Select the **Meso** tab.

5. Make the desired changes:

Long name	Short name	Description
Meso reaction time	**MesoReactTime**	Temporal safety distance [s] (response time), default value 1.20 s
Meso standstill distance	**MesoStandDist**	Meso standstill distance of vehicles, default value 2.00 m. **Meso standstill distance** + **vehicle length** = *effective vehicle length for mesoscopic simulation* The vehicle length depends on the vehicle type.
Meso maximum wait time	**MesoMaxWaitTime**	**Meso maximum waiting time:**: Period after which a vehicle waiting at the node entry enters the node from a minor flow direction, even if the time gap in the major flow direction is too short. This way, a minimum number of vehicles of the minor flow direction get to enter the node, despite the heavy traffic in the major flow direction. Default 120 s, value range 0 s to 100,000 s.

5.9 Defining link behavior types for links and connectors

Using a link behavior type, you can assign the desired type of driving behavior per vehicle class to a link or connector. For example, you define the link behavior type **Slow lane rule in conurbations** and assign it the corresponding default driving behavior **Slow lane rule (motorized)**. Then in the coupled list **Driving behavior**, you restrict the link behavior type **Slow lane rule in conurbations** to the vehicle class **Bus**.

Note: When you open a network file of a Vissim version that is older than Vissim 5.0, the following steps are automatically carried out:

- The link types defined are used to generate link behavior types and display types that are then assigned to links.
- Connectors are assigned the link behavior type and display type of their origin link.

1. Select from the menu **Base Data** > **Link Behavior Types**.

   *The list **Link Behavior Types** opens. Some link behavior types can be predefined.*

   Note: Defined Vissim licenses can be limited to a maximum of two link behavior types.

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Enter the desired values.

Element	Description
No	Unique number of the link behavior type
Name	Identification of the link behavior type
DrivBehavDef	**Default driving behavior**: driving behavior parameter set for driving class for the link behavior type. The vehicle classes whose vehicles use the links of the type, can be allocated different parameter sets (see "Defining driving behavior parameter sets" on page 282).

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **VehClass**: Vehicle class for which the selected driving behavior applies on the link or connector
- **Driving behavior**: For the vehicle classes of your choice, select a driving behavior that differs from the **default driving behavior**

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

5.10 Defining display types

You can define display types. They specify the representation of network objects in the Vissim network, e.g. their fill style, fill color, border line style, border line color or texture. You then assign the desired display type to a network object in the **Display Type** attribute, e.g. the display type `Road gray` to a link.

In Vissim, display types are defined for road, rail, pedestrian areas, obstacles and sections, as well as for elements of escalators and elevators.

When you display network objects in the network editor, the display type settings have priority over the graphic parameter settings for network objects (see "List of graphic parameters for network objects" on page 161).

The add-on module Viswalk allows you to show the following construction elements and specify their display type: areas, obstacles, ramps and stairs, and their display types.

Note: When you open a network file of a Vissim version that is older than Vissim 5.0, the following steps are automatically carried out:

- The link types defined are used to generate link behavior types and display types that are then assigned to links.
- Connectors are assigned the link behavior type and display type of their origin link.

---

1. Select from the menu **Base Data** > **Display Types**.

   *The **Display Types** list opens. Some display types can be predefined.*

   By default, you can edit the list (see "Using lists" on page 93).

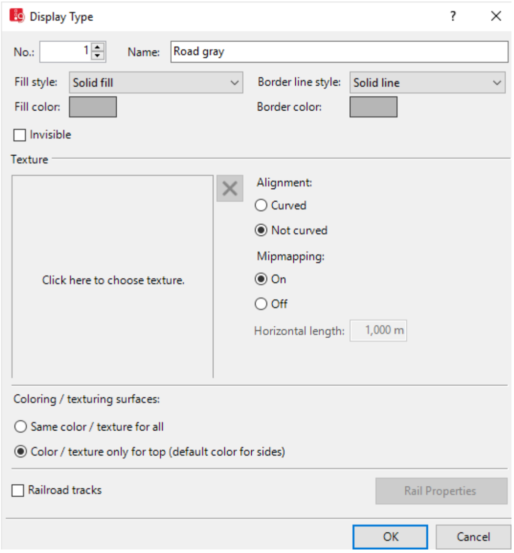
Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

   *The **Display Type** window opens.*



In the **Display Type** window, you enter attribute values. For display types already defined, you can open this window via the **Display Types** list, using the following functions:

- In the **Display Types** list, double-click the row with the desired display type.

The display type may have additional attributes. You can show all attributes and attribute values in the **Display Types** list. You can open the list via the following functions:

- From the **Lists** menu, choose > **Base Data** > **Display Types**.
- Select from the menu **Base Data** > **Display Types**.

You can edit attributes and attribute values in the lists (see "Selecting and editing data in lists" on page 100).

Element	Description
No	Unique number of display type
Name	Name of display type
Invisible	☑ If this option is selected, the display of links and construction elements is limited. - In 2D mode the edge is shown as a dashed line in the color of the display type assigned to the link or the construction element. - in 2D mode hidden during simulation - in 3D mode hidden If vehicles and/or pedestrians are moving on the links and construction elements, they are shown.

Element	Description
Fill style	- **No fill**: show outline only. You cannot select a fill color. - **Solid fill**: show color between outline. Select the color in the **Fill color** box.
Fill color	Color between the outline of links, connectors and construction elements in the network. The graphic parameter **Use display type** of the network object type must be selected. The color is not accounted for in the Wireframe mode.
Border line style	- **No line**: do not show outline. You cannot select an outline color. - **Solid line**: show outline as colored line. Select the color in the **Border color** box.
Border color	Color between outline border of links, connectors and construction elements in the network. The graphic parameter **Use display type** of the network object type must be selected. The color is not accounted for in the Wireframe mode.
Texture	**Texture filename** (**TextureFilename**) In the **Texture** box, select the desired graphic file for display of the link in 3D mode. If a texture is selected, the content of the graphic file of the texture is displayed in the **TextureFilename** column in the **Display Types** list. For textures, graphic files are available in the formats **.jpg* and **.bmp*. By default, they are saved to the directory *..\3DModels\Textures* of your Vissim installation: - *..\3DModels\Textures\Material*: Surfaces of different materials - *..\3DModels\Textures\roads*: Surfaces of roads - *..\3DModels\Textures\signs*: Traffic signs - *..\3DModels\Textures\Signal Head Picto grams*: pictograms for signal heads
Horizontal length	Scales texture to length entered.
Alignment	- **Follow link curvature**: If required, display of the texture is adjusted to the link curvature or connector along the middle line. This is useful, for example, for labeling on the road. - **Do not follow link curvature**: Texture display is not adjusted.
Anisotropic filtering	Only as **AnisoFilt** column in **Display types** table: If this option is selected in the table, the display quality of textures is improved when viewed from a very flat angle. Make sure that in the Control Panel of your computer, in the driver settings for your graphic card, under **Anisotropic filtering**, you select **Application-controlled** or **Use Application Settings**.

Element	Description
Follow link curvature	**Curved**: in 3D mode: - If the option is selected, the texture on the links of this display type is arranged along the center line of the link. This is useful, for example, for labeling on the road. - If the option is not selected, the orientation of the texture is the same for all links, regardless of their curvature. Thus no edges are visible for overlapping links and overlapping connectors.
Mipmapping	- **Selected** (**No Mipmap**) : The texture in the distance is displayed as more blurred. Thus for example, asphalt without markings seems more realistic. - **Deselected** (**No Mipmap**) : The texture is also displayed with maximum resolution at a greater distance from the viewer. This is useful, for example, for labeling on the road.
Coloring / texturing surfaces	- **Same color / texture for all**: Lateral areas are displayed in the same texture as the top. - **Color / texture only for top (default color for sides)**: Lateral areas are displayed in same fill color as the top This also applies when a texture is selected. - **Shaded** : If the option is selected, lateral areas are shaded.
Railroad tracks	**Rail**: If the option is selected, on the link, tracks are displayed in 3D mode. To define the display of tracks and ties, click the **Rail Properties** button.

You can also define rail properties (see "Defining track properties" on page 323).

5.11 Defining track properties

1. Select from the menu **Base Data** > **Display Types**.

   *The **Display Types** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Display Type** window opens.*

4. Select the option **Railroad tracks**.

5. Click the **Rail Properties** button.

   *The **Rail Properties** window opens and shows a preview. If after making changes, you click in the preview, the display is refreshed accordingly.*

6. Make the desired changes:

Element	Description
Rail type	- **None**: do not display tracks - **Default**: Show tracks in default texture
Ties type	- **Embedded**: Tracks are embedded in ties - **None**: do not display ties - **Default**: Tracks are mounted on ties

7. Enter the desired values in the **Rail** tab.

Element	Description
Rail gauge	Distance between inner edges of tracks, default value 1.435 m
Rail height	Flange height **+** head height **+** web thickness calculated
Head width	Width of upper track portion
Head height	Height of upper track portion
Web width	Width of middle track portion
Flange width	Width of lower track portion
Flange height	Height of lower track portion

8. Enter the desired values in the **Ties** tab.

Element	Description
Spacing	Distance between individual ties
Length	Length of sleepers (90° towards movement direction)
Width	Width of ties in movement direction
Height	Vertical thickness of ties type
Texture	Graphic file for display of ties
Horizontal length	Scales texture to length entered

9. Confirm with **OK**.

5.12 Defining levels

You may define multiple levels, e.g. for multistory buildings or bridge structures for links. For levels, you can define links, backgrounds, static 3D models, 3D signal heads and construction elements.

By default, Vissim already contains a level with the **Height** attribute = 0.0.

1. From the **Base Data** menu, choose **Levels**.

*The **Levels** list opens.*

By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button ✚ .

   A new row with default data is inserted.

3. Make the desired changes:

Element	Description
No	Unique number of the level
Name	Name of the level
zCoord	**z-coordinate:** z value of layer in meters

*The level is displayed in the **Levels** list and in the Levels toolbar.*

Tip: Using the Levels toolbar, you can show and hide levels and activate or deactivate their selectability (see "Using the Level toolbar" on page 65).

5.13 Using time intervals

You may define time intervals for the following network object types (see "Defining time intervals for a network object type" on page 326):

- Vehicle routes (parking)
- Partial Vehicle Routes
- Vehicle routes (static)
- Vehicle inputs
- Area behavior types
- Pedestrian routes (partial)
- Pedestrian routes (static)
- Pedestrian inputs
- Managed lanes
- Partial PT routes

To define new time intervals for one of these network object types or to edit defined time intervals, in the attribute list of network objects of this network object type, call the **Time intervals** list (see "Calling time intervals from an attributes list" on page 327).

5.13.1 Defining time intervals for a network object type

In Vissim, a time interval is predefined with the default values **0.00** s up to **MAX** simulation period (see "Defining simulation parameters" on page 840). This time interval is the default used for network object types that can be assigned time intervals (see "Using time intervals" on page 325). You may define additional time intervals for each of these network object types. These time intervals are then only valid for the network objects of this network object type.

1. Select from the menu **Base Data** > **Time Intervals**.

   The ***Time intervals*** *list opens. By default the time interval* ***0.00*** *s up to* ***MAX*** *simulation period is shown. If you do not add another time interval, you can only edit the start time of this interval - not the end.*

   By default, you can edit the list (see "Using lists" on page 93).

   On the toolbar of the ***Time intervals*** *list, the* ***Relation <network object type>*** *list box displays the network object types for which you can define time intervals.*

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

Element	Description
Start	Start time of the time interval in seconds. As at least one interval must be defined, the entries in the first and last row cannot be deleted. The limiting value must not be larger or smaller than the current highest value, but must not be the same as an existing value. In the case of a smaller value, an interval which has already been defined is divided. If you change an entry which you have chosen in the list, the new value must be greater than the preceding value and smaller than the following value in the list. Otherwise, delete the chosen entry and add a new entry, as the sequence of the entries cannot be changed directly.
End	End of the time interval in seconds. The last time interval always ends with **MAX**.

4. Enter the desired values.

5. Repeat the steps until you have defined the desired time intervals for the selected network object type.

Tip: In the attribute list of a network object type that can be assigned time intervals, you can access the **Time intervals** list and edit the respective time intervals (see "Calling time intervals from an attributes list" on page 327).

5.13.2 Calling time intervals from an attributes list

1. Open the attributes list of the desired network object.
2. In the list, right-click the network object of your choice.
3. From the shortcut menu, choose **Edit Time Intervals**.

   *The **Time intervals** list opens. The **Relation <Network object type>** list shows the network object type for which time intervals have been defined in the **Time intervals** list.*
4. Edit the desired entries.

5.14 Toll pricing and defining managed lanes

You can define managed lanes in the **Managed Lanes Facilities** list (see "Defining managed lane facilities" on page 327). You may assign each managed lane the following objects:

- A decision model with the attributes **cost coefficient**, **time coefficient** and **base utility**. In doing so, you can distinguish between individual vehicle classes. The decision model determines the probability of a vehicle actually using the managed lane (see "Defining decision model for managed lane facilities" on page 329).
- Toll pricing calculation models for each time interval with pricing models Based on the number of occupants, the toll pricing model determines the toll, travel time saving and average speed (see "Defining toll pricing calculation models" on page 331).

On the desired link sequence, you define the vehicle routes **Managed** and **General purpose** of the type **Managed Lanes** (see "Defining a vehicle route of the type managed lane" on page 476). In the **Managed Lanes Routing Decisions** list, assign the Managed lanes routing decision of this vehicle route the managed lanes facility of your choice (see "Attributes of managed lanes routing decisions" on page 478).

Toll pricing is not taken into account during dynamic assignment.

5.14.1 Defining managed lane facilities

1. Choose **Managed Lanes Facilities** from the menu **Traffic**.

   *The **Managed Lanes Facilities** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

This line contains the decision model default values for the cost coefficient, the time coefficient and base utility.

3. Enter the desired name.
4. If desired, edit further attribute values in the row.

Element	Description
UpdInt	The update interval specifies how often the travel times and therefore travel time savings and average speed, as well as toll charges are recalculated. Travel times and tolls apply to the current update interval and will only be recalculated when a new update interval begins. For the recalculation of the toll charge, apart from the update interval, user-defined time can be considered: After recalculation at a user-defined time, the update interval will be used again for the next recalculation (see "Modeling vehicle inputs for private transportation" on page 454).

The parameters **LogitA**, **CostCoeffDef**, **TmCoeffDef**, **BaseUtilDef** are included in the Logit model. The Logit model is used as a basis for calculating the probability of a decision to use a managed lane (see "Defining decision model for managed lane facilities" on page 329).

5. Change the values **LogitA**, **CostCoeffDef**, **TmCoeffDef** and **BaseUtilDef** depending on your use case and the length of the managed lane route:

Element	Description
LogitA	**Logit alpha**: default value: 0.05. This value applies to all vehicle classes of the decision model. Examples: Using *Logit alpha = 0.05*, a difference of 20 between the two routes results in a probabilities ratio for choosing the routes of *1:2.718 (1:e)*. Using *Logit alpha = 0.05*, a difference of 40 between the two routes results in a probabilities ratio for choosing the routes of *1:7.389* $(1:e^2)$. Using *Logit alpha = 0.10*, a difference of 20 between the two routes results in a probabilities ratio for choosing the routes of *1:7.389* $(1:e^2)$.
CostCoeffDef	**Cost coefficient (default)**: value is optional depending on vehicle class. Use value ≤ 0 to model the impact of the charged cost. Default value -1.00.
TmCoeffDef	**Time coefficient (default)**: value is optional depending on vehicle class. Use value ≥ 0 to model the impact of the travel time saving. Default value 0.40.
BaseUtilDef	**Base utility (default)**: value is optional depending on vehicle class. Default 0.0

Notes:

- The ratio of the coefficients **Utility Coefficient Toll** and **Utility Coefficient Time** is decisive for modeling real traffic conditions. With the default values **Utility Coefficient Toll** = -1 and **Utility Coefficient Time** = 0.4, one monetary unit is worth as much as a travel time saving of 2.5 min.
- The default values are used for vehicles of a type, which does not belong to the specified vehicle classes. If a vehicle type belongs to several of the specified vehicle classes, the values used are those for the smallest vehicle class belonging to the vehicle type.
- If the travel time on the managed lane route is greater than on the general purpose route, the result is a negative travel time saving. In this case, the travel time saving = 0 is used.

In the next steps, in the list on the left, you can select a managed lane. In the list on the right, edit the objects assigned to it or you can assign objects depending on the relation selected.

6. On the list toolbar, in the **Relations** list, click the desired entry:

- **Decision models** (see "Defining decision model for managed lane facilities" on page 329): Add desired vehicle classes, edit coefficients and base utility.
- **Pricing models by time interval** (see "Defining toll pricing calculation models" on page 331): Edit toll and pricing models based on the number of occupants.
- Edit **Managed lanes routing decisions** (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459): Under Managed Lanes Routing Decisions, **Managed lanes facility** attribute, a managed lane facility must be assigned.

7. In the list on right, on the toolbar, click the **Add** button ✚.

A new row with default data is inserted.

8. If desired, edit further attribute values in the row.

The data is allocated.

Note: When you delete a managed lane facility which is associated with a routing decision, the routing decision is no longer complete. The routing decision will not be deleted. However, the routing decision cannot anymore be considered in the simulation.

5.14.1.1 Defining decision model for managed lane facilities

The decision model determines the actual probability that a vehicle uses the managed lane. This depends on the current utility of the managed lane.

The managed lane’s utility U is calculated according to the following formula:

U(Toll) = Cost coefficient ● Toll rate + Time coefficient ● Time gain + Base utility

Thereby the time gain is the difference between the travel time on the general purpose route and the travel time on the managed lane determined during the last update interval.

The utility of the general purpose route is always zero, since there is neither a toll, nor time gain when compared to itself:

U(general purpose) = 0

The probability of deciding to use the managed lane is calculated according to a Logit model, which applies the following equation:

$$P_{(Toll)} = 1 - \frac{e^{a^{*}U_{Toll\text{-}free}}}{e^{a^{*}U_{Toll\text{-}free}} + e^{a^{*}U_{Toll}}} = 1 - \frac{1}{1+e^{a^{*}U_{Toll}}}$$

1. Choose **Managed Lanes Facilities** from the menu **Traffic**.

   *The **Managed Lanes Facilities** list opens.*
2. Select the required managed lane facility from the left list.
3. On the list toolbar, in the **Relations** list, click > **Decision models**.
4. Right-click on the row header in the right-hand list.
5. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

6. Make the desired changes (see "Defining managed lane facilities" on page 327).

The data is allocated.

5.14.1.2 Example: Using the decision model to calculate a managed lane

The ratio of the coefficients **Utility Coefficient Toll** and **Utility Coefficient Time** is decisive for modeling real traffic conditions. With the default values **Utility Coefficient Toll** = -1 and **Utility Coefficient Time** = 0.4, one monetary unit is worth as much as a travel time saving of 2.5 min.

If **Logit alpha** = 0.05: Increasing the base utility by 20 increases the ratio of probabilities for and against the managed lane by factor **e** (2.718):

- With **Logit alpha** *= 0* and a *base utility = 40* for HOV (heavy occupancy vehicles) vehicles, for each minute saved by using the managed lanes, the share of HOV vehicles on the normal lane is divided by 7. For two minutes the result is *1/e^4* = about *1/55*.

- When you wish to model that most of the HOV vehicles already use the managed lane though the time saving is not known yet, set the base utility accordingly: A base utility of 20, for example, results in that even with utility = 0 (unknown travel time gain) only $1/(1+e)$ (about 27%) of the vehicles will travel on the normal lane.

Alpha ● Utility	Managed lane probability
2	87 %
1.5	82 %
1	73 %
0.5	62 %
0	50 %
- 0.5	38 %
- 1.0	27 %
- 1.5	18 %
- 2.0	13 %

If you increase the **base utility** by 10, with **Alpha = 0.05**, the value of **Alpha ● Utility** increases by 0.5. The probability increases accordingly.

5.14.2 Defining toll pricing calculation models

You can assign pricing models by time interval to managed lanes as a relation (see "Defining managed lane facilities" on page 327).

Function of toll pricing calculation model

To determine toll pricing, you can use a toll pricing calculation model. The toll pricing calculation model can determine the toll depending on the travel time saving and/or average speed on the managed lanes.

Example:

- You want the toll to be 0, if the travel time saving is less than 5 minutes.
- You want the toll to be 5, if the travel time saving is greater than 5 minutes and smaller than 15 minutes.
- You want the toll to be 7, if the travel time saving is greater than 15 minutes and the average speed on the managed lanes is slower than 80 km/h.
- You want the toll to be 10, if the travel time saving is greater than 15 minutes and the average speed on the managed lanes is faster than 80 km/h.

Pricing model of toll pricing calculation model

Each toll pricing calculation model contains a price model. The toll price model determines when and how the managed lane facility calculates the toll charge. For this the occupancy rate of the vehicle is also relevant.

The following occupancy rates are predefined as attributes in the pricing model by time interval:

Occupancy rate	Vehicle occupancy	Vehicle occupants
TollSOV	a vehicle occupant or, if an autonomous vehicle is empty, no vehicle occupant	Driver of non-autonomous vehicle
TollHOV2	two vehicle occupants	driver and one passenger
TollHOV3Plus	three or more vehicle occupants	driver and several passengers

During the simulation, the vehicle occupancy is derived from the occupancy rate of the vehicle type.

Since the vehicle occupancy is always a whole number, the following is valid for the calculation:

- From an occupancy rate of **Toll1** for vehicle type A is derived that all vehicles of type A are occupied by only one person. If an autonomous vehicle is empty, there is no vehicle occupant.
- From an occupancy rate of **Toll1.4** for vehicle type B is derived that 60% of all vehicles of type B are occupied by only one person and 40% by two persons.

1. Select **Toll Pricing Calculation Models** from the menu **Traffic**.

   *The **Toll Pricing Calculation Models** list opens.*

2. In the list, on the toolbar, click the **Add** button ✚.

   A new row with default data is inserted.

3. Enter a number and name.
4. On the list toolbar, in the **Relations** list, click > **TollPricingCalculationModelElement**.
5. In the list on right, on the toolbar, click the **Add** button ✚.

   A new row with default data is inserted.

6. Make the desired changes:

Element	Description
Position	Position of toll pricing calculation model element in the list
TravTmSavFrom, TravTmSavTo	Range of travel time saving with managed lanes compared to use of toll free lanes
Operator	Arithmetically connects the elements of toll pricing calculation model (**travel time saving** and **average speed**) using AND or OR.

Element	Description
AvgSpeedFrom, AvgSpeedTo	Range of average speed on managed lanes
Toll	Toll costs. For fixed price = 0.0 no toll is charged. Also a user-defined toll pricing calculation model can result in a toll fee of 0.0.

Notes:

- Toll = 0.0 does not automatically mean that all vehicles choose this managed lane.
- If you delete a toll pricing calculation model which is still assigned to a managed lanes facility, a constant toll of 0.0 is used.

The toll charge is calculated according to the selected toll pricing calculation model at each managed lanes facility for all three occupancy rates and is valid until the next update time. The update time of the managed lane facilities on the network needs not be identical.

6 Creating and editing a network

In the Network editor, you model a Vissim network with network objects. The following network object types are available for this:

Icon	Network object type
	Links and Connectors (see "Modeling links for vehicles and pedestrians" on page 406), (see "Modeling connectors" on page 420)
	Desired Speed Decisions (see "Modeling links for vehicles and pedestrians" on page 406)
	Reduced Speed Areas (see "Using reduced speed areas to modify desired speed" on page 435)
	Conflict Areas (see "Using conflict areas" on page 560)
	Priority Rules (see "Modeling priority rules" on page 541)
	Stop Signs (see " Modeling stop signs and toll counters" on page 571)
	Signal Heads (see "Modeling signal groups and signal heads" on page 578)
	Detectors (see "Using detectors" on page 593)
	Vehicle Inputs (see "Modeling vehicle inputs for private transportation" on page 454)
	Vehicle Routes (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459)
	Vehicle Attribute Decisions (see "Using vehicle attribute decisions" on page 506)
	Parking Lots (see "Modeling parking lots" on page 493)
	Public Transport Stops (see "Modeling PT stops" on page 511)
	Public Transport Lines (see "Modeling PT lines" on page 518)
	Nodes (see "Modeling nodes" on page 705)
	Data Collection Points (see "Defining data collection points" on page 446)
	Vehicle Travel Times (see "Defining vehicle travel time measurement" on page 447)
	Queue Counters (see "Modeling queue counters" on page 450)
	Flow bundles (see "Visualizing volumes on paths as flow bundles" on page 766)
	Sections (see "Modeling sections" on page 677)
	Background Images (see "Inserting a background image" on page 394)
	Pavement Markings (see "Modeling pavement markings" on page 443)
	3D Traffic Signals (see "Modeling 3D signal heads" on page 584)

Icon	Network object type
	Static 3D Models (see "Using static 3D models" on page 674)
	3D Information Signs (see "Using the 3D information signs" on page 681)
	Vehicles in the network are the result of simulation and cannot be inserted as network objects (see "Displaying vehicles in the network in a list" on page 847).
	Pedestrians in the network are the result of simulation and cannot be inserted as network objects (see "Showing pedestrians in the network in a list" on page 853).
	Areas (see "Modeling construction elements" on page 880)
	Obstacles (see "Modeling construction elements" on page 880)
	Ramps & Stairs (see "Modeling construction elements" on page 880)
	Elevators (see "Modeling elevators" on page 989)
	Pedestrian Inputs (see "Modeling pedestrian inputs" on page 936)
	Pedestrian Routes (see "Modeling routing decisions and routes for pedestrians" on page 939)
	Pedestrian Attribute Decisions (see "Using pedestrian attribute decisions" on page 965)
	Pedestrian Travel Times (see "Defining pedestrian travel time measurement" on page 998)

To insert network objects in a network editor, you have to select the network object type on the Network object toolbar (see "Using the Network object toolbar" on page 61).

Each network object has attributes and attribute values. Many attribute values are predefined by default. When you insert a network object into a Vissimnetwork, you can edit the attribute values. Attribute values can also be edited later on(see "Editing attributes of network objects" on page 350).

6.1 Setting up a road network or PT link network

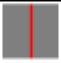
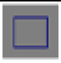
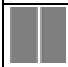
The basic element of a road network in Vissim is the link. Links can run in one direction over one or more lanes. You connect links via connectors; in this way, you construct the link network. The traffic can only flow via connectors from one link to another. It is not sufficient to model links without connectors attached or to have them overlap.

You can also use links and connectors to create a line network for public transportation (see "Modeling short-range public transportation" on page 511).

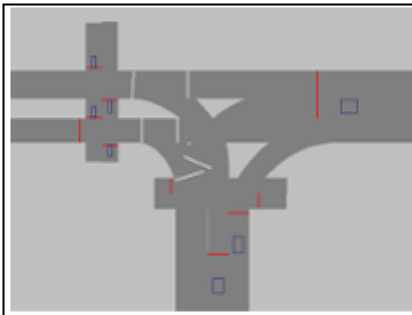
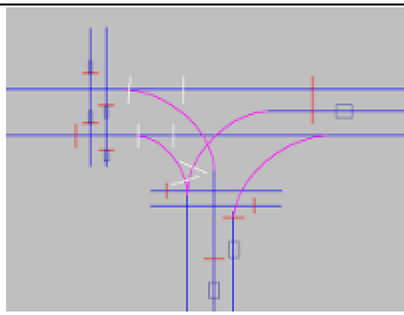
In the road network or line network, you can add the required network objects exactly and define their attributes. Many network objects may lie on a link or connector, e.g. stop signs, routing decisions, PT lines or data collection points. Vehicle inputs may be positioned on links only. You can edit or delete network objects and attributes later on. For example, you may move a network object lying on a link or connector to a different position on the link or connector or copy it to a different link or connector.

6.1.1 Example for a simple network

The example shows a three-legged signalized intersection, with links, connectors, and other network objects:

Icon	Network object
	Signal Heads
	Detectors
	Priority Rules

You can define the colors of network objects via the graphic parameters (see "Editing graphic parameters for network objects" on page 158).

	
Wireframe display disable: Three-legged intersection with two pedestrian crossings	Wireframe display enable: The network is represented via the middle lines of links (straight blue lines) and the middle lines of connectors (pink lines in the curves).

6.1.2 Traffic network data

A Vissim traffic network is made up of the following data:

- Static data, which is not changed by the simulation.
- Dynamic data, which essentially includes all information that describes the simulated traffic.

6.1.2.1 Static data

Static data illustrates the applicable traffic infrastructure. This data is necessary for simulation and manual tests from traffic-dependent signal controls. Static data, for example, includes:

- Links with starting points and end points and perhaps intermediate points. Links are directional roadways on which traffic flows. For each link you specify a number of lanes.
- Connectors between links for the modeling of possible turns and from merging and broadening of lanes
- Position and length from public transport stop
- Position from signal head and stop line and references to the allocated signal groups
- Position and length of detectors used to record vehicles for vehicle-actuated signal control.
- Position of PT calling points that record PT vehicles sending PT telegrams

6.1.2.2 Dynamic data

For the simulation, you define dynamic data, for example:

- Vehicle inputs indicate congestion, including vehicle compositions, as relative shares, for example HGV shares for all links which lead into the network.
- You define link sequences via routing decisions. You define routes via temporally variable traffic volumes that refer to vehicle classes.
- The position and the values from Headway and Time Gaps from Priority Rules. Priority Rules describe priority rules, for example, "right before left" or determines acceptable left turn.
- The curvature, departure times and boarding and alighting from public transport lines

6.1.3 Evaluating vehicular parameters from the network

You can determine from the network different vehicular parameters and subsequently evaluate them (see "Performing evaluations" on page 1001). Thereby, for example, you can apply the following network objects:

- Data collection points for user-defined local data collection measurements, for example, measurement of the number of vehicles of particular vehicle classes, their acceleration and average speed (see "Defining data collection points" on page 446)
- Vehicle travel time measurements for the measurement of total travel times and delays when driving on the respective network segment (see "Defining vehicle travel time measurement" on page 447)
- Queue counter for the measurement of average and maximum queue lengths (see "Modeling queue counters" on page 450)

Note: The add-on module Viswalk allows for a comparison of pedestrian traffic evaluations.

6.2 Copying and pasting network objects into the Network Editor

In the Network editor, you may select stand-alone network objects and dependent static network objects in 2D mode and copy them to the Clipboard (see "Selecting and copying network objects" on page 340). Network objects copied to the Clipboard can be pasted into a network (see "Pasting network objects from the Clipboard" on page 341).

With the copy-and-paste command you can use network objects again, whose course, form or other attributes you have adjusted. You may also select multiple network objects to copy and paste the modeled parts of your Vissim network. This allows you to build your Vissim network more quickly.

If you have defined several levels, you can copy network objects from one level into another level (see "Copying network objects to different level" on page 343).

Stand-alone network objects

Stand-alone network objects are positioned directly in the Network Editor:

- Links, areas, ramps and stairs
- Obstacles
- Sections
- Elevators
- Nodes
- 3D Traffic Signals
- Static 3D Models

You can copy stand-alone network objects to a different position in the currently opened Vissim network (see "Pasting network objects from the Clipboard" on page 341). You may also open another Vissim network and insert network objects into it.

Dependent network objects

Dependent network objects are located on top of stand-alone network objects:

- The connectors selected are copied, if you have also selected the origin and destination link.
- Desired Speed Decisions
- Reduced Speed Areas
- You may copy conflict areas, if all links have been selected that traverse conflict areas.
- Priority rules may be copied from an individual link, if you have selected a From Section and a To Section. If the From Section and To Section lie on different links, you can copy priority rules together with the links they lie on.
- Stop Signs

- Signal Heads
- Detectors
- Vehicle inputs
- Vehicle routes may be copied from a link, if the From Section and To Section lie on the same link. If the From Section and To Section lie on different links, all links of the vehicle routes must be selected. The From Section of a vehicle route may also be copied to another link without the To Section.
- Parking Lots
- Public transport stops
- Public transport lines may be copied from a link, if the From Section and the To Section lie on the same link. If the From Section and To Section lie on different links, all links of the public transport lines must be selected.
- Data Collection Points
- Vehicle travel times may be copied from a link, if the From Section and the To Section lie on the same link. If the From Section and To Section lie on different links, you must also select and copy the respective links.
- Queue Counters
- Backgrounds
- Pavement Markings
- Pedestrian Inputs
- Pedestrian Routes: The From Section of a pedestrian route may also be copied to the same or another area without the To Section.
- Pedestrian Travel Times

You may paste dependent network objects that you have copied from a stand-alone network object into another stand-alone network object of the same type (see "Pasting network objects from the Clipboard" on page 341). You can also open another Vissim network and insert copied, dependent network objects into a stand-alone network object. The stand-alone network object must be of the same network object type as the network object copied.

Copying stand-alone network objects together with dependent network objects

To copy multiple stand-alone network objects together with their dependent network objects, around the network objects, drag open a frame. This allows you to reuse the modeled parts of your Vissim network.

Copying during a simulation

You may copy static network objects during a simulation in the 2D mode. Dynamic network objects cannot be copied during a simulation.

Copying network objects with a reference to a SC

When you copy network objects with a reference to a signal control, Vissim also copies the settings, e.g. when copying detectors, signal heads, 3D signal heads, priority rules or stop signs. When you paste these network objects, a window opens that allows you to select whether Vissim shall use the same SC or a new SC based on the original one.

Copying network objects with a reference to files

When copying network objects that contain file references, the references only are copied. The files are not copied.

Copied base data

When copying the network objects, Vissim also copies base data on which the network objects are based, for example distributions, functions, vehicle types, pedestrian types, vehicle classes, pedestrian classes, and/or behavior parameters.

6.2.1 Selecting and copying network objects

1. Select the desired network objects in the Network Editor (see "Moving network objects in the Network Editor" on page 356).
2. If you have selected dependent network objects with a From Section and a To Section that both lie on stand-alone network objects, make sure that you also select the stand-alone network objects.
3. If you have selected vehicle routes or PT lines, make sure that you also select all the links used by the vehicle routes or PT lines.
4. If you have selected connectors, make sure that you also select the origin and destination links.
5. If you have selected conflict areas, make sure that you also select all links that traverse the conflict areas.
6. Right click the Network Editor.
7. From the shortcut menu, choose **Copy**.

Tips:

- Alternatively, use the following commands to copy network objects to the Clipboard:
  - Key combination Ctrl+C
  - Key combination Ctrl+Ins
  - Network Editor toolbar > **Copy selection**
- You can also duplicate network objects in the Network editor (see "Duplicating network objects" on page 352)

The selected network objects are copied to the Clipboard. Together with the network objects, base data that refers to the network objects is copied.

8. If you have copied network objects with a file reference via a relative path, before inserting them into another Vissim network, make sure that the relative paths specified are also valid for the other Vissim network.
9. If desired, continue to edit the Vissim network.
10. Insert the copied network objects (see "Pasting network objects from the Clipboard" on page 341).

6.2.2 Pasting network objects from the Clipboard

If you have copied stand-alone network objects to the Clipboard, you may paste them into the Network Editor at the position of your choice. You may then edit these network objects, for example move or rotate them, or adjust their course, form or attributes.

You can paste network objects into the same Vissim network from which you have copied them or into another Vissim network, for example another instance of Vissim you have opened. If in the Vissim network, base data is missing to which the pasted network objects in the original network refer, the respective base data is also pasted. Vissim then checks the network for conflicts and discards duplicates (see "Reading a network additionally" on page 361). If identical base data already exists, these are used for the pasted network objects.

If together with the stand-alone network objects, you have selected and copied dependent network objects that lie within the stand-alone network objects, these are pasted together with the stand-alone network objects.

If you have copied dependent network objects from a stand-alone network object, you may paste the dependent network objects into another stand-alone network object of the same type.

You may paste **connectors** from the Clipboard, if they have been copied with their origin and destination link.

You may paste **conflict areas** from the Clipboard, if all links were copied that traverse the conflict areas. If after pasting a link of a conflict area, the pasted link intersects with an existing link, a new conflict area is created.

When from the Clipboard you paste network objects that refer to files, the file references remain intact. The files are not copied.

6.2.2.1 Pasting stand-alone or dependent network objects

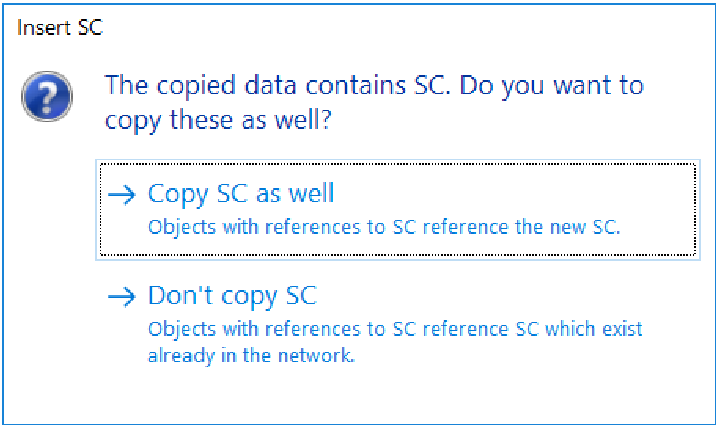
1. Depending on whether you have copied stand-alone or dependent network objects to the Clipboard, carry out the following steps:
   - To paste stand-alone network objects, in the Network Editor, right-click the position where you want to paste the network objects.
   - To paste dependent network objects, in the Network Editor, click the stand-alone network object to which you want to add the dependent network objects.

2. From the shortcut menu, choose **Paste**.

Tip: Alternatively, use the following commands to paste network objects from the Clipboard:

- Stand-alone network objects: In the Network Editor, move the mouse pointer to the position of your choice and press CTRL+V or SHIFT+INS.
- Dependent network objects: In the Network Editor, move the mouse pointer to the stand-alone network object of your choice and press CTRL+V or SHIFT+INS.
- Instead of using the key combinations, press the **Paste from clipboard** button on the Network Editor toolbar.

When you can paste network objects from the Clipboard that contain a reference to a SC, a window opens.



3. Click the button of your choice:

Element	Description
Yes	Adopt the references of inserted network objects into existing SC
No	Define new SC based on the SC that contains references to the inserted network objects The references of the inserted network objects are adjusted to the new SC.

In the Network Editor, stand-alone network objects are inserted at the position of the mouse pointer. If the mouse pointer is positioned outside the Network Editor, the network objects are inserted at a position in the network that is shown as the middle in the active Network Editor.

Dependent network objects are inserted into the stand-alone network object selected. Positioning of the network object depends on several factors, for example the network object type.

Inserted network objects have been selected. You can then move or rotate the network objects.

In the network objects list of the network object type, a new row is added for each network object inserted.

*A new connector is assigned the attribute **Number**, which is a number available > 9,999.*

*For all other network object types, a new network object is given the next higher number available as the **Number** attribute.*

If together with the network objects new data is inserted, the data is also assigned a new number.

4. If desired, you can edit inserted network objects, for example move them to a different position or adjust their attributes.

6.2.3 Copying network objects to different level

In the Network editor, you can select static network objects and copy them to a different level. If dependent network objects have been placed on the network objects you wish to copy, and you select those as well, you can copy the network objects together.

As long as a copyable number of objects was selected, the objects are copied and their **level** attribute is changed, adding the level number offset to the previous level number. The level number offset is the difference between the values of the **number** attribute of the level you copy the objects from to the level you copy the objects to. The level number offset can be negative when you copy from one level with a higher number to another level with a lower number. All levels with numbers resulting from this step must already exist, otherwise the process is canceled.

1. Select the desired network objects in the network editor (see "Moving network objects in the Network Editor" on page 356).
2. If you have selected dependent network objects with a From Section and a To Section that both lie on stand-alone network objects, make sure that you also select the stand-alone network objects.
3. If you have selected vehicle routes or PT lines, make sure that you also select all the links used by the vehicle routes or PT lines.
4. If you have selected connectors, make sure that you also select the origin and destination links.
5. If you have selected conflict areas, make sure that you also select all links that traverse the conflict areas.
6. Right click the Network Editor.
7. From the shortcut menu, choose **Copy to level(s)**.

   *The **Copy to level(s)** window opens. The list box shows the levels defined. It also contains the offset number for the respective level against the level from which the network objects are copied.*
8. In the list box, click the level to which you want to add the network objects.
9. Confirm with **OK**.

The network objects selected are copied to the level selected. Together with the network objects, base data that refers to the network objects is copied.

6.2.4 Saving a selected part of the network

In a Vissim network, you may select stand-alone network objects and save them as a subnetwork to a **.inpx* network file. If on top of a stand-alone network object there are dependent network objects you wish to save, select the dependent network objects as well.

You cannot save dependent network objects without saving the stand-alone network objects they are placed on.

1. Select the desired network objects in the Network Editor (see "Moving network objects in the Network Editor" on page 356).
2. From the **File** menu, choose **Save Subnetwork as**.

   *The **Save File As** window opens.*
3. Make the desired changes:

Field	Description
File type	File format of network file: The default setting is **.inpx*.
Filename	Name of file to which the subnetwork is saved

4. Select the path to the desired directory.
5. Click the **Save** button.

*The network objects selected are saved to the *.inpx network file.*

6.3 Editing network objects, attributes and attribute values

You can edit network objects, their attributes and attribute values via the following elements of the user interface:

Element	Element with editing functions
Menu commands	➤ **Lists** menu: In a list, show the attributes and attribute values of objects of a base data type or network object type ➤ **Base data** menu: In a list, show the attributes and attribute values of objects of a base data type

<table>
<tr><th>Element</th><th>Element with editing functions</th></tr>
<tr><td>Network editor</td><td>
Graphical display and editing of network objects.

When no simulation is running, in the Network editor, you can select, move, copy, delete and paste network objects.
Double-click the network object to open the <Name network object type> window. Attributes and attribute values of the network object are displayed.
Right-click the network object to open the shortcut menu. From the shortcut menu, choose a command of your choice, e.g. show the network object and its attributes in the list of network objects of the network object type.
While the simulation is running, network objects, attributes and attribute values cannot be edited or can only be edited to a very limited extent. When, during a simulation run, you open the <Name network object type> window to display attributes of network objects, a message is displayed informing you of the limited editing options.

Toolbar of the Network editor (see "Network editor toolbar" on page 75): You can copy selected network objects to the Clipboard.
Shortcut menu in the Network Editor with and without selected network objects (see "Network editor context menu" on page 80)
Open <Name Network object type> window (see "Showing attribute values of a network object in the Network editor" on page 351)
</td></tr>
<tr><td>Lists</td><td>
Show attributes and attribute values of base data and network objects in lists (see "Selecting and editing data in lists" on page 100), (see "Editing lists and data via the context menu" on page 103).
Toolbar of lists (see "List toolbar" on page 97)
Shortcut menu in column header, row header, cells (see "Editing lists and data via the context menu" on page 103)
</td></tr>
</table>

Element	Element with editing functions
Network objects toolbar	(see "Using the Network object toolbar" on page 61) ▶ Button **Toggle visibility in current network editor**: Only if the network object toolbar, the network object type is not selected: Shows or hides network objects of the network object type in the active Network editor. ▶  Lock button **Toggle selectability in current network editor** shows an open lock: You can select and edit network objects of this network object type in the Network editor. ▶  Lock button **Toggle selectability in current network editor** shows a closed lock: You cannot select and edit network objects of this network object type in the Network editor. ▶ **Edit graphic parameters** button ▶  button **Toggle label visibility in current network editor**: The labeling of the network objects of the network object type is not displayed. ▶  button **Toggle label visibility in current network editor**: The labeling of the network objects of the network object type is displayed. ▶ Shortcut menu commands (see "Context menu in the network object toolbar" on page 64)

For many object types, you assign the definition of objects to other objects (see "Using coupled lists" on page 119).

6.3.1 Inserting a new network object in a Network Editor

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

On the network object toolbar, the desired network object type must be selected.

Network objects can be superimposed in the Network Editor. While you add a new network object, you can select the superimposed network objects until the network object, to which you want to add the new network object, is marked (see "Selecting a network object from superimposed network objects" on page 360).

6.3.1.1 Commands for inserting new network objects

If on the Network objects toolbar, you selected a network object type, you can insert network objects into the Network editor via the following functions:

- Via the context menu of the network editor If for the network object type, you inserted a start section for the network object and afterwards need to insert one or multiple destination sections, the respective function is displayed in the context menu.
- Depending on the user settings, using the right mouse button or normally using the right mouse button and the CTRL key (see "Right-click behavior and action after creating an object" on page 152)

6.3.1.2 Inserting new network objects in available positions or on other network objects

Independent from network object type, you can enter network objects in available positions or you must place network objects on other network objects. This is described for each network object according to the definition of the network objects (see "Creating and editing a network" on page 334).

- You can enter, for example, links or areas onto available positions in the Network Editor. Thereby, you can cut or overlap network objects.
- You must place network objects of other network object types on network objects. For example, you can insert desired speed decisions, reduced speed areas, priority rules, detectors, parking lots, vehicle routes, vehicle inputs, vehicle travel time measurements, etc. onto links. Insert pedestrian inputs, sections for area measurement or pedestrian travel time measurements onto areas.
- For vehicle travel time measurements and pedestrian travel time measurements, insert a From Section and a To Section.
- You add a From Section for priority rules, vehicle routes and pedestrian routes; you can also add one or more To Sections.
- For network objects that have a start section and multiple destination sections, you can select the start section again later on to insert additional destination sections or move them.

6.3.1.3 Work steps during addition depend on the network object type

After you have begun to add a network object, the further steps until the network object is fully added depend on the network object type. These steps are described in the definition of network objects (see "Creating and editing a network" on page 334). There are the following differences:

Network object types whose length is defined by dragging the mouse

- Links: While dragging the mouse, you can use the left mouse button to set intermediate points and change the direction there later.
- Reduced Speed Areas

- Parking Lots
- Detectors
- Public transport stops

For these network object types, the following applies:

- When the desired length is achieved by dragging the mouse, release the CTRL key and the right mouse button.

- For connectors, the following applies:
- When the desired position is reached on the destination link by dragging the mouse, and the edges of the destination link are marked by arrows along the direction of travel, release the CTRL key and the right mouse button.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

Network object types with a marker

- Desired Speed Decisions
- Stop Signs
- Signal Heads
- Vehicle Inputs
- Data Collection Points
- Queue Counters
- Pavement Markings
- 3D Traffic Signals

For these network object types, the following applies:

- Once the network object is inserted, release the CTRL key and the right mouse button.

Network object types with a start section and a destination section

- Public Transport Lines
- Vehicle travel time measurements
- Pedestrian Travel Time Measurement

1. When the desired position of the destination section is achieved by dragging the mouse, click the right mouse button.
2. Release the keys.

Network object types with a start section and several destination sections

- Priority Rules
- Vehicle Routes
- Pedestrian Routes

For these network object types, the following applies:

1. Repeat the next two steps until all destination sections have been inserted for the network object:
2. Move the mouse pointer to the desired destination section position.
3. Click.
4. After having inserted all destination sections for a network object, click into an empty section of the Network Editor.

Planar network object types

You can add planar network objects as a rectangle or polygon.

- Areas
- Obstacles
- Ramps & Stairs
- Sections

For these network object types, the following applies:

- for a rectangle: If you have drawn a rectangle to the desired size, double click.
- for a polygon: Once you have added the desired number of the polygon points, double click.

Graphic file of a background image

The background is inserted and selected. You can change its size via the corner drag points.

- Click in an empty area of the Network Editor.

Nodes

- Once you have added the desired number of the polygon points, double click.

6.3.1.4 Behavior after addition

Depending on the user settings and the network object type, a window or a list can open - immediately after adding a network object - or none of the two (see "Right-click behavior and action after creating an object" on page 152). In the window or the list, you can enter input attributes for the network object.

When you insert 3D model or 3D signal head, a symbol is inserted and a window opens. There, you can select the desired file.

When you insert a background, a window opens. There you can select the desired graphic file.

After insertion, a network object is automatically selected. This allows you, for example, to copy it or position it exactly.

6.3.1.5 Adding a network object to a level

If you have defined levels and you add a new stand-alone object such as a link, area, stair or ramp, an obstacle, a background graphic, a static 3D model or a 3D signal head, the network object is added in the lowest-number level visible in the current network editor. If all levels are invisible or no network editor is open, the network object is generated in the lowest-number level. You can assign the network object a different level (see "Attributes of links" on page 409).

6.3.1.6 Adding a network object at the beginning or end of a connector

You can place a network object on the same link coordinates as the beginning of an outbound connector or exactly at the end of a connector. Vehicles that change links at this position will recognize the network object. This means that a conflict area, e.g., that ends precisely at the end of a connector, will work as expected.

6.3.1.7 Canceling addition of network objects

You want to cancel the insertion of a network object depending on the network object type:

- To cancel the insertion of a start section, in the Network Editor, move the mouse pointer to an empty section and release the right mouse button.
- To cancel the insertion of a destination section, in the Network Editor, move the mouse pointer to an empty section and double-click.
- To cancel the insertion when a window with attributes is open, release the mouse buttons and press the ESC key.
- To cancel the insert of a construction element, release the mouse buttons and press the ESC key.

6.3.2 Editing attributes of network objects

You can edit attributes of network objects in the list of network objects of a network object type. For some network object types, you can select attributes in the **<Name network object type>** window.

The maximum value for all integer input values is 4 294 967 295.

6.3.2.1 Editing attributes in a list

1. From the **Lists** menu, choose the desired entry.

Tip: Alternatively, you can show the list via the following functions:
- On the Network objects toolbar, right-click the desired network object type and from the shortcut menu, choose **Show List**.
- In the Network editor, right-click a network work and from the shortcut menu, choose **Show In List**.

The list with the attributes of the network objects of the network object type opens.

By default, the column titles show the short names of the attributes. The short names are abbreviations of long names (see "Showing short or long names of attributes in column headers" on page 156).

By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Edit the attributes (see "Selecting and editing data in lists" on page 100):

6.3.2.2 Attributes in the *Editing <Name network object type>* window

For some network objects, a **<Name Network object type>** window is automatically opened when you define a network object and have selected that you want the program to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). You can enter entry attributes in this window. By default, the entry attributes are also displayed in lists.

For network objects which are already defined, you can call up this window via the following functions and edit within the attributes:

- In the network objects list of the network object type (see "Functions available in the shortcut menu of the row header" on page 104)
- In the Network Editor (see "Showing attribute values of a network object in the Network editor" on page 351)

While the simulation is running, network objects, attributes and attribute values cannot be edited or can only be edited to a very limited extent. When, during a simulation run, you open the **<Name network object type>** window to display attributes of network objects, a message is displayed informing you of the limited editing options.

6.3.3 Showing attribute values of a network object in the Network editor

In the Network editor, you can highlight network objects and open windows for some network object types. The windows display attribute values for the network objects. For all network object types, you can show attributes in lists. Important entry attributes are described for the network objects (see "Creating and editing a network" on page 334).

1. In the Network editor, double-click the network object of your choice.

*The **<Name network object type>** window or **<Name network object type>** list opens.*

2. You can change the settings if you wish.

3. Confirm with **OK**.

6.3.4 Direct and indirect attributes

A network object has attributes that allow you to save input data and output data:

- **Direct attribute**: The data refer directly to the network object. Example: **Length** attribute of a link.
- **Indirect attribute**: The data result from the relation between two network objects. Example: For a network object **Link**, you can select the network object type **Display type** as a relation. A direct attribute of this display type is **Fill style**. You can select the fill style chosen for a link as an indirect attribute for links and show it in the **Links** list for each link defined. You can edit an indirect attribute if it is an n:1 or 1:1 relation of a network object.

Direct and indirect attributes can also be used to define user-defined attributes (see "Using user-defined attributes" on page 210).

6.3.5 Duplicating network objects

You can select and duplicate individual or multiple network objects in lists and network editors. In the network editor, you can duplicate the following network objects:

- One or more selected independent network objects, for example, from network object type links, connectors, areas, ramps, nodes, sections, static 3D objects, 3D signal heads, background. You can also duplicate independent network objects of different network object types. You can move duplicate independent network objects in the network editor. Dependent network objects which lie on selected, independent network objects are not duplicated in the process, unless they are selected.
- One or more selected dependent network objects which lie on an independent network object. You can also duplicate dependent network objects of different network object types. You can move duplicated dependent network objects in the network editor to the independent network object or another independent network object of the same network object type.

- Select the desired function.

<table>
<tr><th>Element</th><th>Functions to duplicate</th></tr>
<tr><td>Lists</td><td>Shortcut menu > Duplicate: You can also select more cells and therefore duplicate more network objects. The new network object is added in the network editor in the same position as the duplicated network object and is selected.</td></tr>
<tr><td>Network Editors</td><td>Shortcut menu > Duplicate: The new network object is added in the network editor in the same position as the duplicated network object and is selected.
Hold down the CTRL key, click in the network object and drag the new network object to the desired position:
for independent network objects, for example, links or areas, to the desired position in the network editor
for dependent network objects which, for example, lie on links or areas to the desired link or area</td></tr>
</table>

In the list of the network objects of the network object type, a new row is added.

*A new connector is assigned the attribute **Number**, which is a number available > 9,999.*

*For all other network object types, a new network object is given the next higher number available as the **Number** attribute.*

6.3.6 Moving network objects in the Network Editor

In the Network editor, you can move stand-alone network objects, e.g. links or areas. In doing so, the following network objects are also moved:

- Start and end points of connectors that are not selected, when their starting link and destination link are selected and moved. If you also wish to move the connector, you must also select this.
- Dependent network objects that lie on top of stand-alone network objects that are moved, e.g. stop signs, parking lots, start sections and destination sections of routes and routing decisions on links, or obstacles, ramps and stairways in areas.

In the network editor, you can also move one or several dependent network objects that are on stand-alone network objects, for example stop signs or parking lots to links. These network objects can only be moved on a stand-alone network object or onto another stand-alone network object. If you drag the selected network objects out of the stand-alone network object into a free area in the network editor and release the mouse button, it is deleted.

Tip: Alternatively, in lists showing the attributes of a network object, you can change the attribute value **Position**, if the network object has this attribute.

6.3.6.1 Moving stand-alone network objects

1. Select the desired network objects in the network editor (see "Moving network objects in the Network Editor" on page 356).
2. Hold down the mouse button and move the cursor to the desired position.

3. Release the mouse button.

Note: By moving the beginning or end of a connector or an entire connector from one link to another link, PrT and PT routes are interrupted.

6.3.6.2 Moving dependent network objects

1. Select in the network editor the desired network objects, which are on stand-alone network objects (see "Moving network objects in the Network Editor" on page 356).
2. Hold down the mouse button and move the cursor to the desired position.

6.3.7 Moving network object sections

In the network editor, you can move an individual start or destination section to the same link or to another link or connector.

1. Click the header of the Network Editor.
2. In the Network editor, click the desired section and keep the mouse button pressed.

   The mouse pointer becomes a  *symbol. The section and link or connector are highlighted.*
3. Move the start section or destination section to the desired position on the same link or connector or to a different link or connector.

   When you move the mouse pointer to a different link or connector, the latter is highlighted and you can move the section there.

   If you want to move the section to a position on a link or connector that lies under the link or connector currently highlighted, use the TAB *key to highlight the underlying link or connector (see "Selecting a network object from superimposed network objects" on page 360).*

   Click into an empty section of the Network Editor to cancel the insertion.
4. Release the mouse button.

6.3.8 Calling up network object specific functions in the network editor

You can call up functions for the different network object types in the network editor via the context menu next to the standard functions, which are only possible for the currently selected network object types, for example, **Links** > **Split Links**. These functions are described with the network objects.

6.3.9 Rotating network objects

You can turn an individual, independent network object or select and turn several network objects.

6.3.9.1 Turn individual network object

You can turn an individual network object with network object types areas, obstacles, ramps, stairways, intersections, backgrounds, static 3D models, 3D traffic signals and sections.

1. Click on the network object type of the network object in the network object toolbar.
2. Click the desired network object.

   At the corner points of the network object, curved arrows with two arrow heads are displayed.
3. Click the desired curved arrow and hold down the mouse button.
4. To align the network object, reduce the rotation speed by moving the mouse pointer straight away from the network object.
5. Drag the mouse pointer in circles to the desired direction.
6. Release the mouse button.

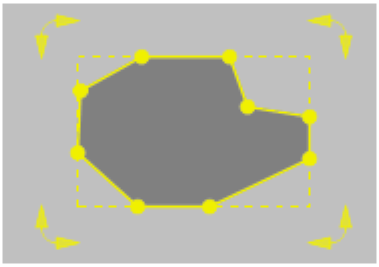
   The selected network object is turned around the turning point.
7. Release the keys.

6.3.9.2 Select several network objects

You can turn several independent network objects with the network object types links, connectors, areas, obstacles, ramps, stairways, intersections, static 3D models, 3D traffic signals and sections. These may also be network objects with different network object types.

1. Hold down the CTRL key and click on the desired stage sequence.

   A dashed line frame is drawn aground the maximum dimension of the network objects. At the corner points of the dashed line frame, curved arrows with two arrow heads are displayed:

> Tip: Alternatively you can also hold down the left mouse button and draw a frame over the desired network objects.

2. Point the mouse pointer to the destination area of your choice.
3. Hold down the ALT key, click in the Network Editor on the desired construction element and circularly move it with the mouse pointer in the desired direction.

The selected network objects are turned around their center point.

4. Release the keys.

6.3.9.3 Key combinations for turning network objects

You can use the following key combinations for turning:

Hotkeys	Description
CTRL+ALT	Duplicate and turn selected network objects.
CTRL+SHIFT	Duplicate selected network objects and position at intervals of 22.5°.
CTRL+ALT+SHIFT	Duplicate selected network objects and turn at intervals of 22.5°.

6.3.10 Deleting network objects

You can select and delete network objects. If you delete network objects that have an assignment to other network objects or onto which other network objects have been positioned, these may also be deleted. If you would like to keep the allocated network objects, assign them to another network object before deleting.

Element	Functions to delete
Lists	- Context menu > **Delete** - Key DEL The network objects selected in the list are deleted. The network objects selected in the network editor are not deleted.
Network Editors	- Context menu > **Delete** - Key DEL - In Network Objects, drag the inserted network objects from there and release the mouse button The network objects selected in the network editor are deleted. The network objects selected in a list are not deleted.

6.4 Displaying and selecting network objects

You can show and edit network objects and their attributes in lists. In Network Editors you can add network objects in the 2D mode and position them exactly in the network. You can re-select and re-edit network objects. In 3D mode you can view the network from different perspectives.

You can run simulations in 2D and 3D mode. During the simulation you can select vehicles or pedestrians, automatically display them in the Quick View and mark them in lists, for example, in order to evaluate dynamic data from the simulation directly in the simulation.

6.4.1 Moving network objects in the Network Editor

In a Network Editor, you can select network objects in the 2D or 3D mode. You can select network objects without having to select the network object type in the Network objects bar.

You can then edit the network objects, e.g. move them in the Network Editor, change attributes in the Quick View, or access functions via the context menu.

If in the Network Editor you point the mouse pointer to a network object, it is highlighted. This makes it easier to select the network. You can then select the network object.

6.4.1.1 Selecting or deselecting network objects

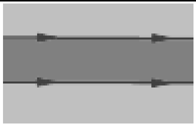
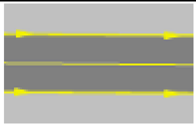
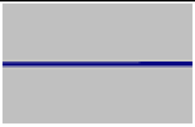
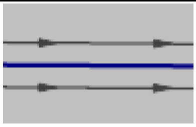
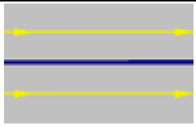
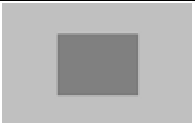
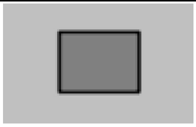
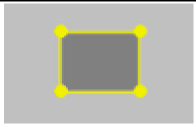
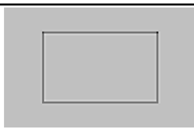
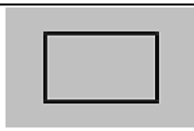
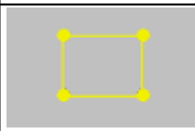
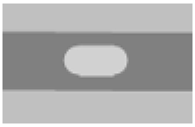
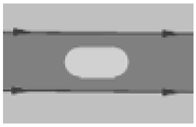
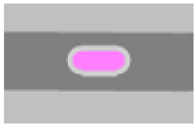
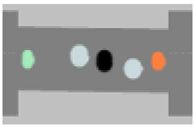
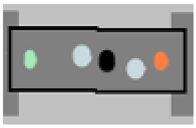
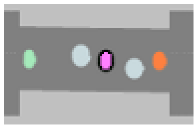
- Make the desired changes:

Purpose	Description
Selecting a network object	• Click on a network object which is not selected. When you right-click a network object, the shortcut menu of the Network Editor is opened (see "Network editor context menu" on page 80). • If network objects have multiple sections, you can click the start section or destination section to e.g. move it or open the context menu. • Vehicle routes and pedestrian routes: Click the destination section • Public transport lines, vehicle travel time measurements, pedestrian travel time measurements: Click the start section or the destination section
Selecting multiple network objects in the 2D mode	Alternatives: • Hold the left mouse button down and draw a frame • Hold down the CTRL key and click the network objects
Selecting multiple network objects in the 3D mode	Hold down the CTRL key and click the network objects
Undoing the selection of all network objects	Alternatives: • Click on an area which is not selected. • Right-click in an area that is not selected.
Undoing the selection of individual network objects	Hold down the CTRL key and click the selected network objects you want to deselect.

The network objects selected are highlighted in the Network editor. Attribute values are displayed in Quick View (see "Using the Quick View" on page 68)

Note: You can select a synchronization with network editors for lists and other lists. Then network objects that you select are automatically highlighted in the list and network editor (see "List toolbar" on page 97).

6.4.1.2 Examples of visualization of network objects

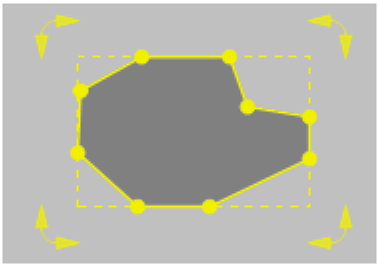
Network object	**Not selected**	**Mouse pointer points to network object**	**Selected**
Link			
Link in wireframe view			
Area			
Area in wireframe view			
Vehicle during simulation		Highlights the link: 	Highlights the vehicle: 
Pedestrian during simulation		Highlights the network object: 	Highlights the pedestrian: 

6.4.1.3 Visualization of selected network objects that lie on a link in an area

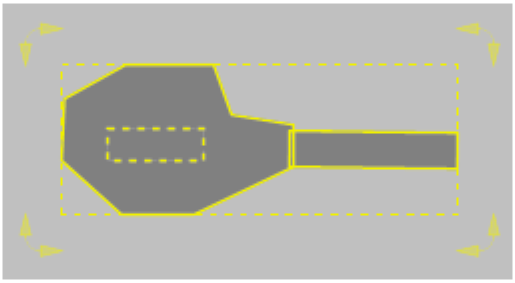
When you select a network object that lies on a link or in an area, the network object as well as the link or area are highlighted. This makes it easier for you to identify the network objects that belong together, particularly if several network objects are lying on top of each other. The highlighted link or area is displayed as shown in the diagram in the table above, in the column **Mouse pointer points to network object**: The edge of a link is marked by arrows pointing in the direction of travel. The edge of an area is marked by a bold black line.

6.4.1.4 Marker frame shows extension

A marker frame with a dashed line indicates the maximum horizontal and vertical extension of a selected network object of some types, e.g. areas. Curved arrows with two arrow heads mark points around which the network object can be rotated:



This also applies when multiple network objects are selected.



6.4.2 Selecting network objects in the Network editor and showing them in a list

In the network editor you can select network objects of a particular network object type and show them, together with their attributes, in a list of network objects with the particular network type.

1. In the Network Editor, right-click the network object of your choice.
2. From the context menu, choose entry **Show In List**.

   The list of defined network objects for the network object type opens.

The objects selected in the Network editor are marked in the list, if the list is synchronized (see "List toolbar" on page 97).

6.4.3 Showing the names of the network objects at the click position

You can show a list of network objects located at the click position. If several network objects overlap each other, this makes it easier for you to select the desired network object.

1. In the Network Editor, right-click the network object of your choice.
2. Choose **Objects At Click Position** from the context menu.

The network objects are shown in the context menu.

3. Click on the desired entry.

The display in the Network Editor is adjusted. The network object is selected.

6.4.4 Zooming to network objects in the network editor

In the network editor you can select the size of the network so that the selected network objects are automatically completely displayed.

- In the context menu, select **Zoom To Selection**.

6.4.5 Selecting a network object from superimposed network objects

If several network objects are superimposed in a Network Editor, you can select these consecutively. This facilitates, for example, the addition of new network objects on a desired link.

1. On the Network Editor, click on the position at which several network objects are superimposed.

   The icon is active in the network editor toolbar.

2. Click on the icon , until the desired network object is selected.

Tips:

- Alternatively you can use the TAB key.
- You can also press the TAB key to consecutively select the superimposed network objects, while holding down the right mouse button and the CTRL key by default to add a new network object on the desired network object.

6.4.6 Viewing and positioning label of a network object

In the Network Editor, you can view the attributes of the network object in a text box and position the text box.

6.4.6.1 Showing label

1. On the Network Objects toolbar, in the row of the desired network object type,click the **Edit graphic parameters** button.

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

2. Make sure that the option is selected for the **Label visibility** attribute.

3. Click the button next to the **Label attribute**.

   The list of attributes opens.

4. Select the desired entry.

5. Confirm with **OK**.

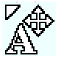
6. Click next to the list with the graphic parameters.

The label is shown in the network object.

6.4.6.2 Positioning label

1. Click on the network object type of the network object in the network object toolbar.

2. In the Network Editor, move the cursor on the label of the desired network object.

   The mouse pointer becomes a  *symbol.*

3. Hold down the left mouse button and drag the label to the position of your choice.

4. Release the mouse button.

6.4.7 Resetting the label position

In the network editor, you can reset the label position of a network object to zero.

1. On the Network objects toolbar, click the network object type.

2. In the Network Editor, right-click the network object of your choice.

3. In the context menu, select **Reset label position**.

6.5 Importing a network

You can import the following data:

- An abstract network model from Synchro 7
- An abstract network model (ANM) from Visum
- Import an abstract network model from Synchro 7 adaptively into an open network. Thereby, you can continue to edit the original network in Synchro 7 and import parts thereof.
- Import desired data additionally from a saved Vissim network file into an open network

Tip: For pedestrian areas or obstacles, you can import AutoCAD data (see "Importing walkable areas and obstacles from AutoCAD" on page 882).

6.5.1 Reading a network additionally

You can read network objects from another Vissim network into the current network. In this way you can select under **Conflict avoidance** whether Vissim the cross-overs of the numbering between the imported and available network objects and base data should be checked, and also in **Conflict handling** choose, how Vissim these conflicts should be solved in the event of cross-overs.

Translating English names in the *Name* attribute

When reading in additional data, you normally do not want objects duplicated that are identical in both network files. This is particularly true for base data objects, e.g. vehicle types.

If two networks mainly differ due to the different user preferences set, they also differ in terms of the names used in the **Name** attribute of their base data objects, e.g. for vehicle types (English **Car**, German **Pkw**). These names are included in several languages in the default network file *defaults.inpx* that is delivered with Vissim. Even if all other attribute values are identical, different names for the same object can cause a conflict and duplicate object data when additional data is read in. However, you can translate the English names in the **Name** attribute into the language of the network file opened into which you want to read in additional data. Only English names in the default network file *defaults.inpx* delivered with Vissim will be translated.

1. Close the network file into which you want to read in an additional network file with English attribute values.
2. In a text editor, open the network file you want to read in.

   By default, the second row contains version information, e.g. `<network version="200" vissimVersion="8.00 - 00* [55350]" >`.
3. Into the row before the closing parenthesis, insert a space and the following entry: `translateStrings="true"`

   In the above example, this would be: `<network version="200" vissimVersion="8.00 - 00* [55350]" translateStrings="true">`.
4. Save the network file.
5. Import the network file as described below.

If you save the network file after reading it in, the entry `translateStrings="true"` *is deleted.*

Consider reference points

Vissim accounts for reference points in both networks (see "Mapping Vissim network to background position" on page 396):

- Neither of the two networks have a point on the background map that is assigned to a corresponding point in the network.

  Vissim inserts network objects in the Network Editor at the same position they were at in the Network Editor of the original network.
- One of the networks has a point on the background map that has been assigned to a corresponding point in the network:

Vissim inserts network objects in the Network Editor at the same position they were at in the Network Editor of the original network. The reference point that exists in one of the networks is used to assign the network to a point on the background map.

- Both networks have a point on the background map that is assigned to a corresponding point in the network. These reference points are different in both networks:

  Vissim inserts the network objects at the position in the Network Editor that corresponds with the coordinates of the point on the background map. This might not be the same position where the network objects were located in the Network Editor of the original network. In the network that is read in additionally, Vissim does not change the existing assignment of the point on the background map to the corresponding point in the network.

Importing a network

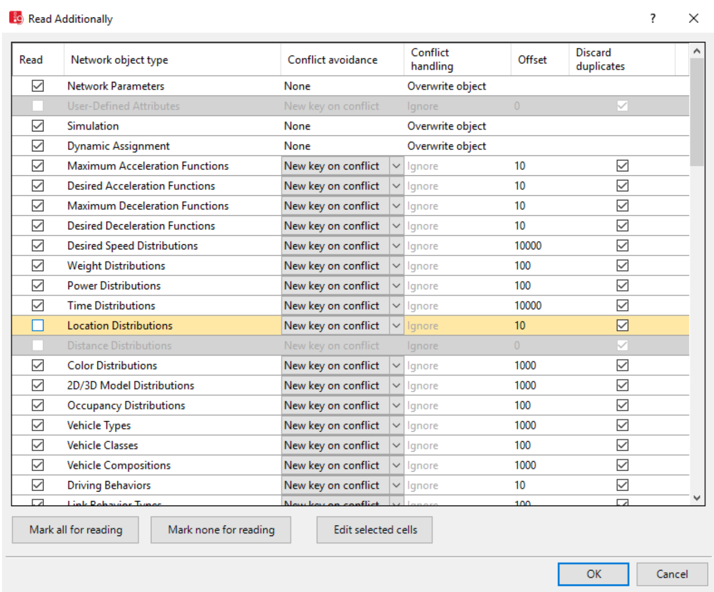
1. Ensure that the network in which you would like to import is open and saved.
2. In the **File** menu, choose > **Read Additionally** > **Network**.

Tip: You can also use **Read Additionally Here** in a Network Editor context menu to read in a file. The center of the read-in network will be positioned at the point where you right-clicked with the mouse.

---

3. Select the network file **.inpx* from which data should be read additionally.

   *The **Read Additionally** window opens.*



4. To sort a column, click the column header.

5. In the next steps, select for all network object types whether you want to import them and how Vissim conflicts and duplicates should be handled.

6. Make the desired changes:

Element	Description
Read	☑ If this option is selected, network objects of this network object type are imported.
Network object type	Name of the network object types which you can import

Element	Description
Conflict avoidance	Defines the behavior when network objects are imported that have the same key as existing network objects of the same type. The key is typically a number. - **None**: Keys remain unchanged. The behavior is defined in the field **Conflict handling**. - **New key on conflict** (default setting): Network objects with the same keys as existing network objects are assigned a new number. In the **Conflict handling** field, a number is entered that in case of a conflict is added to the number of the network objects imported. - **New key for all**: all imported network objects are renumbered. In the **Conflict handling** field, a number is entered that is always added to the number of the network objects imported.

Element	Description
Conflict handling	In the **Conflict avoidance** field, click > **None** to choose from the following options should the key of a network object you want to import be identical to the key of an existing network object of the same type. - **Ignore**: The network object you wanted to import is discarded and the existing network object remains unchanged. Where all the network objects to be imported are discarded and the Vissim network has not been changed, after the Read Additionally, the message **The network has not been changed** appears. - **Overwrite object**: The network object imported replaces the existing network object. - **Cancel**: A message is opened. The import is canceled.
	If in the **Conflict avoidance** field, **New key on conflict** or **New key for all** is selected, Vissim suggests an **Offset** value, which is then added to the number of the network objects you wish to import. You may overwrite this value. Vissim calculates the suggested **Offset** values based on the following: - For numbers of the network objects to be imported, of a network object type

Element	Description
	< 10,000: The **Offset** value is set to the next higher decimal power. - For numbers of the network objects to be imported, of a network object type > 10,000: The **Offset** value is set to the next higher multiple of 10,000. - If all link numbers are less than1,000 and all connector numbers are less than 10,000, Vissim uses a small offset. This way the numbering convention, which says that link numbers are less than 1,000 and connector numbers are less than 10,000, is maintained. Otherwise the offset is set to the next multiple of 10,000.

Element	Description
Discard duplicates	Only for base data and network object types without geometrical positions: ☐ If this option is not selected and you have selected the entry **New key on conflict** in the field **Conflict avoidance** or **New key for all** and you have entered an offset in the field **Conflict handling**, the duplicates are preserved in the network objects. Example: If both networks contain vehicle types with the numbers 1 to 6, the numbers of the imported vehicle types are changed at an offset = 1,000 in 1,001 to 1,006. ☑ If this option is selected, except for the number, the attributes of the network objects of the network object type are compared in each network. If the imported network object is identical to the existing network object except for the number, the imported network object is discarded. For base data, this option is selected by default.

Element	Description
Edit selected	The **Edit Selected** window opens. For the selected rows, you may edit the settings for: - **Read objects**: ☑ If this option is selected, network objects of these network object types are imported. This option is connected with the **Read** option in the **Read Additionally** window. - **Conflict avoidance**: see earlier in this table - **Conflict handling**: see earlier in this table

7. Confirm with **OK**.

*The network objects are read in and selected. Network Editors with **Auto-Zoom Synchronization** selected choose a section that is large enough to show all selected network objects. Backgrounds do not rotate.*

6.5.2 Importing ANM data

You can export from Visum as of Visum 10 data abstracted network models (ANM files) in XML format, or create ANM files with other programs. The abstract network models in ANM files consist of nodes and edges. You have the following options to import ANM data:

- Select ANM file, configure data import and start data import (see "Selecting ANM file, configuring and starting data import" on page 367)
- Adaptive import of ANM data (see "Adaptive import of ANM data" on page 369)
- In Visum, use the subnetwork generator to generate a subnetwork. Open Visum directly from Vissim and import the subnetwork into Vissim. For further information, refer to the Visum manual.

6.5.2.1 Properties of the network generated

When you import ANM files into Vissim, a new Vissim network, with the geometry of links and connectors, is generated. Optionally, nodes can contain additional elements, for example, lanes, lane turns, crosswalks, pockets, control types, signalizations and detectors with the determined vehicle classes (see "Generated network objects from the ANM import" on page 372).

Notes:

- The ANM import only generates segment nodes. For the editing in Vissim, you can convert segment nodes into polygon nodes (see "Converting segment nodes" on page 716).
- You can also import files which have been exported from SITRAFFIC OFFICE.

Volumes and routing are defined in **.anmroutes* files and can be imported in Vissim. This allows you to use this data in the dynamic assignment or as static routes.

Projection in ANM data

If a projection is specified in ANM data, the ANM coordinates are converted into Cartesian coordinates.

If no projection is specified:

- A message is displayed, saying that no projection was recognized.
- ANM coordinates are interpreted as Cartesian coordinates.
- ANM coordinates are not converted.

Information on data for dynamic assignment

Origin-destination matrices and a Path file **.weg* are generated.

Notes:

- The route volumes in a path file, which are generated via an ANM import, must not be in whole numbers because the result of the assignment with Visum can have decimals.
- During export, the route volumes of the dynamic assignment are located in the path file as volume per ANM time interval. During import, they are recalculated in volumes per evaluation interval of the dynamic assignment.
- For the dynamic assignment, these values are randomly rounded in Vissim. This rounding is effected depending on the rounded share. For random rounding, the sum of the matrix values remain somewhat constant. Example: There is a 30% chance that 0.3 is rounded to 1 and a 70% chance it is rounded to 0.

Information on data for static routes

- Vehicle inputs and routing decisions with static routes are generated.
- Each routing decision for static routes contain a name, which contains the number of the ANM origin zone.
- The unique IDs of the ANM routes are taken over as routing numbers from static routing decisions. These routes can be found in the **.anmroutes* file and the respective OD relation is determined.

Information on data for mesoscopic simulation

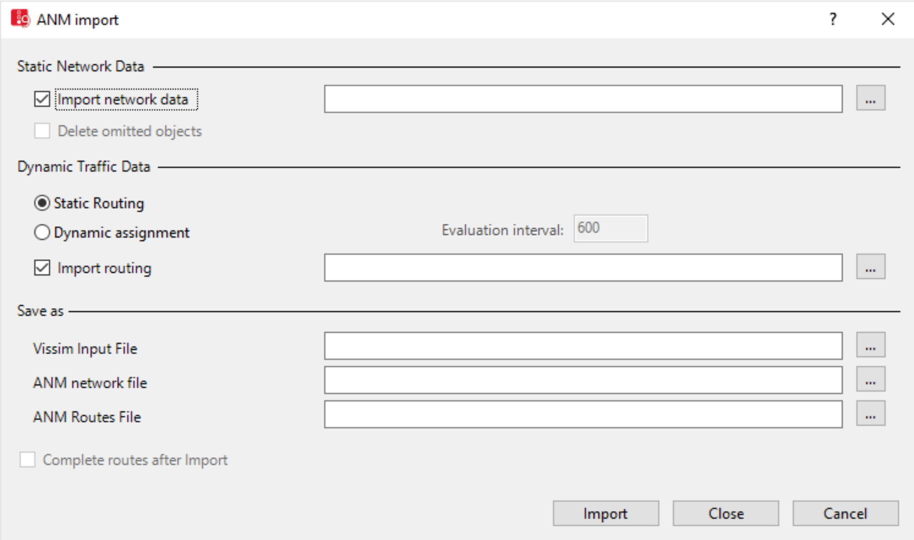
To perform a mesoscopic simulation after ANM data import, in the **ANM import** window, in the **Dynamic Traffic Data** section, select **Dynamic assignment**.

You also need to select this option when importing only one **.anm* file without any routes and/or matrices. This way you ensure that the zones and/or parking lots as well as nodes of the network object type **Node** are generated.

6.5.3 Selecting ANM file, configuring and starting data import

1. From the **File** menu, choose > **Import** > **ANM (Vistro/Visum)**.

   *The **ANM import** window opens.*



2. Make the desired changes:

Element	Description
Import network data	If this option is selected: Enter a path and name of the **.anm* file for the import of the abstract network model. If this option is not selected, you can force a new import of the same routing data, which you initially imported with the abstract network model. Select the option **Import routing**.
Static Routing	Import routing for static routes Note: If you had selected the option **Static Routing** for the initial import of the network data, no parking lots (zone connectors) were created. In this case, no adaptive import of routing data is possible for the dynamic assignment at a later time.
Dynamic assignment	Importing data for dynamic assignment. This way you ensure that the zones and/or parking lots as well as nodes of the network object type **Node** are generated.
Evaluation interval	Define the time interval for routing data of the dynamic assignment in which costs are calculated and paths searched.

Element	Description
Import routing	☑ If this option is selected, routing data for static routing or for the dynamic assignment is imported. ☐ If this option is not selected, then only the abstract network model is imported. **Warning: An ANM import generates a network file, which refers to the data in the **.anm* file. This allows for the possibility of a current loaded network to be overwritten or deleted.**
Vissim Input File	**.inpx* network file, in which the network is saved. If you do not specify a network file, adaptive import will not be possible.
ANM network file	The backup file **.panm* is copied to the folder in which the **.inpx* network file is saved.
ANM Routes File	The **.panmroutes* file is copied into the folder, in which the **.inpx* network file is saved.
Show warnings during Import	☑ If this option is selected, you must confirm every warning on the screen. The warnings are recorded in a **log file** and can be shown on the screen after the import. Note the warnings and messages in the **Messages** window during the ANM import. ☐ If this option is not selected, only the **log file** is generated and the list of warnings can be shown on the screen.
Complete routes after Import	only for adaptive ANM import: ☑ If this option is selected, the available Vissim routes, which have been interrupted by the adaptive import, are automatically completed.

3. Click the **Import** button.

Notes:

- As an alternative to the ANM import, you can drag and drop the **.anm* file from the explorer to the Vissim window.
- If no network is loaded, the **.anm* file is imported.
- If a network is loaded, which has been initially imported as a **.anm* file, you can select whether this file should be read as adaptive or initial.

A network with network objects is generated (see "Generated network objects from the ANM import" on page 372).

4. To show ANM import messages, from the **View** menu, choose > **Messages**.

6.5.4 Adaptive import of ANM data

You import ANM data adaptively, if the Vissim network was originally generated through an ANM import.

6.5.4.1 Differences to standard ANM import

When the Vissim network was generated through an ANM import, the ANM raw data were saved by Vissim and even after being manually edited in the Network editor were not discarded. When an edited **.anm* file is imported adaptively, Vissim only adopts the changes compared to the originally saved ANM data and adjusts the Vissim network accordingly. If a node is changed in the **.anm* file, only the node and the edges connected to it are generated anew. This way, only manual changes made to this small part of the Vissim network are lost. All other manual changes made to the rest of the network and the Vissim network objects added in the Network editor are kept.

If after performing an ANM import, you define network objects on links and then select adaptive ANM Import, the network objects of the following network object types are retained. This applies to Vissim versions from 8.00-14 and 9.00-05.

- Public transport stops
- Detectors
- Parking Lots
- Desired Speed Decisions
- Vehicle Travel Time Measurements
- Data collection points
- Queue Counters
- Signal Heads
- Stop Signs
- Reduced Speed Areas
- Priority rules
- Pavement Markings

Conflict areas are not retained.

6.5.4.2 Use cases and properties of adaptive ANM import

- Adopting changes of a Visum network into another network that a) was previously exported in Vissim and b) was edited manually after the import, without losing major changes.
- Importing another demand scenario (matrix and assignment results) from Visum. The static Vissim network remains unchanged. Only new parking lots, routing decisions, paths and routes may be added.

6.5.4.3 Checking nodes

When you start adaptive import, an additional check is performed of the Vissim nodes. If a Vissim node corresponds to an ANM zone, Vissim checks whether the current edge structure matches the internal ANM attributes, e.g. the ANM zone connectors IDs of the node.

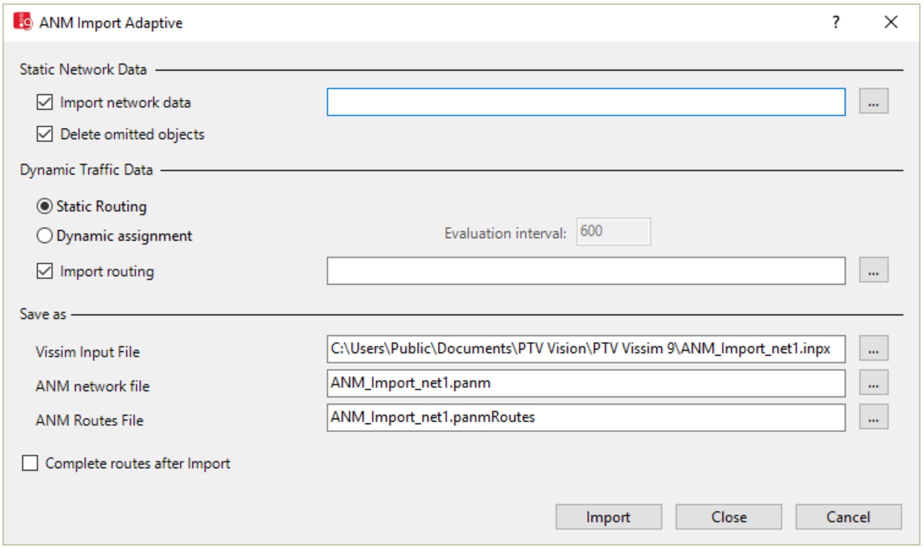
Notes:

- The node geometry cannot be restored, even if only small changes have been made, e.g. if the driving behavior type, name of a node and/or name of a link have changed.
- In this case, restoration of the node geometry is enforced for parallel links.

6.5.4.4 Starting adaptive ANM import

1. From the **File** menu, choose > **Import** > **ANM Adaptive**.

   *The **ANM Import Adaptive** window opens.*



Adaptive ANM import uses all parameters of the original ANM import (see "Importing ANM data" on page 366). Adaptive ANM import also provides the following options:

Element	Description
Delete omitted objects	Omitted objects might occur when after a first ANM export, the Visum network is edited in Visum and network objects are deleted that were part of the first import into Vissim. These network objects are not included in the **.anm* file that is newly created for adaptive import. ☑ Select this option to delete the following network objects in the Vissim network: - Network objects missing in the **.anm* file - Network objects that were automatically generated based on missing Visum network objects ☐ Deselect this option if you want the Vissim network to remain unchanged.in this respect.
Complete routes after Import	☑ Select this option to complete existing Vissim routes again that were disrupted during adaptive import.

2. Make the desired changes.
3. Click the **Import** button.

A network with network objects is generated (see "Generated network objects from the ANM import" on page 372).

6.5.5 Generated network objects from the ANM import

From exported Visum network objects, the ANM import generates the following objects in Vissim:

Visum network objects	Vissim network objects
Transport system	Vehicle types and Vehicle classes
Node	Nodes are generated as segment nodes (see "Modeling nodes" on page 705).

Visum network objects	Vissim network objects
Link	- Links can have multiple lanes, independent from the imported geometry of the lanes. In the **Messages** window, the links are named as follows: ANM-ID (from node number to node number) - One link per link section - The beginning of the widening marks a new section. - You can generate multiple connectors between two links or connectors from or to the same lane. This can be exported as of Visum 11. - The emergency stop distance for turning-connectors in the node equals the length of the widening minus 10 meters. - The time gap at the front for a permissive left with one parallel pedestrian crossing equals 2.0 seconds. Thereby it is normally possible for the oncoming right turner with a 0.5 second time gap to drive off before the left turner after the conflict area becomes free. - Through ANM import, in a link, a link segment may be very short and in a very wide angle to the adjacent link segment. For a uniform representation of the link course, Vissim deletes the short link segment and moves the adjacent link segments and their polygon points up to the middle of the deleted link segment.
Lane	- Lanes with fine lane allowance or closure of vehicle types - If adjacent ANM lanes have no common vehicle type, separate Vissim connectors are generated and not connectors with multiple lanes.
Turn	- Connectors with the respective angle with reduced speed areas. - Turning movement with right turn arrows are generated with stop signs, signal heads and conflict areas. - The conflict areas of turn volumes are generated with a status. - Multiple turning lanes of a lane to different lanes on the same exit link are allowed.
TSys closure at turn	Closure of connectors for transport systems The closure for a TSys from type PuT is only taken over in Vissim if a vehicle combination is defined in Visum which allocates the TSys.
Zone and connector	- for dynamic assignment: Parking lots or - for static routing: vehicle inputs and static routing decisions - If available, additional links and nodes - From connectors to Vissim links, which represent zone connectors, conflict areas are generated. Thereby, vehicles, which either reach or leave the "normal" network via the connectors, do not disturb the rest of traffic. - From Visum connectors, generated links and connectors are normally generated with a switched off option **Visualization**. Thereby, vehicles which drive on these links and connectors are not visible.

Visum network objects	Vissim network objects
Link attribute Type	A link behavior type is generated with number and name. A pre-defined display type is assigned: - ANM standard - Pedestrian crossing (1 m before stop line) - Zone connector A connector is assigned a link behavior type and a display type of **FromLink**.
Link attribute v0 PrT	Desired speed distribution and allocation of a desired speed decision
Stop point	Public transport stops The length and the type **Bay** or **Cap** of the PT stops result from the parameters for the stop points, which were set for the ANM export in Visum.
Vehicle journey (Amount in exported time interval)	PT lines including the optional PT telegram attributes of PT lines If the **Vehicle combination** or **TelegramLineSendsPTTelegram** attribute differ for two vehicle journeys in Visum, they are allocated to different Vissim lines.
SC and signal groups	SC and signal groups. SC of the type Vissig or Epics/Balance-Local with PTV Visum SIGNALIZATIONTYPE **Fixed time** for the respective **type**: **Fixed time** or **Epics/Balance-Local**. For Vissig SC, the **program file** *vissig_controller.dll* is used. For Epics/Balance-Local SC, *Epics_Balance-Local_Controller.dll* is used.
SG allocation to lanes according to Junction Editor	Allocation of signal groups to signal heads on lanes
Time interval	Evaluation interval for the dynamic assignment
from the control type of the node	- Signal Heads - Stop Signs - Conflict Areas
Detectors	Detectors including their length and determined vehicle classes In Visum, the position of detectors must be lane-based on lane turns.

Visum network objects	Vissim network objects
Roundabouts	Vissim creates the network objects required to form roundabouts. For dynamic assignment with mesoscopic simulation, Vissim automatically generates meso network nodes based on approach 1 (see "Rules and examples for defining meso network nodes" on page 810). These meso network nodes do not require any subsequent editing.
Follow-up gap, critical gap	In Visum, there are follow-up gaps and critical gaps for turns. At nodes they are only available for legs. Visum saves follow-up gap and critical gap values to the ANM file for links and turns.

6.5.5.1 Visum zone connectors

- If, as recommended, only a maximum of one origin and destination connection is available on a Visum node with only one adjacent node, the parking lots, vehicle input and routing decisions for these connections are placed on the available links to or from the adjacent node.
- If the connecting node has multiple adjacent nodes or if multiple zones are connected, an additional node and an additional link with a parking lot or a vehicle input and a routing decision are generated per connection. This link leads to a connector in the middle of the node. Thereby, the node geometry is not taken into consideration.

6.5.5.2 Reduced speed areas on Vissim connectors

Reduced speed areas are automatically created on connectors from a specified bend of the turn. Thereby, the coordinates of the adjacent links and the angle between the points are used.

Normally, automatically generated reduced speed areas with a length of 2 m are placed in the middle of the link.

From this, the resulting starting position *x = Length of the connector : 2 - 1 m*

If the length of the connector is < 2 m, the reduced speed area is the same length as the connector.

The following speeds are automatically assigned:

- Turns which are located on the inside (left turns in right-side traffic): 25 km/h
- Turns which are located on the outside (right turns in right-side traffic): 15 km/h

The value range of the automatically generated desired speed distributions is -10% to +10 %.

> Note: The deceleration is normally 2.0 ms^2. If the vehicle classes contain a vehicle type of the category HGV, bus or tram, the deceleration is 1.3 ms^2.

6.5.5.3 Network objects on roundabouts

Roundabouts are imported with complete roundabout geometry.

- For roundabouts Vissim defines reduced speed areas on the connectors that lead to a roundabout and desired speed decisions so that the reduced speed applies to the complete roundabout.
- Vissim shortens links on roundabouts and links which lead to or from roundabouts as that the display is correct.
- If the roundabout has a bypass Vissim generates a partial route decision and two subroutes for each bypass. Vehicles then stay on the outer lane, the bypass.
- If the roundabout has a bypass at which there are detectors, where appropriate, Vissim generates multiple detectors for multiple links. You therefore have the chance to delete unwanted detectors.
- To avoid lane changes in multi-lane roundabouts for the vehicle classes **HGV** and **Bus**, Vissim assigns the respective links and connectors the attribute **Blocked vehicle classes**. Vehicles of the vehicle classes **HGV** and **Bus** stay on the outer lane.
- Vissim inserts conflict areas in roundabouts at the following locations:
  - Entrances
  - Exits
  - Crosswalks
  - Bypass entrances
  - Bypass exits
- If a link with only one lane leads into a roundabout with several lanes, this link is connected to the roundabout via two connectors. One connector leads to the outer lane of the roundabout and the other connector leads to the inner lane of the roundabout.

6.5.5.4 Network objects for mesoscopic simulation

For imported network objects of the following network object types, the following applies for mesoscopic simulation:

Visum network objects	Vissim network objects
Nodes	In Vissim, nodes are generated from Visum nodes. Their use is defined through the attributes **Use for dynamic assignment** (**UseForDynAssign**) and **Use for mesoscopic simulation (UseForMeso**) (see "Attributes of nodes" on page 709). Nodes with the attribute **UseForDynAssign** are taken into account when path data is saved. Nodes with the attribute **UseForMeso** are taken into account when the meso graph is created (see "Mesoscopic node-edge model" on page 804), (see "Modeling meso network nodes" on page 809). Follow-up gap and critical gap at meso nodes: - The value of each critical gap at a conflict area is saved to the **Meso critical gap** of the respective meso turn conflict (see "Attributes of meso turn conflicts" on page 835). - The value of each follow-up gap at a node is saved to the **Meso follow-up gap** of the respective defining link of the meso turn (see "Attributes of meso turns" on page 833). For channelized turns, the meso node at the node exit is used. By using attribute values, in mesoscopic simulation, you can influence the behavior of vehicles at nodal points (see "Node control in mesoscopic simulation" on page 807).
Links	By default, the link attribute **Meso speed model** is set to **Vehicle-based** (see "Attributes of links" on page 409), (see "Car following model for mesoscopic simulation" on page 803). The link attribute **Meso speed** is adopted from the *v0PrT* speed of the Visum link (see "Attributes of links" on page 409).
Turns	Reduced speed areas are generated on connectors. The **Meso speed** attribute of these connectors is set to the minimum speed of the reduced speed areas.
Crosswalks	Crosswalks have no function in mesoscopic simulation. Consequently, crosswalks are not integrated into meso nodes.

6.5.6 Importing data from the add-on module Synchro 7

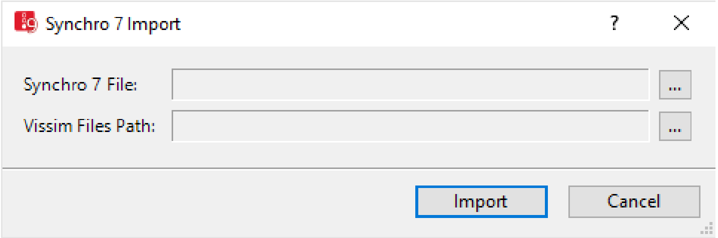
If the add-on module Synchro 7 is contained in the Vissim license, you can import a complete Vissim network including SC and signal times tables from Synchro 7.

Note: The import is not possible for data generated with a version before Synchro 7.

Vissim imports all elements from the Synchro 7 file, including network geometry, volumes, turns, vehicle compositions, node control and signalization. The complete signalization is generated during the import as individual RBC files (ring barrier control) for Vissim.

1. Save the Synchro 7 network in Synchro 7 in the data format **.csv*.
2. In Vissim, from the **File** menu, choose > **Import** > **Synchro 7**.

*The **Synchro 7 Import** window opens.*



3. Select the desired Synchro 7 file for the import.
4. Select the path for the folder in which the generated Vissim files, network files **.inpx* and **.rbc* files should be saved.
5. Click the **Import** button.

   *Vissim starts the **Synchro 7 Import** and generates a new Vissim network.*

Note: The file name of the imported **.csv* file is taken over for the **.panm* and **.inpx* files.

As for the ANM import, you can also edit a network generated with Synchro 7 import later on in the source application and then import the data adaptively into Vissim.

6.5.7 Adaptive import process for abstract network models

1. Import the network into Vissim (see "Importing data from the add-on module Synchro 7" on page 377)
2. Save the network in Vissim.
3. Edit the network in Vissim. For example, you can adjust the course of the link polygons or define the travel time sections.
4. Define the simulation parameters.
5. Carry out the simulation.
6. Check the result of the simulation. For example, you can determine that the signal control is not optimal. This must be adjusted in the source application Synchro 7 in the source network.
7. Make the desired changes in Synchro 7.
8. Save the source network in Synchro 7.
9. Export the source network for the adaptive import in Vissim.
10. Import the network in Vissim adaptively (see "Importing Synchro 7 network adaptively" on page 379)

Vissim compares the originally imported data with the new data of the abstract network model. In the case that, for example, only the differences for the signalization exist, the data for the signalization is regenerated in Vissim. Thereby, all the manually adjusted links, connectors and travel time sections are preserved in Vissim.

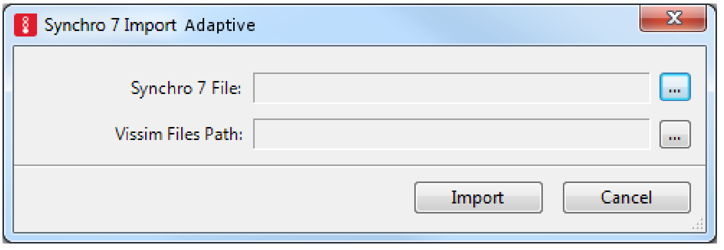
Notes:

- The adaptive import is only possible if the current network is originally generated with the same external application.
- A network must be opened at the time of data import.
- Contrary to a modeled network, an imported network is not displayed correctly on an existing background map or another, file-based background. If you are using a background image, position the imported network so that it lines up with it (see "Mapping Vissim network to background position" on page 396), (see "Positioning background image" on page 402).
- The quality of the imported network depends on the exactness of the imported data and normally requires only minimal adjustments.

6.5.8 Importing Synchro 7 network adaptively

In your Vissim network, you can import a Synchro 7 network and update it. For example, after you have changed the signalization in the source Synchro 7 network.

1. From the **File** menu, choose > **Import** > **Synchro 7 Adaptive**.



2. Check the path and the file name.
3. Click **Import**.

The Synchro 7 network is re-imported.

6.5.9 Importing openDRIVE network *.xodr

You can select an **.xodr* openDRIVE file and import it into an empty **.inpx* Vissim network file. During data import, Vissim uses the openDRIVE reference line as a basis to define links and connectors. In doing so, the software accounts for the types of openDRIVE reference lines, for example **line**, **spiral**, **arc**, **poly3**, **paramPoly3**. Vissim uses Vissim to set up the Vissimnetwork and connector data from the openDRIVE file: Vissim creates a link from every lane section. Vissim connects these links via connectors. To do so, Vissim uses openDIVE junction data. For links and connectors, Vissim sets the default attribute values and generates lanes. Conflict areas are generated by Vissim at the default locations (see "Using conflict areas" on page 560). Vissim does not generate any additional network objects and does not adopt any additional data from the openDRIVE file:

- Vissim does not generate nodes.
- Vissim does **not** adopt the following data:
  - z coordinates, all links and connectors generated are on the same level.
  - Signalization information
  - Lane markings
  - Lane change
  - Lane closures
  - Speed limits

6.5.9.1 Importing an *.xodr file and displaying the Vissimnetwork

1. In Vissim, from the **File** menu, choose > **Import** > **Synchro 7**.

   *The **CAD Import** window opens.*

2. Select the **.fma* file of your choice.
3. Click **Open**.

Vissim creates a Vissim from the openDRIVE data imported. In the network editor, the Vissim network is centered on the intersection of the equator and the international zero meridian. By default, you can edit the Vissim network.

4. Save the Vissim network file.

6.5.9.2 openDRIVE bead in Vissim - Overview

Vissim uses the following openDRIVE beads:

header	Vissim verifies the version of the openDRIVE file based on **revMajor**, **revMinor**. Vissim imports openDRIVE files up to version 1.4.
road::link::predecessor	Connector
road::link::successor	Connector
road::planview::geometry	Vissim uses subtypes to create link polygons.
road::lanes::lanesection::...lane	If in a **lane section** the subobject **lane** is present, then depending on the positive or negative index of the lane section, Vissim will generate a link with a lane for one direction or a link with two lanes, one per direction. For a lane section with a positive index, Vissim will create a link with one lane for the opposite direction. Vissim only imports and accounts for lanes of the type **driving**.
...link::predecessor	Connector
...link::successor	Connector
...width	The width is defined as a constant based on the width of the lane in Vissim which is closer to the reference line than the imported lane. The width is not defined as a polynomial. If the width of the imported lane is < 1 m, Vissim will set the

	width to 1 m.
junction	Vissim creates connectors to turns based on links. Vissim does not create nodes. predecessor and successor data from the bead **junction** are not processed.

Note: Importing routes

If your Vissim network is based on the data of an ANM import, you can calculate a PrT assignment for the original Visum network, select the demand segments of your choice, and export spline point coordinates and other data to a **.rcf* file. Select this file in Vissim and import the content into the Vissim network. Based on the data, Vissim defines network objects for vehicle routes.

6.5.10 Data stored in the **.rcf* file

Visum stores the following data in the **.rcf*file when routes are exported:

- Numbers of the zones in which routes begin and end
- Coordinates of the link polygons
- Volume for each vehicle route
- Vehicle types
- Projection information

For information on exporting routes from Visum, refer to the Visum manual.

6.5.11 Use cases for route import

- **Initial route import**: You import the route into a Vissim network based on ANM import data. The ANM import data includes a Vissim network without zones or vehicle route-specific network objects, such as static vehicle routes, static vehicle routing decisions, vehicle inputs. When dynamic assignment is performed, for OD matrices, parking lots,zones and the path file **.weg*. An initial route import defines these network objects in the Vissim network. In the network editor, Vissim also creates cross-sections for static vehicle routing decisions and vehicle inputs on links and connectors and allocates zones to the following network objects based on the zone numbers in the **.rcf* file:
  - for static vehicle routes in the **Destination zone** attribute
  - for static vehicle routing decisions in the **Zone** attribute
  - for vehicle inputs in the **Zone** attribute

  When importing routes, you allocate vehicle types from Visum to vehicle types in Vissim.

  You can also carry out an initial route import if after the ANM import into the Vissim network, you already defined vehicle route-specific network objects and allocated them to zones. If the numbers of the zones allocated to these network objects and the position of the network objects in the Vissim network match the coordinates in the **.rcf* file, the network objects are retained in Vissim.

- **Re-import routes**: You can repeat a route import, for example, because in Visum you used different parameters to calculate the PrT assignment, which resulted in new volumes that you want to assign to the vehicle inputs in the Vissim network. Since in this case, the vehicle routes, vehicle routing decisions and vehicle inputs have either been defined and assigned through the initial route import or manually by yourself, Vissim updates the vehicle route-specific network objects in the network that match objects in the **.rcf* file when the route is re-imported. To identify objects from the **.rcf* file and their corresponding objects in the Vissim network, Vissim uses the number of the assigned zone, the position of the network object in the Vissim network and the coordinates in the **.rcf*file.

You can also choose to have Vissim define new vehicle route-specific network objects and their cross-sections in the Vissim network, if the Visum network or the Vissim network have been changed in such a way that an allocation to existing network objects is no longer possible, but new vehicle routes, vehicle routing decisions and vehicle inputs or zones are defined and vehicle route-specific network objects can be allocated to zones.

6.5.12 Conditions and restrictions for route import

You can import the **.rcf* file via initial route import or via a re-import of routes under the following conditions:

- The Vissim network is larger than the Visumnetwork.
- The Vissim network was edited after the original ANM import.
- The node numbers of the Vissim network and Visum network differ from each other.
- If in the Vissim network, zones have already been defined that have the same number as the zones in the **.rcf* file, Vissim allocates these zones to the network objects.

For the position of the coordinates, Vissim takes data from the Visum network, available through projection, into account.

6.5.13 Desired speed distributions at parking lots

If you have assigned vehicle compositions to parking lots that use new vehicle routes, these vehicle compositions will be deleted through route import. As a result, at these parking lots, the desired speed distributions of the vehicle types assigned to these vehicle compositions are also lost. If you need desired speed distributions at parking lots, define the desired speed decision in the Vissim network downstream of the respective parking lot. Assign the desired vehicle class and desired speed distribution to the desired speed decision.

6.5.13.1 Importing the *.rcf file

1. In Vissim, from the **File** menu, choose > **Import** > **Route (coordinates)**.

   *The **Route import (coordinates)** window opens.*

2. Select the **.rcf* route file of your choice.

3. Click on **Open**.

*The **Route import (coordinates)** window opens.*

4. Select the desired entries.

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Routing</td><td>

Static routing: Modeling path selection in the Vissim network is based on static routing decisions, static routing and vehicle inputs (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459), (see "Modeling vehicle inputs for private transportation" on page 454).
Dynamic assignment: Modeling path selection in the Vissim network is based on dynamic assignment (see "Using the dynamic assignment add-on module" on page 692). You must have a license for the add-on module.

If your Vissim version does not include a license for the dynamic assignment add-on module, there will be no routing options available. Modeling path selection in the Vissim network is based on static routing decisions, static routing and vehicle inputs.
</td></tr>
<tr><td>Usage of network objects</td><td>
Use matching existing network objects and create missing ones: If this option is selected and depending on whether Dynamic Traffic Data is selected, Vissim checks whether there are vehicle route-specific network objects such as static vehicle routes, static vehicle routing decisions, vehicle inputs with volumes and allocated zones in the Vissim network that correspond to those stored in the *.rcf file. When dynamic assignment is performed, for OD matrices, parking lots,zones and the path file *.weg. If Vissim finds the corresponding network object, it is used. Otherwise, Vissim defines the network object. Thereby the following applies:

If in the Vissim network, the network object has been placed at the same position as specified by the coordinates in the *.rcf file for the network object and the number of the allocated zone also matches, then the network object is adopted from Vissim. The existing network object is not overwritten. To replace the existing network object with route import data, delete the network object before you import the route.
If in the Vissim network, the network object has not yet been positioned as defined in the *.rcf file, Vissim will define the network object and insert a cross-section.
If in Vissim, this network object has not been allocated to a zone, Vissim will allocate it to a zone based on the data in the *.rcf file:

for static vehicle routes in the Destination zone attribute
for static vehicle routing decisions in the Zone attribute
for vehicle inputs in the Zone attribute

Select Use matching existing network objects and create missing ones in the following cases:

When you import routes into a Vissim network for the first time and you want Vissim to create vehicle route-specific network objects.
When you re-import routes into a Vissim network and you want Vissim to define newly added vehicle route-specific network objects and retain the existing ones.

</td></tr>
</table>

	Use only existing network objects: If this option is selected, Vissim does not define any new network objects or cross-sections. Vissim replaces vehicle routing decisions and vehicle routes in the Vissim network that correspond to the network objects in the **.rcf* file. This procedure is based on the positions of the vehicle routing decisions in the Vissim network, the positions of the cross sections of vehicle routes and the identically allocated numbers of the zones. Deselect the option **Create new cross-sections**, when you want to re-import routes and do not want Vissim to define new vehicle route-specific network objects. Existing vehicle route-specific network objects are then retained. This is useful, for example, when all vehicle route-specific network objects have been defined and allocated to zones, you have used different parameters in Visum to calculate the PrT assignment, which has resulted in new volumes and you only want to allocate those to the vehicle inputs in Vissim.

Allocating vehicle types in a table	
Vehicle type in file	Vehicle type saved from Visum to an **.rcf* file
Vehicle type in Vissim the network	From the currently open Vissim network, select the vehicle type that corresponds to the vehicle type listed in Visum, in the **Vehicle type file** column.

5. Confirm with **OK**.

Vissim imports routes. Vissim redefines network objects or overwrites existing vehicle route-specific network objects or network objects of dynamic assignment. Relevant network objects to which no zone has been assigned will not be affected by route import.

By default, you can edit the Vissim network.

6. Check the network objects and data created or edited through data import to ensure that they still meet your simulation requirements.

7. Save the Vissim network file.

6.6 Exporting data

You can export the following data from Vissim:

- Nodes and edges from dynamic assignment for visualization in Visum (see "Exporting nodes and edges for visualization in Visum" on page 385)
- Nodes and edges from dynamic assignment for assignment in Visum (see "Exporting nodes and edges for assignment in Visum" on page 386)
- Polygon data of the links and walkable areas for 3ds Max (see "Exporting static network data for 3ds Max" on page 391)

- Public transport stops and public transport lines (see "Exporting PT stops and PT lines for Visum" on page 390)

Export files are saved in the folder of the currently opened file **.inpx*.

During data export, all coordinates of the Sphere-Mercator projection are written to the **.net* file. The file format **.net* Visum recognizes the projection.

You cannot export Vissim networks, that have been exported to Visum, from Visum and import them in Vissim.

6.6.1 Exporting nodes and edges for visualization in Visum

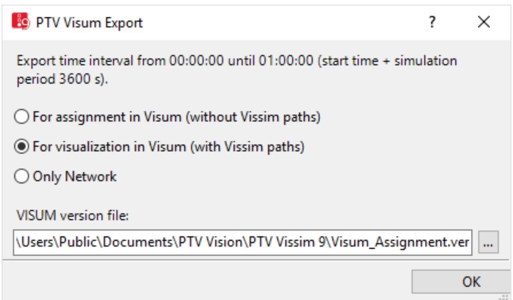
After dynamic assignment, you can export the network and the paths from dynamic assignment for visualization in Visum. You can graphically represent and analyze paths and volume in Visum. Vissim creates the following files in the process:

- Visum version file **.ver*
- Visum network file **.net*
- several Visum route files **.rim*

The data of the network file and route files is also contained in the version file.

1. Open the desired network in Vissim.
2. Make sure that a dynamic assignment was performed.
3. From the menu **File**, choose > **Export** > **PTV Visum (Nodes/Edges)**.

   The ***PTV Visum Export*** *window opens.*



4. Select **For visualization in Visum (with Vissim paths)**.
5. Click the button .
6. Specify a folder and a file name for the version file **.ver*.

> Note: You can specify only the folder and the file name for the version file. Parallel to the file **.ver*, this export creates a Visum network file **.net* and several Visum route files **.rim*.

7. Confirm with **OK**.

Notes:

- A Visum matrix file **.mtx* is generated for the export from the total demand of the Vissim matrices. Only the demand in the export time interval (start time of the simulation **+** simulation time) is taken into account for data export (see "Defining simulation parameters" on page 840). If a Vissim matrix is not completely in the export time interval, only the share which lies in the export time interval is exported. The demand is combined. Thus you do not obtain separate matrices for the various Vissim vehicle types and vehicle classes.
- Route import files are exported only if the Vissim cost files and path files are available because route import files contain the resulting routes and volumes from the dynamic assignment. A separate file **.rim* is generated for each calculation interval of the dynamic assignment.

Notes:

- If no cost files and path files exist, a warning is issued and only the network data and matrix data is exported.
- Closures of edges or connectors for vehicle classes are transferred to Visum. Thus they can be taken into account for an assignment. Visum does not use any paths by default that are not also possible in Vissim. This excludes disjoint parallel edges between two Vissim nodes that assume different turn relations or lead to different turn relations. These are subsequently exported as a common edge to Visum. This can lead the assignment in Visum to paths which cannot be used in Vissim.

Tip: You will find additional information on the subject of **Importing routes** in Visum in the Visum Help, in the section **Using interfaces for data exchange**.

6.6.2 Exporting nodes and edges for assignment in Visum

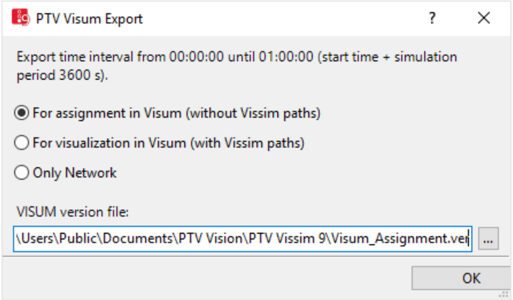
After a dynamic assignment, you can export the network and demand from the dynamic assignment for assignment in Visum (see "Using an assignment from Visum for dynamic assignment" on page 793). Vissim creates the following files in the process:

- Visum version file **.ver*
- Visum network file **.net*
- Visum matrix files **.mtx*

The data of the network file and matrix file is also contained in the version file.

1. Open the desired network in Vissim.
2. Make sure that a dynamic assignment was performed.
3. From the **File** menu, choose > **Export** > **PTV Visum (Nodes/Edges)**.

   *The **PTV Visum Export** window opens.*



4. Select the option **For assignment in Visum (without Vissim paths)**.
5. Click the button .
6. Specify a folder and a file name for the version file **.ver*.

Notes:

- You can specify only the folder and the file name for the version file. Parallel to the file **.ver*, this export creates a Visum network file **.net* and a Visum matrix file **.mtx* with the same file name as the version file. The files are saved in the folder of the version file.
- If the Vissim network contains edges which cannot be exported to Visum, these are shown in the **Messages** window. You can still execute or cancel the export. Vissim supports you when repairing the nodes and edges.

---

7. Confirm with **OK**.

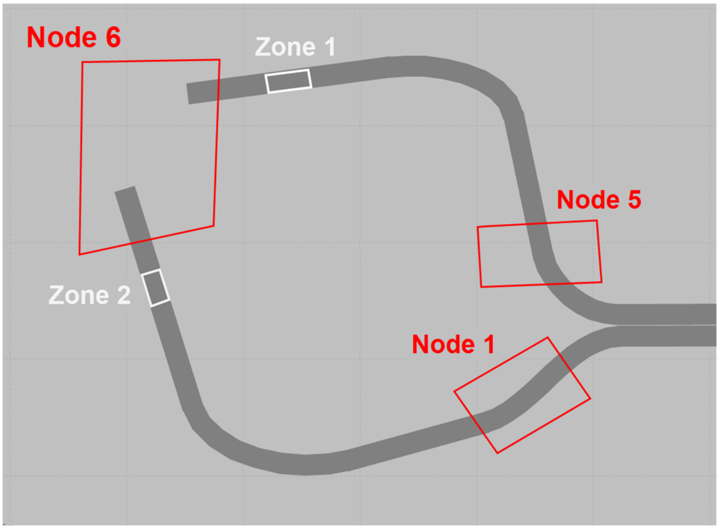
The export starts. If the Vissim network contains nodes with non-unique zone connectors (parking lot zone), a message is displayed. May you still perfom the data export or cancel it.

8. If you cancel the export, you can cancel the non-unique zone connectors (see "Canceling non-unique zone connectors" on page 387). Then perform the export again.

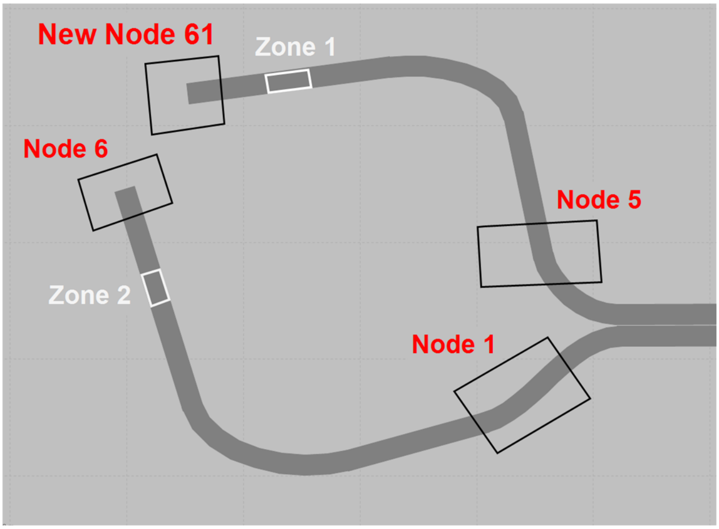
6.6.2.1 Canceling non-unique zone connectors

When exporting for an assignment in Visum, the Vissim network can contain nodes with non-unique zone connectors (parking lot zone). A message is then displayed. You can cancel the export and the non-unique zone connectors.

Example of a Vissim network with a node with a non-unique zone connector (parking lot zone):



- Cancel the zone connector with an additional node:



6.6.2.2 Opening exported data in PTV Visum

After the export from Vissim, execute the following steps in Visum:

- Open version file **.ver*: see Visum Help under **Using Visum** > **Basics of program operation** > **Opening and saving files**
- Open network file **.net*: see Visum Help under **Using Visum** > **Basics of program operation** > **Opening and saving files**

- Import routes file **.rim*: see Visum Help under **Using Visum** > **Using interfaces for data exchange** > **Reading connections and routes** > **Importing routes**
- Import matrix file **.mtx*: see Visum Help under **Using Visum** > **Modeling demand** > **Managing, showing and analyzing matrices** > **Reading an external matrix into a network model**
- Connect matrix to demand segment: see Visum Help under **Using Visum** > **Modeling demand** > **Managing demand objects** > **Managing matrices** > **Connecting demand matrices and demand segments**

Notes:

- The Visum version file created during the export contains all necessary data. It is not necessary to open the other Visum files created during the export.
- If no version file was created during the export, you can instead individually open the other files created during the export.

A Visum network with network objects is generated (see "Network objects generated in Visum from the Vissim export" on page 389).

6.6.2.3 Network objects generated in Visum from the Vissim export

From Vissim network objects, which were exported after a dynamic assignment, the import in Visum generates the following network objects:

Visum network object or attribute	Description
TSys, Mode, DSeg	Visum generates: - a PrT transport system - a PrT mode - a PrT demand segment - Vehicle types or vehicle classes from Vissim are not exported as different transport systems, modes or demand segments. - An additional DSeg-PrT is generated for each evaluation interval of the dynamic assignment from Vissim to Visum, when exporting routes from Vissim.
Nodes	Nodes are created according to the Vissim node. The Visum node numbers correspond to the Vissim node numbers. The maximum permitted node number in Visum is 2,147,483,647. Vissim nodes with a higher number are renumbered. The new numbering starts with the smallest free Vissim node number. Properties of nodes, for example, type, geometry, signaling, orientation, etc., are neither exported nor generated.

Visum network object or attribute	Description
Links	Links are generated according to the Vissim edges between the nodes. Parallel edges: In Vissim several edges are possible between two adjacent nodes. If there are several edges between two nodes, only the shortest edge is exported. The Vissim network structure is checked in the process. If the modeling is not suitable, a warning is issued, for example, for parallel edges. Details are shown in the **Messages** window.
Link number	Link numbers start with 1. Thus the numbering of links does not correspond to the Vissim link number or the number of another Vissim network object.
Link type	The link type is set to 0 for all links. Thus the link type does not correspond to the behavior type of Vissim links.
Link length	The length of link is calculated from the corresponding Vissim edge.
Number of lanes	The number of lanes is set to the minimum number of lanes of all Vissim links and connectors which belong to the edge.
Capacity PrT	*NumberVeh/h = Number of lanes • 900* *Capacity = NumberVeh/h • (simulation duration/3,600)* Duration = Simulation time (see "Defining simulation parameters" on page 840) Corresponds to "Vehicles per simulation time"
v0 PrT	**v0 IV** is calculated based on speed distributions of Vissim origin parking lots and the desired speed distribution per edge. Speed for a distribution: 85 percentile - Only the default speed distribution is taken into account for parking lots. - For speed decisions, the average of all distributions is calculated: 85 percentile of each individual distribution
Turns	Turns are generated from the Vissim edges within the node. A Visum turn is permitted if the corresponding Vissim edge exists. The capacity is set to 99,999 for all turns. U-turns are assigned type 4, other turns are assigned type 0.
Zones	Zones are created according to the Vissim zones. The zone position is calculated from the position of the parking lot which is assigned to the zone.
Zone connectors (Zone – Node)	Zone connectors are created according to the Vissim parking lots and zones.

6.6.3 Exporting PT stops and PT lines for Visum

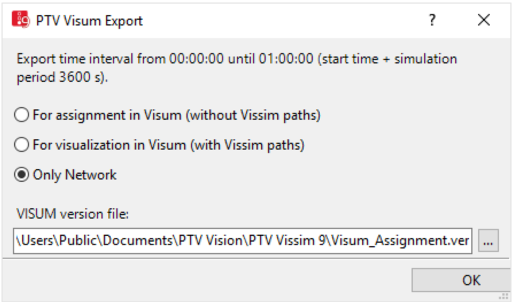
After a dynamic assignment, you can export the network of PT stops and PT lines from dynamic assignment for visualization in Visum. Paths and matrices are not exported. In Visum, PT stops and PT lines are displayed graphically. Vissim creates the following files in the process:

- Visum version file **.ver*
- Visum network file **.net*

The data of the network file is also contained in the version file.

1. Open the desired network in Vissim.
2. Make sure that a dynamic assignment was performed.
3. From the menu **File**, choose > **Export** > **PTV Visum (Nodes/Edges)**.

   *The **PTV Visum Export** window opens.*

4. Select **Only Network**.
5. Click the button .
6. Specify a folder and a file name for the version file **.ver*.

   Note: You can specify only the folder and the file name for the version file. In parallel to the **.ver* file, a Visum network file **.net* is created during data export.

7. Confirm with **OK**.

6.6.4 Exporting static network data for 3ds Max

You can export the data of polygons of links and walkable areas from your Vissim network into a **.txt* file. You can then import this file into 3ds Max.

1. From the **File** menu, choose > **Export** > **3DS MAX**.

A window opens.

2. Enter the desired file name.
3. Click the **Save** button.

The data is saved in two data blocks to the **.txt* file. There does not necessarily have to be data in both data blocks.

Structure of TXT file

- First row: coordinates used for later calculations of the correct position of polygon points.
- Data block **Links** for links
- Data block **PedestrianAreas** for pedestrian areas with coordinates of stairways/ramps and other pedestrian areas

The data blocks consist of a data record for each network object and have the following structure:

"Number of network object", comma-separated [list of coordinates X,Y,Z], new row: **g**

Example:

```
[870.10447,7438.97385,0.0]
Links
"10000",[49.36338,-17.03216,0.00000],[49.32021,-17.04763,0.00000],[52.51336,-
15.69895,0.00000],[52.51911,-15.69774,0.00000],[53.83595,-18.11195,0.00000],
[53.75370,-18.15874,0.00000],[50.40885,-19.84607,0.00000],[50.27076,-
19.89165,0.00000]
g
...
EndLinks
PedestrianAreas
"1:ped1",[144.64448,7536.22667,0.00000],[367.81823,7536.22667,0.00000],
[372.37280,7590.88147,0.00000],[39.88945,7595.43603,0.00000]
g
"2:ped2",[422.47302,7538.50396,0.00000],[531.78262,7533.94939,0.00000],
[529.50533,7588.60419,0.00000],[408.80933,7588.60419,0.00000]
g
EndPedestrianAreas
```

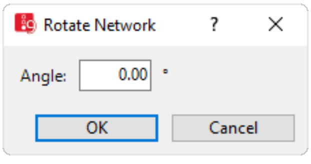
Tip: Information on data import into 3ds Max can be found in the directory *...\API\3dsMaxExport* of your Vissim installation.

6.7 Rotating the network

In the network editor you can rotate the network and its network objects. These also include static 3D models, 3D signal heads and keyframes (camera positions with a viewing direction) as well as backgrounds.

Positive values rotate the network counter-clockwise. Negative values rotate the network clockwise.

1. In the **Edit** menu, select > **Rotate Network**.



2. Enter the desired value in degrees in the **Angle** field.

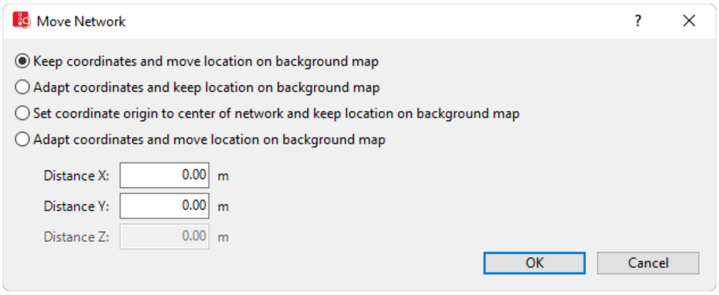
6.8 Moving the network

In the network editor you can move the entire network or selected network objects. These also include backgrounds, static 3D models, 3D signal heads and keyframes (camera positions with a viewing direction).

Background images are not moved with the network in the Z direction.

If in the Network Settings you selected imperial units, these are displayed in the **Move Network** window. Vissim converts the values entered into meters.

1. In the **Edit** menu, select > **Move Network**.



2. Make the desired changes:

Element	Description
Keep coordinates and move location on background map	➤ The network coordinates remain unchanged. ➤ The network is moved. ➤ The reference point of the map is reset. Corresponds to the function of mapping a Vissim network to a background map (see "Mapping Vissim network to background position" on page 396). Select this option, if for example the network is not mapped correctly to the background map. **Distance Z** cannot be changed.
Adapt coordinates and keep location on background map	➤ The network coordinates are reset. ➤ The network is not moved. ➤ The reference point on the map remains unchanged. Select this option, for example, to transfer coordinates of a plan or ground plan. This allows you to enter values for the distances **Distance X**, **Distance Y**, **Distance Z**.

Element	Description
Set coordinate origin to center of network and keep location on background map	- The network coordinates are set to the center point 0.000,0.000. - The network is not moved. - The reference point on the map remains unchanged. As neither the network nor the map reference point were moved, the distances cannot be changed. Select this option, for example, when you have moved the network too far, the coordinate values of the reference point in the network are now very high and you want to set them to 0.000,0.000.
Adapt coordinates and move location on background map	- The network coordinates are reset. - The network is moved. - The reference point of the map is reset. Corresponds to the function of moving selected, individual network objects in the network editor.

3. If desired and if the option selected allows for the entry, into the **Distance X**, **Distance Y**, **Distance Z** boxes, enter the values of your choice.
4. Confirm with **OK**.

6.9 Inserting a background image

You can insert true to scale digital maps as graphic files or maps from online map services as the background for a detailed Vissim network model. You can then model your Vissim network using the background.

6.9.1 Using live maps from the Internet

If your computer has a permanent Internet connection, you can display maps from online map providers as a background in 2D mode and in 3D mode for a detailed Vissim network model.

Map services provide high-resolution aerial photos, satellite images and detailed city plans for many regions. In addition to commercial offerings, such as Microsoft Bing Maps, there are such freely available map services as OpenStreetMap whose data can be used under certain conditions.

Notes:

- The use of data from map services is subject to licensing conditions. Please obtain information in advance about these conditions and take these into account when working on projects and transferring results.
- Backgrounds of Bing Maps are only available to customers who have maintenance agreements.
- Before using a map from the Internet for the first time, click the map provider's hyperlink at the bottom of the Network editor. Read the license terms.

Vissim provides some map services by default in the graphic parameters. You can select the desired map service for each open Network Editor from these. In contrast to inserted background graphics, the depiction is exclusively controlled via the graphics parameters in the relevant Network Editor. There are no other administration functions for live maps.

Vissim always uses the newest map material available. The map data from Bing Maps is not stored permanently on your hard disk as a result of the Bing Maps usage terms. The map material is newly loaded when you start Vissim start or when you open the network again. When Vissim has been opened, the map data is kept in the cache. This way, it can be displayed more quickly. The data matching the current network section is loaded at the relevant zoom level and displayed. Depending on the map service and the acceptance conditions, the live maps have different resolutions in various areas. If the maximum resolution is reached at a particular zoom level, the live map cannot be shown sharper; if you zoom in closer and the pixels on the live map are shown larger.

Advantages of using live maps available on the internet

- You do not need to prepare and load graphics files for backgrounds, but rather can start the modeling directly on the live map.
- The most up-to-date maps from the map service are always available.
- You can switch projects between computers without having to consider that paths are stated for the saved maps.
- No storage space is required for permanently storing maps on your computer.
- No storage space is required for storing maps on your computer.
- You can position your existing Vissim network on the corresponding position on the live map (see "Mapping Vissim network to background position" on page 396).
- You can display a particular map section and save it in the default layout file (see "Save Layout as Default" on page 123). In future Vissim will open with the saved map section.

URL addresses for background maps

Access to background maps available on the internet may depend on the configuration of your proxy server. For the proxy server to be able to create access rules, you might have to specify the following URL addresses when configuring the proxy server:

- BingMaps: d*ev.virtualearth.net/REST/V1/Imagery/Metadata/**
- OpenStreetMap: **.tile.openstreetmap.org/**

For further information, please visit our webpages at: PTV Visum FAQs (#15306). > **Graphics** > **(#VIS15306) The display of the background map or inserting (static) Internet maps from BING or OSM is prevented by the proxyserver.**

6.9.1.1 Activating live maps from the Internet

1. Check that your computer has an active Internet connection.

2. Make sure the reference point of the live map is not identical with the reference point of the network (see " Assigning a live map if reference points are identical" on page 397).
3. On the toolbar of the Network editor, click the button **Edit basic graphic parameters**.

   The list of graphic parameters opens (see "List of base graphic parameters for network editors" on page 171).
4. Select the option **Show map**.

   A live map from the Internet is shown in the Network Editor.
5. In the **Map provider** list, click the desired map service.

   *A live map from the selected map service is shown in the Network Editor. The entry **Default map service <map service name>** is used with **Bing Maps (Aerial View)** and **OpenStreetMap (Mapnik)**. If you do not have a license for Bing Maps, **OpenStreetMap (Mapnik)** is used automatically.*

   If the map service copyright data is shown in the Network Editor but no live map is shown, check your Internet connection.
6. Click next to the list when you want to close it.

Tip: After you select a map provider and the live map is displayed, you can use the **Toggle background maps** icon on the network editor toolbar to display or hide the live map (see "Network editor toolbar" on page 75).

6.9.1.2 Mapping Vissim network to background position

You can define a point in the live map as a reference point and assign the corresponding coordinate from Vissim.

Assignment is possible for an existing Vissim network. Thereby you assign a point of your Vissim network to a corresponding point in the live map. Vissim scales and moves the live map to the corresponding position of the Vissim network. The coordinates of both reference points are shown in the network settings.

The assignment does not change the arrangement of the network objects or their dimensions.

Note: Avoid using the **Move network** function to assign the background map and Vissim network (see "Moving the network" on page 393). This will avoid wrong scaling. Take the following steps to assign the live map:

Assigning coordinates in the live map

1. Make sure that the desired live map is shown (see "Activating live maps from the Internet" on page 395).

   By default, background maps can show a very large section of the map. It may therefore be necessary to zoom deep into the live map and shift it a long way until your Vissim network and the desired map section are displayed.

2. Zoom in on your Vissim network until you reach an enlargement where your Vissim network is shown in enough detail to allow you to assign the coordinates with the desired accuracy.
3. Right-click on the reference position in your Vissim network that you want to assign to a point on the live map.
4. Select the **Map this Point to Background Position** entry in the context menu.

   The mouse pointer becomes a cross. A preview shows the selected network objects and makes it easier to map them.
5. Move the mouse pointer to the desired point in the live map.
6. Click on the point.

   The Vissim network is positioned over the desired location on the live map. The coordinates of Vissim and the live map are assigned. You can also assign the coordinates again.
7. If desired, check the coordinates in the network settings (see "Showing reference points" on page 208).

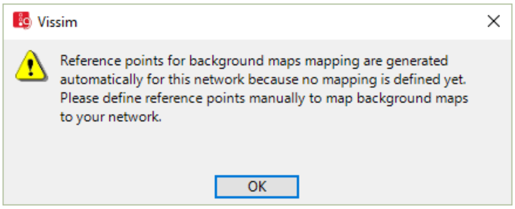
If, for example, by moving the network the coordinates of the network's reference point are outside the network, Vissim resets the reference point to the middle of the network.

6.9.1.3 Assigning a live map if reference points are identical

You cannot activate a live map, if the coordinates of its reference point are identical to the coordinates of the network's reference point. The might be due to the following:

- ANM import of an ANM file that does not yet contain any projection information. The reference points are assigned the coordinates (0.000, 0.000).
- ANM import of an ANM file with an empty projection. The reference points are assigned the coordinates (0.000, 0.000).
- You opened an **.inp* file of a Vissim version earlier than 6. The origin is set in the center of the network.
- You opened an **.inpx* file of a Vissim version earlier than 6.00-06. The origin is set in the center of the network.

If the coordinates of the reference point of the live map are identical to those of the network's reference point and you try to activate a live map, the following message is displayed:



1. Confirm with **OK**.

2. Assign the reference points of the network and the live map (see "Mapping Vissim network to background position" on page 396).
3. Check the coordinates in the network settings (see "Showing reference points" on page 208).

6.9.2 Using background images

You can insert true to scale digital maps as a background image for a detailed Vissim network model as graphics files.

You can scan similar maps and save them as a graphic file. You load the graphic file of the digital map as the background image in Vissim and display it in a Network Editor. You can move and scale the background image in the Network Editor. You then position the network objects on the background image.

- The selection of file formats is limited in the 64-bit version of Vissim. Vector graphics in the file formats **.dwg* and **.dxf* may not be displayed correctly. You can use graphics programs to convert vector graphics to raster graphics such as **.bmp* or **.jpg*.
- The memory requirement of background images depends on:
  - File format and compression of the graphic file, especially with the file format **.jpg*.
  - Screen resolution and color depth
  - In particular for 3D mode of the memory and graphic card memory
  - Thus the maximum possible file size also depends on your hardware.
- A warning is issued if the graphic file is larger than the available main memory. This provides the option of loading the file nevertheless.
- If a graphic file is too large, it cannot be displayed correctly. For example, a black or white area is displayed. Reduce the graphic file with a graphics editing program, for example, by zooming out of the screen section or by reducing the resolution.
- When a background file is read into Vissim, it is displayed in the Network Editor in tiles. You can change the screen section during the drawing of the tiles in 3D mode. As long as the background image is not fully loaded, the entire network is shown and then after that the section, which was selected before the background image was imported.

6.9.2.1 Supported file formats for background graphics

Vissim supports the following file formats for graphic files:

Raster formats	
**.bmp*	Windows bitmap, two-dimensional
**.jpg*	graphic compressed according to JPEG (Joint Photographic Experts Group) standard
**.png*	Portable Network Graphics, compressed with no loss in quality
**.tga*	Targa (Truevision Advanced Raster Graphics Array) image file, uncompressed or compressed with no loss in quality. Also saves values for alpha channel, gamma correction and text as meta-information.

Raster formats	
**.tif*	Tagged Image File Format, uncompressed or packbits compressed
**.sid*	Mr. SID files, display with best possible resolution
**.ecw*	Enhanced Compressed Wavelet, for large raster graphics, e.g. aerial images. The **.ecw* file format provides the following advantages: - high compression rates - visually flawless image quality - Contrary to other file formats, it does not cause any noteworthy delays while navigating in the Vissim network. Compression rates recommended for Vissim: - 9:1 for high quality - 20:1 for good quality with shorter loading time

Vector formats	
**.dwg*	Is updated for each new version of Autodesk AutoCAD™
**.dxf*	Is updated for each new version of Autodesk AutoCAD™
**.emf*	Windows Enhanced Metafile
**.wmf*	Windows Metafile
**.shp*	Shapefile for geodata

- 32-bit edition and 64-bit edition of Vissim support the file formats of **.dwg* versions up to DWG2013.
- AutoCAD saves the current layout with the position, translation vector and rotation with a **.dwg* or **.dxf* file, if the file is saved in AutoCAD.
- These layout settings are taken into consideration by Vissim as much as possible for the display of a loaded background graphic.
- The import of pedestrian areas and obstacles of a **.dwg* file do not take these layout settings into consideration for the graphic display.
- When you save a **.dwg* or **.dxf* file in AutoCAD, ensure that the current layout settings are suitable for use of the file as a background graphic in Vissim. This applies, in particular, before loading a **.dwg* file as a background graphic, after construction elements for pedestrians have already been imported from this file in Vissim.

6.9.2.2 Scanning maps and site plans

The following steps outline the procedure for scanning maps and site plans:

- Maps and site plans to be scanned should include a north arrow and a linear scale.
- Create an overview map that shows all relevant nodes of the transport network which is to be examined.

- Use individual signal control plans for each intersection showing detector locations, if you want to insert detectors in Vissim.
- Ensure that the scanned plans have a high contrast.
- Maps and plans should be oriented to North direction.
- Use a copy machine to reduce plans in case they do not fit the available scanner.
- A plan in A4 format should be scanned with 300 dpi resolution. The bigger the plan and the higher the resolution the bigger the bitmap file size. The memory requirements of the background file, the load time in Vissim and the network setup increase with the growing resolution.
- Save the scanned background map to one of the supported bitmap formats (for example **.bmp*, **.jpg* or uncompressed **.tif*).

6.9.2.3 Loading image file as background image

You can insert a true to scale digital map as a background image for a detailed Vissim network model. Only insert graphic files, which make up the entire examination area. You load the graphic file of the digital map as the background image in Vissim and display it in a Network Editor. The load time is dependent upon the file size.

Load the graphics in file formats which are suitable for graphics, for example, **.bmp* or **.wmf*. If a file format cannot be loaded, a message is displayed.

When loading graphic files that contain information on the size and/or position of the image, you can specify whether or not you want Vissim to consider this data.

Vissim does not use map projection. It does not consider data obtained through map projection, as could be included in files created with CAD software. Using a graphic file based on map projection to model the background of your Vissimnetwork may distort your length information and simulation results. Before loading a graphic file in Vissimthat contains data of a map projection, in the CAD program, change the coordinate system to a metric system without projection, e.g. to AutoCAD Map 3D.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

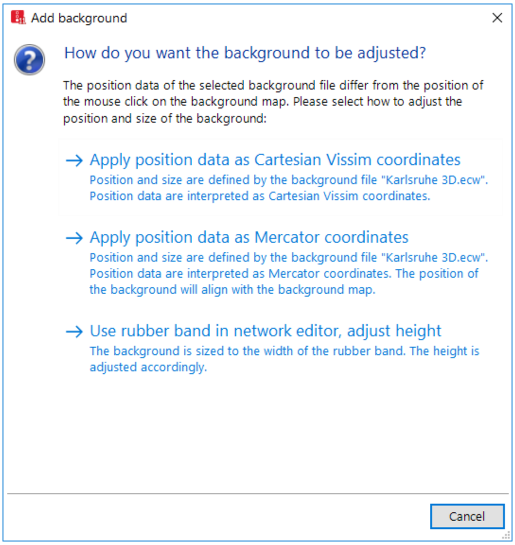
1. On the Network objects toolbar, click **Background Images**.
2. Hold down the CTRL key and in the Network Editor, right-click on the desired position.
3. From the shortcut menu, choose **Add alias**.

*The window **Select Bitmap File** opens.*

4. Select the graphic file which should be imported.
5. Click the **Open** button.

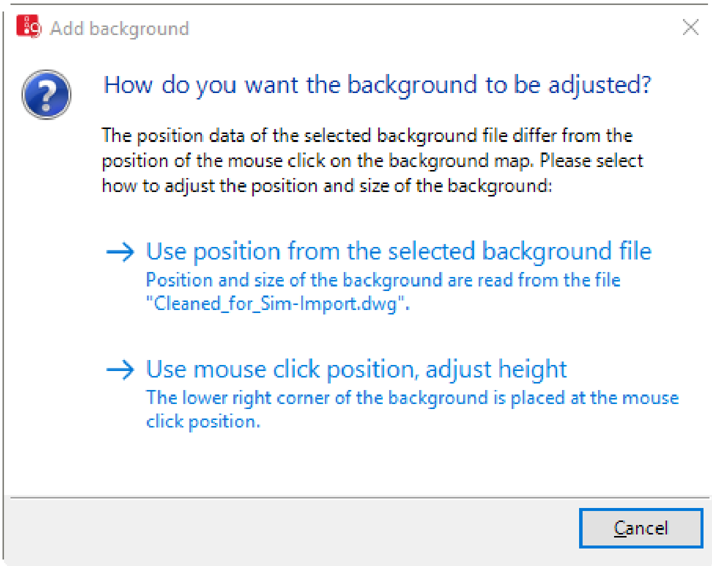
*Depending on the file format and the image size and/or position stored in the file,the **Add background** window opens and displays several options:*

- If the graphic file contains information about the size and/or position of the image and is saved in **.dwg* or **.shp*, the **Add background** window opens and displays the following options:



6. When this window opens, click the desired option.

- If the graphic file contains information about the size and/or position of the image and is not saved in the file format **.dwg* or **.shp*, but in **.bgr* or **.hgr*, the **Add background** window opens and displays the following options:



7. When this window opens, click the desired option.

Element	Description
Use position data from image file (<name of file>.<file format>)	only active if the graphic file selected is not a Vissim background image file of the format **.bgr* or **.hgr*, but e.g. an image file in the **.dwg* format: The background is inserted at the position specified in the image file.
Use position of selec- ted background	only active if the graphic file selected is a Vissim background image file of the format **.bgr* or **.hgr*:The background is inserted at the position specified in the world file.
Use position you clicked on, adjust height	Does not account for position specified in graphic file, inserts background in the Network editor at the position you click.

8. On the Network Editor toolbar, click the **Show entire network** button.

The digital map is shown in its entirety.

*The name and a miniature view of the digital map is shown in the **Background images toolbar**.*

6.9.2.4 Positioning background image

You can position a background image in a Network Editor. Next, you can scale a background image and place a network object on a background image.

1. On the Network objects toolbar, click **Background Images**.
2. In the Network Editor, click on the background image of your choice.

A background image is marked.

3. Hold down the left mouse button and drag the background image to the position of your choice.

   When you move, the background becomes transparent.

4. Release the mouse button.

6.9.2.5 Scaling the background image

A loaded background image is not represented according to scale even when the file has a scale. For the precise modeling of a network, you must scale the map precisely. Use a large distance, for example > 100 m. For the adjustment of the background image and the Vissim units, zoom in on an object or a link with original measurements known to you. For example, this can be the scale of the map or the edge between two corners of a building or geographical points.

1. On the Network objects toolbar, click **Background Images**.
2. In the Network Editor, click on the background image of your choice.

   A background image is marked.

3. Right click on the Network Editor.
4. From the context menu, select the entry **Set Scale**.

   The mouse pointer becomes a cross.

5. Click on the desired starting point of the edge and drag it while holding down the mouse button to the desired end point of the edge.
6. Release the mouse button.

   *The **Scale** window opens.*

7. Enter the original length of the edge.
8. Confirm with **OK**.

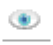
Tip: Alternatively, you can point the cursor to a corner of the background in the network editor and hold the mouse button down and drag to reduce or increase the background to the desired size.

6.9.2.6 Hiding and showing background image

1. Select from the menu **View** > **Backgrounds**.

   The Background Image toolbar is displayed.

2. Click on the desired icon:

Element	Description
	Background image is currently displayed. Hide background image.
	Background image is hidden. Display background image.

6.9.2.7 Attributes of background images

1. From the **Lists** menu, choose > **Graphics & Presentation** > **Background Images**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

*The **Background Images** list opens.*

The list in the tab contains, amongst others, the following attributes:

Column	Description
PathFilename	Path and filename of graphic file
CoordTRX	**Coordinate top right (x)**: Coordinate (x) of the upper right corner of the graphic
CoordTRY	**Coordinate top right (y)**: Coordinate (x) of the upper right corner of the graphic
CoordBLX	**Coordinate bottom left (x)**: Coordinate (x) of the bottom left corner of the graphic
CoordBLY	**Coordinate bottom left (y)**: Coordinate (x) of the bottom left corner of the graphic
CoordBLPt	**Coordinates bottom left**: Corner point **CoordBLX** and **CoordBLY**. When you change one value, the value associated with it in **CoordTRPt** is automatically changed as well.
CoordTRPt	**Coordinates top right**: Corner point **CoordTRX** and **CoordTRY**. When you change one value, the value associated with it in **CoordBLPt** is automatically changed as well.
Level	Name of level
ZOffset	Z offset: Height relative to the height of the level. Avoid values between -0.2 m and -0.5 m.
TileSizeHoriz	**Tile size horizontal**: Width The background image is loaded tile by tile. Tiles are recommended for large networks with large background images. In 3D mode in particular, the time for opening the graphic files depends on the file size of the graphics files.
TileSizeVert	**Tile size vertical**: Height

Column	Description
Res3D	Maximum **3D resolution** for displaying background images in 3D for raster graphics and vector graphics. This depends on your computer's graphics hardware and the number of currently loaded background images. - Very low: 1,024 - Low: 2,048 - Average: 4,096 - High: 8,192 - Tiles
AnisoFilt	**Anisotropic filtering**: ☑ If this option is selected, the display quality of textures is improved when viewed from a very flat angle.
Angle	Angle in degrees, by which the background is rotated. The background is also rotated when the Vissim network is rotated (see "Rotating the network" on page 392). You can rotate a background in the Network editor (see "Rotating network objects" on page 354).

2. On the list toolbar, in the **Relations** list, click > **Level**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the right contains attributes and attribute values of levels allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119).

3. Enter the desired data.

The data is allocated.

6.9.2.8 Modeling the network for background images

- You can temporarily create Vissim links as an aid for locating the corners of buildings, in order to place more graphic files with higher detailing correctly in the Vissim network.
- Finally, model the network and intersections more carefully with regard to the detailed graphic files.
- Then delete links which are no longer needed.
- To show or hide all backgrounds, in the active Network Editor, use the key combination CTRL+B.

6.10 Modeling the road network

When modeling transport infrastructure in Vissim, you can choose between a basic test environment and true to scale traffic flow modeling.

Testing environment

A less detailed model is sufficient, if you e.g, want to analyze several test cases for traffic-actuated VAP or VS-PLUS logic by activating the detector types manually. However, if you want to simulate traffic flows for performance analyses, you will need a more sophisticated model.

For testing environments, it is sufficient to position approach links of intersections roughly where you want them. Add signal heads and detectors to the approach lanes, so that you can later easily identify these network objects. True to scale modeling is not required. So network and nodes do not have to be based on scanned plans.

True to scale modeling

Vissim allows you to model traffic flows and real conditions true to scale. You can position the network objects of your choice accurately to the millimeter, e.g. links and connectors, signal groups and signal heads, individual intersections, a specific sequence or network of nodes. To ensure true to scale modeling:

- From Visum, CROSSIG, P2 or other applications that support Vissim network files, import a true to scale network (see "Importing a network" on page 361)
- You use a background map for modeling.
- If you do not want to use a background map for modeling:
  - Import true to scale data from Synchro 7 (add-on module for optimization of signal control (see "Importing data from the add-on module Synchro 7" on page 377) and (see "Importing Synchro 7 network adaptively" on page 379)).
  - Scan maps of the area.
  - Add the scanned maps as the background (see "Inserting a background image" on page 394).
  - Your Vissim network is based on these background graphics.

6.10.1 Modeling links for vehicles and pedestrians

You can model links on which vehicles and pedestrians move. This vehicle and pedestrian must be defined as vehicle types.

Notes:

- In Viswalk, pedestrians can also move on links. These links must be defined as pedestrian area (see "Defining pedestrian links" on page 924).
- Ensure that a background image is loaded and the scale is selected (see "Inserting a background image" on page 394)

Within a link section, the number of lanes cannot be changed. If the number of lanes should deviate, you must set a link for every link section. Subsequently, you can split links and change the number of links (see "Splitting links" on page 419).

Deleting a link or lane: If for a link the attribute **Has overtaking lane** is selected, you can delete the link. If you want to delete a lane, the corresponding link must have at least three lanes, including the overtaking lane.

6.10.1.1 Defining sequence and modeling techniques for links

- Define the links for main roads in one direction.
- Add intermediate points and model the curvature of a link.
- Define the opposite lane. Thereby you can generate the newly created link with a different number of lanes. Allow the links to run corresponding to the line of a road.
- If you add additional links, connect them via connectors.
- Determine the number of arms per junction and add links. In the junction area, allow links in the middle of the junction to begin and end. If the number of lanes changes, pay attention to the correct connection via connectors.
- Define the number of lanes per link on the nodes.
- Define the number of turning lanes in the node.
- Define the possible turns, lane expansions and lane reductions at the connectors.

6.10.1.2 Defining links

Define links in the Network Editor in the movement direction. You can define straight links or add intermediate points to change the course of the link. You can also define a circular link to model a roundabout in the desired size and in a symmetrical, circular shape.

Tips:

- Alternatively, you can define links in the Network Editor via the context menu > **Add New Link**, if the network object type **Links** is selected in the network object toolbar.
- You can set that when you enter network objects you do not have to press the CTRL key (see "Right-click behavior and action after creating an object" on page 152).

Defining straight links

1. On the Network objects toolbar, click **Links**.
2. In the Network Editor, point the mouse pointer to the desired starting point of the link.
3. Press the CTRL key, hold down the right mouse button and drag the mouse to the desired end point of the link.
4. Release the keys.

   The link is shown in color in the network editor. The link contains attribute values and lane attribute values of the link last created. If no link was created since the network was loaded or if the previously created link was deleted, default values are used. The movement direction is shown by arrows at the edges of the link. The ***Link*** *window opens. The new link is assigned the attribute* ***Number****, which is the next higher number available for links.*

5. Edit the attributes (see "Attributes of links" on page 409).
6. Confirm with **OK**.

The attributes are saved in the ***Links*** *list.*

Using intermediate points to define links with a curve shape

While dragging open a link, you can set intermediate points to change the link direction. This allows you to already model complex curves when inserting a link.

1. On the Network objects toolbar, click **Links**.
2. In the Network Editor, point the mouse pointer to the desired starting point of the link.
3. Hold down the CTRL key and the right mouse button, and drag the mouse to the desired first point of a curve.

   Release the CTRL key. Keep the right mouse button held down until you have inserted the entire link. Use the left mouse button to insert intermediate points.
4. Hold down the right mouse button and click.
5. Continue to add intermediate points this way, until you have reached the final point of your link.
6. Release the keys.

   The link is shown in color in the network editor. The movement direction is shown by arrows at the edges of the link. You can move the intermediate points, which allows you to model the link precisely. If you zoom out too far, arrows and intermediate point cannot be fully displayed. The ***Link*** *window opens.*
7. Edit the attributes (see "Attributes of links" on page 409).
8. Confirm with **OK**.

The attributes are saved in the ***Links*** *list.*

Defining a circular link

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, right-click the desired starting point of the circular link.
3. From the shortcut menu, choose **Add Circular Link**.

   The circular link is displayed when the mouse pointer points to the Network editor. The movement direction is shown by arrows at the edges of the link. By default, the circular link has 48 intermediate points. The visualization of intermediate points depends on the diameter of the roundabout and the magnification selected for the Network editor.
4. In the Network editor,use the mouse pointer to drag the circular link in the desired direction until the required size is reached.

   The length and diameter of the circular link are displayed in the status bar, to the right of the coordinates of the mouse pointer position.

5. Click.

   *The **Link** window opens. The new link is assigned the attribute **Number**, which is the next higher number available.*

6. Edit the attributes (see "Attributes of links" on page 409).
7. Confirm with **OK**.

*The attributes are saved in the **Links** list. Vissim adds a link from the end of the circular link to the beginning of the circular link and thus closes the circular link. The attributes of this connector are saved to the **Links** list. The new connector is assigned the attribute **Number**, which is the next higher number available for connectors.*

You can use the same command for the roundabout that are available for links, e.g.:

- Invert direction(see "Inverting direction" on page 420)
- Add lane (see "Inserting lanes on the left or right" on page 420)
- Add or delete points (see "Adding points to a link" on page 431)

6.10.1.3 Attributes of links

The **Links** window opens when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Links list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

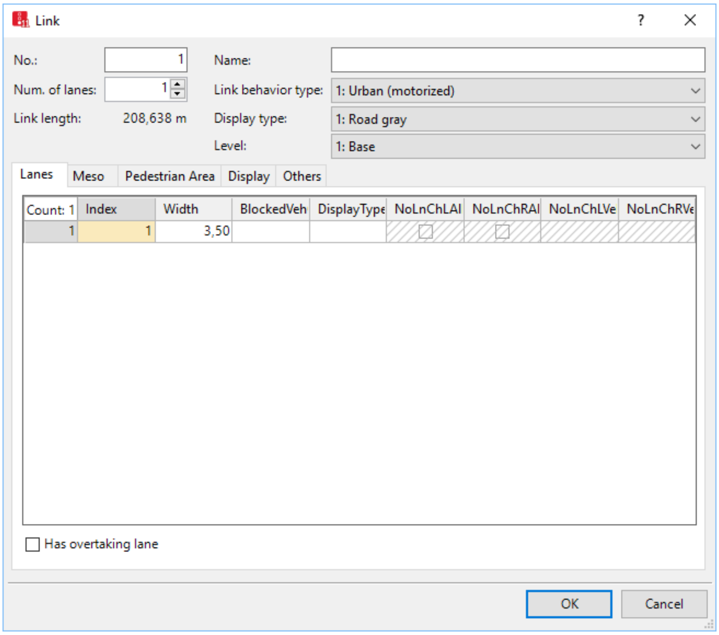
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



Basic attributes of links

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number of the link
Name	Designation of the link
Count Lanes	**Number of lanes** (**NumLanes**). The table in the **Lanes** tab is automatically adjusted. If there already is a lane and you increase the number of lanes, the new lane is inserted in the Network editor and adopts attributes from the existing lane.
Link Length	**Length2D**: Length of the link in meters **Length3D** accounts for z-offset of the link

Element	Description
Behavior type	**Link Behavior Type** (**LinkBehavType**): Driving behavior for the link (see "Defining link behavior types for links and connectors" on page 318) and (see "Defining driving behavior parameter sets" on page 282). If the **Is pedestrian area** attribute (**Use as pedestrian area** option) is selected, the behavior type **None** is automatically selected.
Display type	Colored display of the link (see "Defining display types" on page 320). In the coupled list **Lanes**, in the **Display Type** column, you can edit the **Display Type** attribute for individual lanes of the link. The coupled list **Lanes** is selected in the **Links** list, in the **Relations** list box.
Level	For modeling of multistory buildings or bridge structures: level on which the link is located
Has passing lane	**HasOvtLn**: The inner lane may only be used for overtaking maneuvers on the oncoming lane. This is only possible on links with at least two lanes (see "Modeling overtaking maneuvers on the lane of oncoming traffic" on page 508). ☑ If this option is selected, for right hand traffic, the outer left lane is displayed as the passing lane, with hatched background. For left hand traffic, the lane on the far right is the passing lane, displayed with hatched background. If a passing lane and regular lane of a link of the opposite direction overlap for long enough, the overlapping area may be used for passing. Only select this attribute for links on which passing is actually allowed in reality. Avoid passing lanes on which overtaking is not possible in reality, e.g. at junctions or in traffic controlled areas. You can also select this attribute for several, successive links that are connected via connectors and have at least two lanes. Vehicles can then use the entire overlapping area for overtaking maneuvers. If the passing lane is closed for a vehicle class, the vehicles of this class cannot use the passing lane for overtaking. You can place other network objects, e.g. data collection points, on passing lanes. Passing lanes are not shown in 3D mode.

Lanes tab

The list in the tab contains, amongst others, the following attributes:

Column	Description
Index	Unique number of the lane. You cannot change this entry later on.
Width	Width of the lane If several lanes are defined, several rows are displayed. You can define different widths. The width has an effect on: - the graphic display of a link - the possibility of whether a vehicle can overtake within a lane. For this, overtaking within a lane must be selected in the driving behavior parameters (see "Editing the driving behavior parameter Lateral behavior" on page 308). The width does not have any effect on the speeds.
BlockedVehClasses	**Blocked vehicle classes** on this lane. - If the link of the lane has a vehicle input, the vehicles of this vehicle class are not used on this lane. - The vehicles of this vehicle class do not carry out a lane change into this lane. This also applies if this would be necessary because of their route. - Vehicles of the particular vehicle class attempt to leave the lane as quickly as possible, if an adjacent lane is not blocked for the vehicle class. - If all of the lanes in a link are blocked for a vehicle class, the vehicles of this vehicle class travel along the link without changing lane.
DisplayType	Color of lane (see "Defining display types" on page 320)
NoLnChLAllVehTypes, NoLnChRAllVehTypes	**No lane change left – all vehicle types** and **No lane change right– all vehicle types**: If this option is selected, vehicles may not change lanes. A prohibition of lane changes is shown in the 2D and 3D mode by means of a solid line.
NoLnChLVehClasses, NoLnChRVehClasses	**No lane change left - vehicle classes** and **No lane change right - vehicle classes**: Vehicle classes, whose vehicles must not change from a chosen lane to the adjacent lane in the direction of travel. A prohibition of lane changes is shown in the 2D and 3D mode by means of a solid line.

Notes:

- If vehicles are to be able to overtake within a lane, you must select overtaking within a lane in the driving behavior parameters (see "Editing the driving behavior parameter Lateral behavior" on page 308).
- For lane changes, you must define links with multiple lanes. You cannot define lane changes between adjacent links.
- No lane change also applies for a change of lane which would have to be made according to the route. Therefore, make sure that lane changes due to the route are either completed before the prohibition of lane changes, or can only be made after the prohibition of lane changes.
- Cooperative lane change ignores the lane change ban (see "Editing the driving behavior parameter Lane change behavior" on page 300).

The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.

Meso tab

Link attributes for mesoscopic simulation:

Long name	Short name	Description
Meso speed model	**MesoSpeedModel**	Specifies how the speed of vehicles on this link is determined. • **Vehicle-based**: Vehicles always drive at their desired speed • **Link-based**: Vehicles drives at the speed defined for the attribute **Meso speed**.
Meso speed	**MesoSpeed**	Meso speed is used exclusively in combination with the meso speed model **Link related** (see "Car following model for mesoscopic simulation" on page 803). In this case, the meso speed defines the speed for all vehicles on the link. Default value 50.0 km/h
Meso follow-up time	**MesoFollowUpGap**	Follow-up gap between two vehicles in the same traffic flow. Edit this attribute in the **Meso turns** list or in the coupled list **Nodes** - **Meso turns** (see "Attributes of meso turns" on page 833), (see "Attributes of nodes" on page 709).

Pedestrian Area tab

Attributes of links, if they are meant to be used by pedestrians and not by vehicles:

LongName	Short name	Description
Is ped-estrian area	**IsPedArea**	Only with Viswalk: ☑ If this option is selected, the link is defined as a pedestrian area (see "Modeling links as ped-estrian areas" on page 922).
Pedestrian Behavior section		

LongName	Short name	Description
Area behavior type	**AreaBehavType**	Is used to model occasional changes to the speed or other parameter (see " Modeling area-based walking behavior" on page 932).
Desired speed factor		Factor for changing the desired speed of all pedestrians, default value100 %, value range 10 % to 300 %. Using the desired speed factor, you can reduce the desired speed on the route when pedestrians move slower than at their originally desired speed, for example when walking on bad ground or carefully crossing a road. Using the desired speed factor, you can increase the desired speed on the route when pedestrians move faster than at their originally desired speed, for example when quickly crossing a road.
Conflicts with vehicles section		
Consider vehicles in dynamic potential	**ConsVehInDynPot**	: ☑ If this option is selected, dynamic potentials of pedestrians consider current vehicle positions in conflict areas. This attribute also has an effect on the oncoming lane. For the pedestrian route locations of the pedestrian route leading across the link and for which you have selected the dynamic potential, select a calculation interval that is sufficiently short.(see "Dynamic potential attributes" on page 972).
G for vehicles	**VehDynPotG**	**Vehicles dynamic potential G**: Dynamic potential parameters affecting the general strength of vehicles, default value: 3. This attribute also has an effect on the oncoming lane.

Display tab

Attributes for the display of the link. The attributes do not influence the driving behavior.

The tab contains, amongst others, the following attributes:

Element	Description
3D	
z-offset (start)	**ZOffset** (**zOffsetStart**): Starting point of z-coordinates of link for 3D display.
z-offset (end)	**z-offset (end)** (**zOffsetEnd**): End poiont of z-coordinates of link for 3D display.

Element	Description
By default, z-offset (start) and z-offset (end) do not have any impact on the driving behavior when it comes to upward or downward gradients. If the z-coordinates in your Vissim network have been entered correctly, you can have Vissim calculate upward and downward gradients. In this case, the option **Use gradient from z coordinates** must be selected (see "Selecting network settings for vehicle behavior" on page 203). If you change the values of the z-offset (start) or the z-offset (end) and have inserted intermediate points into the link, Vissim will recalculate the z-offset values of the intermediate points. To ensure that the upward or downward gradient is harmonious, Vissim calculates a spline for the vertical course of the link.	
Thickness (3D)	**Thickness** for display of the link in 3D mode.

Element	Description
Visualization	
Individual vehicles	**Show individual vehicles** (**ShowVeh**): ☑ Select this option to show vehicles in the 2D mode. ☐ If this option is not checked, no vehicles are indicated in the 2D mode. With this, you can indicate underpasses or tunnel sections. This option applies for the entire link. Therefore you must define a separate link for each underpass or for each tunnel. Note: Do not use this option in 3D mode, but rather model the height details of the links correctly!
Show classified values	**ShowClsfValues**: ☑ Select this option to show classified values (not to show the display type selected). To show classified values, in the graphic parameters for links, select a color scheme and an attribute (see "Assigning a color to links based on aggregated parameters" on page 179).
Label	☐ If this option is not checked, the labeling of the link is not shown.
Show link bar	☑ Select this option to to show links with link bars (see "List of graphic parameters for network objects" on page 161).

Other tab

The tab contains, amongst others, the following attributes:

Element	Description
Gradient	Uphill and downhill slopes of the link in percent. Downhill slopes have a negative value. The value impacts the driving behavior via the maximum acceleration and maximum deceleration on a link. - by -0.1 m/s² per gradient percent incline. The maximum accelerating power decreases when the deceleration power increases. - by 0.1 m/s² per gradient percent downgrade. The accelerating power increases when the deceleration power decreases. Per default, uphill and downhill slopes in 3D mode do not affect the display (z-coordinates) of links. You can edit z-coordinates via the **z-Offset** attribute. If the z-coordinates in your Vissim network have been entered correctly, you can have Vissim calculate uphill and downhill slopes. In this case, the option **Use gradient from z coordinates** must be selected (see "Selecting network settings for vehicle behavior" on page 203).
Overtake only PT	**OvtOnlyPT**:Vehicles which travel on a route with at least two lanes may overtake a stationary Public Transport vehicle during the change of passengers if there is enough room ahead of it. In all other cases, overtaking is not possible.

Element	Description
Evaluation	
Vehicle record	**Vehicle record active** (**VehRecAct**): ☑ Select this option to record link data for the vehicle record.
Lane changes evaluation active	**LnChgEvalAct**: ☑ Select this option to record lane data for the **Lane Change** evaluation.
Link evaluation active	**LinkEvalAct**: ☑ Select this option to record link data for the link evaluation. If you selected the attribute **Use as Pedestrian Record**, you can still record link data for the pedestrian record.
Segment length	The segment length which is taken into account in the link evaluations
Network performance evaluation active	**NetPerfEvalAct**: ☑ If this option is selected, the link is taken into account for network performance evaluation. To select individual link sequences using network performance evaluation, deselect this attribute for all other links. In the network performance evaluation of a micro simulation or a mesoscopic simulation, parking spaces and vehicle inputs are only counted for the output attributes **Demand (latent)** and **Delay (latent)**, if for their links, **Network performance evaluation active** is selected. The output attribute **Vehicles (arrived)** only records vehicles that have driven on a link for which the attribute **Network performance evaluation active** is selected. In the network performance evaluation of a mesoscopic simulation, the data is used for all meso edges that lead across at least one link for which the attribute **Network performance evaluation active** is selected.

Element	Description
Dynamic assignment	
Cost	distance-dependent **costs per km** (**CostPerKm**). Only relevant for **Dynamic Assignment** (see "Using the dynamic assignment add-on module" on page 692)
Surcharge 1 **Surcharge 2**	one-time surcharges that are taken into account for path evaluation. In the dynamic assignment, the costs for the vehicles which travel on this link are determined.

Element	Description
Overtaking in the opposing lane	
Overtaking speed factor:	**OvtSpeedFact**: Factor by which the vehicle wants to overtake, increasing its desired speed. Default 1.30.

The following attributes are only relevant for modeling overtaking maneuvers on the oncoming lane:

Element	Description
Look ahead distance	**Look ahead distance for overtaking** (**LookAheadDistOvt**): Distance that the overtaking vehicle can view on this link, upstream of the oncoming lane. At this distance oncoming traffic is perceived by drivers. At the end of this distance a virtual, oncoming vehicle is assumed, if on this link there is a vehicle input, a PT line or an inbound connector further upstream. The shorter the look ahead distance for overtaking is, the smaller the likelihood of being able to overtake. Default 250 m.
Assumed speed of oncoming traffic	**AssumSpeedOncom**: Speed of vehicles in oncoming lane in the following situations: - Vehicles that appear in the oncoming lane at the end of the look ahead distance of the vehicle wishing to overtake - Vehicles that may appear within the look ahead distance of the vehicle wishing to overtake, e.g. due to vehicle input or a parking lot located there. The higher the assumed speed of oncoming traffic is, the smaller the possibility of overtaking in these situations. Default value 60 km/h. If there is an oncoming vehicle within the look ahead distance of the vehicle wishing to overtake, Vissim uses its current speed.

The following attribute is only relevant for matrix correction:

Element	Description
Counted data	**VehClassCountedData**: Shows the count data of the selected vehicle class, if configured in the matrix correction procedure (see "Correcting demand matrices" on page 789).

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Dynamic vehicle routing decisions (see "Defining dynamic routing decisions" on page 762)
- Lanes: The attributes are described further above.
- Vehicles: only during running simulation: attributes of vehicles on this link
- Vehicle travel time measurements (see "Defining vehicle travel time measurement" on page 447)
- Vehicle route closures (see "Attributes of route closures" on page 484)
- Vehicle routes (partial) (see "Attributes of partial vehicle routes" on page 488)
- Vehicle partial routing decisions (see "Attributes of partial vehicle routing decisions" on page 487)
- Vehicle inputs (see "Modeling vehicle inputs for private transportation" on page 454)
- Blocked vehicle classes for dynamic assignment (see "Defining the vehicle class" on page 280)
- Conflict areas (see "Using conflict areas" on page 560)
- Managed lane routes (see "Attributes of managed lane routes" on page 479)
- Managed lanes routing decisions (see "Attributes of managed lanes routing decisions" on page 478)
- Public transport lines (see "Modeling PT lines" on page 518)
- Partial PT Route (see "Attributes of partial PT routes" on page 540)
- PT partial routing decisions (see "Attributes of PT partial routing decisions" on page 539)
- Parking lots (see "Modeling parking lots" on page 493)
- Parking routes (see "Attributes of parking routes" on page 476)
- Parking routing decisions (see "Attributes of parking routing decisions" on page 474)
- Points 3D: Coordinates and **ZOffset** of start point, intermediate points and destination point of link or connector
- Static vehicle routes (see "Attributes of static vehicle routes" on page 470)

- Static vehicle routing decisions (see "Attributes of static vehicle routing decisions" on page 468)
- Queue counters (see "Modeling queue counters" on page 450)
- Link evaluation segments: not for links for which attribute **Use as pedestrian area** has been selected. Segments of link for link evaluation (see "Showing data from links in lists" on page 1103). Start point, end point and length of each segment on the link in [m].

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.10.1.4 Splitting links

You can divide a link into two links.

1. On the Network objects toolb**ar, click Links**.
2. In the network editor, right-click on the desired position in the link.
3. In the context menu, select **Split Link Here**.

The ***Split Link*** *window opens.*

4. Make the desired changes:

Element	Description
Splits at	Length from the start point of the link into which you have clicked. You can change the length.
1. New Link	Number and length of the first link. You cannot change this value.
2. New Link	Number and length of the new, second link. You can change the number. The link is shown in the link list.
Generate connector automatically	☑ If this option is checked, the links are connected with a connector. The connector is shown in the link list.

5. Confirm with **OK**.

The link is divided. The first link is highlighted in the network editor.

6.10.1.5 Generating an opposite lane

Next to a particular link you can generate a lane with the same route which runs in the opposite direction. For this, you enter the number of lanes. The new link is independent from the original link.

1. Right-click the desired link.
2. In the context menu, select the entry **Generate Opposite Direction**.

   The ***Generate Opposite Direction*** *window opens.*

3. Enter the number of lanes.
4. Confirm with **OK**.

*The link is shown in the network editor and saved in the **Links** list.*

6.10.1.6 Inserting lanes on the left or right

In the network editor you can individually insert further lanes to the left or right of the link in the movement direction.

1. On the Network objects toolbar, click **Links**.
2. Right-click the desired link.
3. In the context menu, select the entry **Add Lane Left** or **Add Lane Right**.

> Tip: Alternatively, you can add further lanes in the **Link** window in the **Lanes** tab via the lane list context menu.

The new lane is inserted into the Network editor and adopts attributes from the existing lane.

6.10.1.7 Inverting direction

You can invert the direction of travel for a link.

1. On the Network objects toolbar, click **Links**.
2. Right-click the desired link.

   The edges of the link are marked with arrows in the movement direction.
3. Select the entry **Reverse Direction** in the context menu.

   The movement direction for the link is reversed. The edges of the link are marked with arrows in the movement direction.

6.10.1.8 Using nodes defined from individual link segments

The following applies if you have exported nodes from Visum which are defined by link segments:

- If you select the polygon, the segment nodes in the polygon are not highlighted.
- You cannot change the two statuses **belongs to selection** and **does not belong to selection** of these segment nodes by clicking on them.
- Movement of the links does not have any effect. The position of the label may change.
- If you delete the node, the segment nodes in the polygon are also deleted.

6.10.2 Modeling connectors

In order for vehicles to continue their journey on the following links, you must connect these links with connectors. You also set connectors for modeling the turn relations at nodes.

Connectors can only be inserted between two links. You cannot connect connectors to each other. Connectors have attributes and options which are comparable to those of links (see "Attributes of connectors" on page 422). You may move the start or end point of a connector to a different link later on.

6.10.2.1 Defining connectors

You can insert connectors in the network editor, for example between the end of a link and the start of another link. When dragging open a connector, you can set intermediate points and change the direction there. This also allows you to already model complex curves when inserting a connector.

The connector can connect two links that have different z-coordinates. In this case, Vissim calculates the z coordinates of the splines points of the connector for a harmonious course with a spline.

Notes:

- Overlap the link and the connector as little as possible. This avoids modeling errors.
- Make sure there is no connector, connecting links with a large difference in altitude (e.g. 0.5 m) over a very short distance (e.g. 1 m), particularly when you select the option **Use gradient from z coordinates** (see "Selecting network settings for vehicle behavior" on page 203).
- When you open **.inp* files of previous Vissim versions, Vissim 6 deletes the two automatically created intermediate points that lie very close to the beginning and end of the connector. This makes it easier for you to select the start and end point of the connector in the Network editor.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Links**.
2. With the mouse pointer, point to the position in the link at which the connector is to begin.

   The edges of the link are marked with arrows in the movement direction.
3. Press the CTRL key, hold down the right mouse button, and drag the mouse to the first desired end point of a curve.

Until the whole connector has been inserted, keep the CTRL key and the right mouse button held down. Use the left mouse button to insert intermediate points.

4. Hold down the CTRL key and the right mouse button and click.
5. Continue to add intermediate points until the connector reaches the position within the link where you want it to end.

   *In the network editor the connector is shown in color between the two links. The number of intermediate points of the connector just previously created is added to this one. The other attribute values are adopted from the destination link. You can move the intermediate points to model the connector precisely (see "Editing points in links or connectors" on page 431). The **Connector** window opens. A new connector is assigned the attribute **Number**, which is a number available > 9,999.*
6. Release the keys.

   The connector adopts the following destination link attributes:

   - Behavior type
   - Display type
   - Thickness (3D)
   - Gradient
   - Visualization
   - Show classified values
   - Lane change: No lane change
   - Lane change: Blocked vehicle classes
7. Edit the attributes (see "Attributes of connectors" on page 422).
8. Confirm with **OK**.

*The attributes are saved in the **Links** list.*

You can recalculate the spline and thus adjust the course of the connector, e.g. after you have moved points (see "Recalculating the spline" on page 434).

6.10.2.2 Attributes of connectors

The **Connectors** window opens when you insert a network object and have selected to have the Edit dialog automatically opened after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Connectors list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

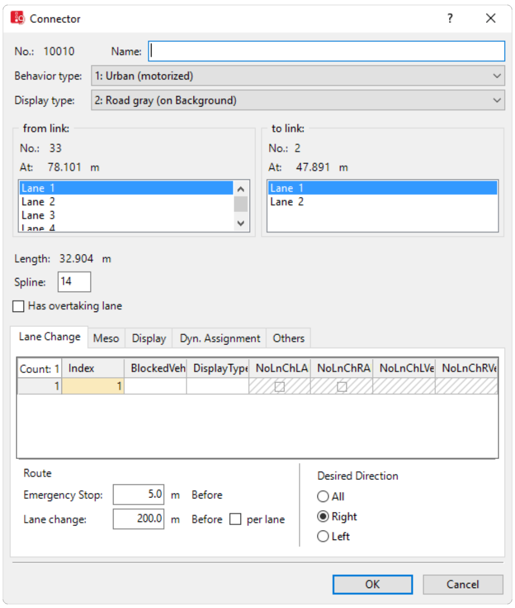
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



Basic attributes of connectors

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number of the connector
Name	Name of the connector
Behavior type	**Link Behavior Type** (**LinkBehavType**): Driving behavior for the connector (see "Defining link behavior types for links and connectors" on page 318) and (see "Defining driving behavior parameter sets" on page 282).
Display Type	Colored display of the connector (see "Defining display types" on page 320)
from link **to link**	Lanes of the exit link (**FromLink**) to lanes of the next link (**ToLink**) between which the connector is inserted. Always select the same number of lanes in both lists. This assignment can also be subsequently edited.
Length	➤ **Length2D** of the connector in the network in meters ➤ **Length3D** accounts for z-offset of the link
Spline	Intermediate points for the graphical representation. Intermediate points are not relevant for the driving behavior of vehicles driving on them. If you enter more intermediate points, you can model the connector more precisely. Two intermediate points are sufficient for straight connectors. Up to 15 points may be useful for longer connectors, for example, for turns. When you e.g. move the start or end point of a connector to a different link or another lane, the intermediate points and the course of the connector are recalculated. With z-Offset, you can enter the altitude above the ground for each intermediate point (see "Editing the z-offset of intermediate points" on page 433).

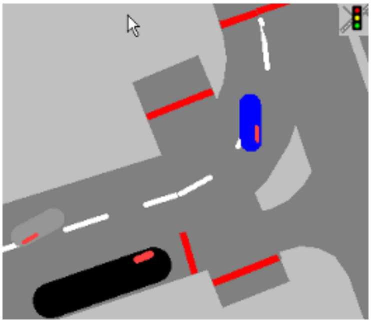
Element	Description
Has passing lane	**HasOvtLn**: The inner lane may only be used for overtaking maneuvers on the oncoming lane. This is only possible on links with at least two lanes (see "Modeling overtaking maneuvers on the lane of oncoming traffic" on page 508). ☑ If this option is selected, for right hand traffic, the outer left lane is displayed as the passing lane, with hatched background. For left hand traffic, the lane on the far right is the passing lane, displayed with hatched background. If a passing lane and regular lane of a link of the opposite direction overlap for long enough, the overlapping area may be used for passing. Only select this attribute for links on which passing is actually allowed in reality. Avoid passing lanes on which overtaking is not possible in reality, e.g. at junctions or in traffic controlled areas. You can also select this attribute for several, successive links that are connected via connectors and have at least two lanes. Vehicles can then use the entire overlapping area for overtaking maneuvers. If the passing lane is closed for a vehicle class, the vehicles of this class cannot use the passing lane for overtaking. You can place other network objects, e.g. data collection points, on passing lanes. Passing lanes are not shown in 3D mode.

Lane Change tab

The tab contains, amongst others, the following attributes:

Element	Description
BlockedVehClasses	**Blocked vehicle classes**: For each lane, you can select the vehicle classes for which the lane is closed (see "Attributes of links" on page 409). If a toll route traverses a connector blocked for all vehicle classes, the corresponding managed lanes routing decision does not apply.
NoLnChLAllVehTypes, **NoLnChRAllVehTypes**	**No lane change left – all vehicle types** and **No lane change right– all vehicle types**: ☑ If this option is selected, vehicles may not change lanes. A prohibition of lane changes is shown in the 2D and 3D mode by means of a solid line.
Display type	Color of lane (see "Defining display types" on page 320)
NoLnChLVehClasses, **NoLnChRVehClasses**	**No lane change left - vehicle classes** and **No lane change right - vehicle classes**: Vehicle classes, whose vehicles must not change from a chosen lane to the adjacent lane in the direction of travel. A prohibition of lane changes is shown in the 2D and 3D mode by means of a solid line.

Element	Description
Emergency Stop	**Emergency stop distance** (**EmergStopDist**): Is used to model the lane change rule of vehicles that follow their route, or in dynamic assignment their path, default value = minimum length = 5 m. If these lanes could not be reached before the connector at the **Emergency Stop** position, the vehicle stops and waits for a sufficiently large gap. The system measures upstream starting from the beginning of the connector. If a vehicle has to make more than one lane change, 5 m per lane is also taken into account in each case. If the current lane has an odd number, 2.5 m are also added to the total length of the emergency stop distance. This prevents a conflict from occurring due to identical positions of 2 vehicles which are set to change lanes on neighboring lanes. Example: A vehicle in lane 1 must change to lane 4 to follow its route or its path. An emergency stop position of 10 m was defined for the subsequent connector. The following relevant emergency stop distance is obtained for lane 1: $10 + 5 + 5 + 2.5 = 22.5\ m$ For lane 2 accordingly: $10 + 5 = 15\ m$ for lane 3: $10 + 2.5 = 12.5\ m$
	The actual emergency stop position is calculated as the difference between: Coordinate of the link where the connector starts minus the emergency stop distance. The result is an integer. Decimal places are not taken into account. Example: If the connector starts at 67.2 m into the link and 12.5 m have been specified for the emergency stop, this results in $67.2 - 12.5 = 54.7$, emergency stop position: 54 m. The emergency stop distance of a connector **A** can reach upstream to another connector **B**. If this does not lead to a lane, from which the connector **A** also departs, the vehicles cannot switch lanes. In this case, Vissim automatically moves the emergency stop position upstream at least 0.1 m until the first link, where the necessary lane change is possible.
Lane change	**Lane change distance** (**LnChgDist**): Is used to model the lane change rule of vehicles that follow their route, or in dynamic assignment their path. Distance before the connector from which those vehicles, whose route or path leads across this connector, try to choose the lane in which they reach the connector without changing lanes. Standard value: 200 m, minimum value 10 m. The value must be $>= emergency\ stop + 5\ m$.

Element	Description
Per lane	**Lane change distance is per lane** (**FsWechsDististProFs**) ☑ If this option is selected, the entered **Lane change** attribute value is multiplied by the number of lane changes which a vehicle requires to reach the connector. Example: Before reaching a connector with a lane change distance of 200 m per lane, which starts from lane 1 only, a vehicle in lane 3 already starts to look for a gap to change lanes 400 m before the connector starts.
Desired Direction	**Direction** (**Direction**): Shows direction-indicator blinking signal on the vehicle during a simulation run, if the route of the vehicle leads via this connector and there is routing information available for all vehicles traversing this route:  - **All**: Vehicles do not use turn signal (straight). - **Right**: Vehicles use right turn signal. - **Left**: Vehicles use left turn signal.
	Vehicles without a desired direction and vehicle route always drive on the next connector with the criterion **All**. If no such connector exists at the end of the link, these vehicles are removed from the Vissim network. The desired direction is displayed in the following order of priority: 1. Current lane change 2. Desired lane change 3. Desired direction of current link 4. Desired direction of the next route link with the desired direction **right** or **left**, if the vehicle is located within the value range defined in the attribute **Lane change distance** (**Lane change** box). 5. Turn signal direction through external driver model in the *set driver model.DLL* file

Meso tab

The attributes of the connector for mesoscopic simulation corresponds to the attributes of links (see "Attributes of links" on page 409).

Display tab

Element	Description
Thickness (3D)	**Thickness** for the display of the connector in 3D mode.
Visualization	☑ If this option is checked, the vehicles are indicated in the 2D mode. ☐ If this option is not checked, no vehicles are indicated in the 2D mode. With this, you can indicate underpasses or tunnel sections. This option applies for the entire connector. Therefore you must define a separate connector for each underpass or for each tunnel. Note: Do not use this option in 3D mode, but rather model the height details of the connectors correctly!
Show classified values	**ShowClsfValues**: ☑ Select this option to show classified values (not to show the display type selected). To show classified values, in the graphic parameters for links, select a color scheme and an attribute (see "Assigning a color to links based on aggregated parameters" on page 179).
Label	☐ If this option is not selected, the labeling of the connector is not displayed.

Dyn. Assignment tab

Only for the add-on module Dynamic Assignment (see "Using the dynamic assignment add-on module" on page 692).

Element	Description
Connector closed to	**Blocked vehicle classes for dynamic assignment** (**BlockedVehClassesDynAssign**): Via this list, you can model a multi-modal network for dynamic assignment by closing the connector for one or more vehicle classes. Thus the connector for the route selection of blocked vehicle classes is not available.
Cost	distance-dependent **costs per km** (**CostPerKm**)
Surcharge 1, Surcharge 2	one-time surcharges that are taken into account for path evaluation. With this data, in the dynamic assignment the costs for the vehicles which travel on this connector are determined.

Other tab

The tab contains, amongst others, the following attributes:

Element	Description
Gradient	Uphill and downhill slopes of the connector in percent. Downhill slopes have a negative value. The value impacts the driving behavior via the maximum acceleration and maximum deceleration on a connector. - by -0.1 m/s² per gradient percent incline. The maximum accelerating power decreases when the deceleration power increases. - by 0.1 m/s² per gradient percent downgrade. The accelerating power increases when the deceleration power decreases. Per default, uphill and downhill slopes in 3D mode do not affect the display (z-coordinates) of connectors. You can edit z-coordinates via the **z-Offset** attribute of the connector. If the z-coordinates in your Vissim network have been entered correctly, you can have Vissim calculate uphill and downhill slopes. In this case, the option **Use gradient from z coordinates** must be selected (see "Selecting network settings for vehicle behavior" on page 203).
Link evaluation	**Link evaluation active** (**LinkEvalAct**): Select this option to record link data for the link evaluation. This option is only relevant, if you selected the attribute **Show classified values**. **Segment length**: Segment length for the link evaluation, default value 10.0 m (see "Showing data from links in lists" on page 1103)
Overtaking speed factor:	**OvtSpeedFact**: Factor by which the vehicle wants to overtake, increasing its desired speed. Default 1.3

The following attributes are only relevant for modeling overtaking maneuvers on the oncoming lane:

Element	Description
Look ahead distance	**Look ahead distance for overtaking** (**LookAheadDistOvt**): Distance that the overtaking vehicle can view on this link, upstream of the oncoming lane. At this distance oncoming traffic is perceived by drivers. At the end of this distance a virtual, oncoming vehicle is assumed, if on this connector there is a vehicle input or a PT line. The shorter the look ahead distance for overtaking is, the smaller the likelihood of being able to overtake. Default 250 m.
Assumed speed of oncoming traffic	**AssumSpeedOncom**: Assumed speed of oncoming traffic for vehicle that wants to overtake. The higher the assumed speed of oncoming traffic is, the smaller the possibility of overtaking. Default value 60 km/h.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119). They correspond to those of links (see "Attributes of links" on page 409).

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.10.2.3 Assigning lanes between links

In the attributes and options of a connector, you can assign the lanes from the link at which the connector begins to the lanes of the link at which the connector ends.

The **Connector** window opens automatically when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Connectors list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359)
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see "Using coupled lists" on page 119).

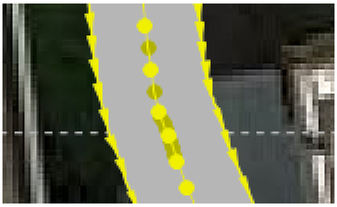
1. In the **from link** field, click on the desired lanes of the link in which the connector begins.
2. If you wish to select further lanes, hold down the SHIFT key and click on the other desired lanes.
3. In the **to link** field, click on the desired lanes of the link in which the connector ends.

   *The selected lanes in the **from link** and the **to link** field are highlighted.*
4. Confirm with **OK**.

6.10.3 Editing points in links or connectors

You can insert, move and delete points in links and connectors. This enables you to model the course of the road.

If points lie very close to each other, they are displayed darker depending on the Zoom level set in the Network editor.



You cannot click or edit the darker points. To edit these points, zoom in further until they are shown in a lighter color again. If points lie so close to each other that you cannot edit a point using the current Zoom level, all points are hidden.

If you move the mouse pointer to start point, end point or other point of a link or connector, this point is shown enlarged. This will make it easier for you to click it.

You can also recalculate the course of a link or connector (see "Recalculating the spline" on page 434).

6.10.3.1 Adding points to a link

You can insert points into a link to model a curve. You may define the z offset for each point (see "Editing the z-offset of intermediate points" on page 433).

1. On the Network objects toolbar, select the network object type **Links**.
2. In the Network editor, zoom into the link of your choice.
3. In the Network editor, right click the desired position in the link.

   The movement direction is shown by arrows at the edges of the link.
4. From the context menu, choose **Add Point**.

Per default, the link is inserted as a yellow intermediate point. If you zoom out very far, Vissim arrows and intermediate points are no longer fully displayed.

If within a link, a z-offset (start), z-offset (end), and/or intermediate points with a z-offset define of an upward or downward gradient, for a new intermediate point, a value for the z-offset is interpolated.

6.10.3.2 Generating a spline

In the Network editor, on a link or connector, you can select spline points, insert spline points and convert the link, connector or a section into a spline. This allows you to create a more harmonious curve. After further changes on the curve, select **Recalculate Spline** to re-establish a more harmonious curve (see "Recalculating the spline" on page 434).

Creating a spline based on the entire link

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, right-click desired link or connector.
3. From the shortcut menu, choose **Generate Spline**.

The spline points are inserted into the link or connector. If the link contains multiple spline points, several new spline points are inserted in between the existing ones. You can move the spline points to model the course of the link.

Selecting spline points, inserting new spline points and generating a spline

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, click the link or connector of your choice.
3. Press the ALT key and hold down the left mouse button. Drag the mouse pointer to the spline point up to which you want to insert additional spline points or until the end point of the link or the connector.

   The spline points are highlighted. A highlighted line connects the spline points.
4. Release the keys.

   *The **Convert section to spline** window opens.*
5. Make the desired changes:

Element	Description
Keep current intermediate points	☑ Select this option to insert intermediate points in addition to the the existing points.
Number of interm. points (per segment)	Enter the number of intermediate points.

The intermediate points are inserted into the link or connector. If you have highlighted several successive sections of a link, new points are inserted into each section. You can move the spline points to model the course of the link.

Only Generate Spline between Adjacent Points

1. On the Network objects toolbar, click **Links**.

2. In the Network editor, click the link or connector of your choice.
3. In the link, right click the desired intermediate point.
4. From the shortcut menu, choose **Generate Spline For Adjacent Sections**.

   *The **Convert section to spline** window opens.*
5. Make the desired changes:

Element	Description
Keep current intermediate points	☑ Select this option to insert intermediate points in addition to the the existing points.
Number of interm. points (per segment)	Enter the number of intermediate points.

The intermediate or spline points are inserted into the link or connector between the point selected and its adjacent point. You can move the spline points to model the course of the link.

6.10.3.3 Moving points

In the network editor, you can move the points on a link or connector in order to model the course of a road.

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, click on the link or the connector.
3. Click on the point and hold down the mouse button.
4. Move the mouse pointer to the desired position.
5. Release the mouse button.

You can create a harmonious curve again for the link or connector (see "Recalculating the spline" on page 434).

6.10.3.4 Editing the z-offset of intermediate points

With z offset, you can enter the altitude above the ground for each intermediate point of a link or connector. For links you can also do so for the start and end points.

1. On the Network objects toolbar, select the network object type **Links**.
2. In the Network editor, zoom into the intermediate point of your choice.
3. In the Network editor, right click the desired intermediate point.
4. From the shortcut menu, choose **Edit Z-Offset Of Spline Point**.

*The window **Spline Point** opens.*

5. Into the **z-Offset** box, enter the desired altitude above the ground.
6. Confirm with **OK**.

6.10.3.5 Recalculating the spline

In the network editor you can recalculate the course of a road or a connector on the basis of the points, for example if you have inserted new points and wish to adjust the course of the road. You can select whether the position and the height are to be recalculated. Vissim calculates the distance or connector for a harmonious course with a spline.

1. On the Network objects toolbar, click **Links**.
2. In the network editor, right click on the desired link or connector.
3. To select further links or connectors, hold down the CTRL key and click the other links or connectors of your choice.
4. Choose the desired entry from the context menu:

Element	Description
Recalculate Spline	Recalculates the x, y, z coordinates for the spline
Recalculate Spline (X/Y Only)	Recalculates the x, y coordinates for the spline
Recalculate Spline (Height Only)	Recalculates the z coordinates for the spline

The course of the road is adjusted.

Tip: Alternatively, you use **Recalculate spline** to select the desired links or connectors and then press CTRL+R.

6.10.3.6 Defining the height of a spline point

You can define the z coordinate for each point in a link or connector.

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, click on the link or the connector.
3. Double-click on the point.

   *The window **Spline Point** opens.*
4. In the **Z-offset** box, enter the desired value in meters.
5. Confirm with **OK**.

6.10.3.7 Deleting points

You can delete a single point or several points.

1. On the Network objects toolbar, click **Links**.
2. In the Network editor, click on the link or the connector.
3. Click on the point which you wish to delete and keep the mouse button pressed.
4. Move the mouse pointer to the point which you wish to retain as the last point.
5. Release the mouse button.

The points are deleted. The course of the road is adjusted.

6.10.4 Changing the desired speed

You can change the desired speed (DesSpeed), using the following network objects:

- Reduced speed areas for a temporary change (see "Using reduced speed areas to modify desired speed" on page 435).
- Desired speed decisions for a permanent change, for example on motorways, for traffic signs which limit the speed, or at the entry and exit of a town. Desired decisions are placed at the position where the change is to begin. For multi-lane links, position a desired speed decision on each lane and define the attributes.

  With desired speed decisions, a vehicle only reduces its speed once it has entered a desired speed decisions section. The vehicle is then assigned a new desired speed and changes its speed accordingly. This is when its desired acceleration is used.

When entering the network, each vehicle is assigned a fractile value for speed distribution. This value remains unchanged during the entire simulation time. If this value is 40 %, the vehicle is always assigned a 40 percentile of the desired speed distribution for desired speed decisions. If the fractile is 100 %, the vehicle is always assigned the highest distributed speed.

Tip: You can show the numbers of the speed distributions assigned for desired speed decisions and reduced speed areas via the graphic parameters **Label visibility** and **Label attribute DesSpeedDistr<No.>**.

6.10.4.1 Using reduced speed areas to modify desired speed

With reduced speed areas, vehicles automatically decelerate before entering the area and enter it at a reduced speed. After leaving the reduced speed area, the vehicle automatically accelerates until it reaches its desired speed again.

When entering the network, each vehicle is assigned a fractile value for speed distribution. This value remains unchanged during the entire simulation time.

As reduced speed areas are mainly used to model curves, reduced speed areas are often used for connectors between two links.

For multi-lane links, position a reduced speed area on each lane and specify it.

A reduced speed area may extend beyond several links or connectors. To define a larger reduced speed area, specify multiple consecutive reduced speed areas that all lie on one link.

Note: If two reduced speed areas with identical attributes have been defined on two consecutive links, the vehicles affected will keep the new speed, even when driving between the two areas.

When a vehicle approaches a reduced speed area, it is assigned a new desired speed. The new speed has been specified in the desired speed distribution of the respective vehicle class.

A reduced speed area only applies for vehicles of the selected vehicle classes.

Note: Do not select a vehicle class for a reduced speed area whose vehicles you want to keep a higher or lower speed when they pass the reduced speed area.

After leaving the reduced speed area, the vehicle automatically reaches its desired speed again. Acceleration at the end of the reduced speed area is achieved through the technical and driving options of the driver vehicle unit.

Assigning a reduced speed area a higher speed

You can assign a vehicle a higher than its current speed when it is approaching a reduced speed area.

- A vehicle driving faster than the vehicle class specific speed for a reduced speed area will reduce its speed when approaching the reduced speed area. So when the vehicle reaches it, it will be driving at the lower, vehicle class specific speed of the reduced speed area. Braking starts, as soon as required to reach the reduced speed. The braking deceleration value is positive.
- A vehicle driving slower than the vehicle class specific speed of the reduced speed area will only change its speed once it has reached the reduced speed area. A reduced speed area is not meant to cause a slower vehicle to accelerate to a certain speed until it reaches the reduced speed area. For slower vehicles, a reduced speed area has the effect of a higher desired speed decision and only becomes effective once the vehicle has entered the reduced speed area.

Influence of reduced speed area on delay measurement

Preceding vehicles or different network objects located further downstream might lead to a vehicle to go below its desired speed. Reduced speed areas are network objects. Going below the desired speed results in a time delay that is added to the loss time. This accumulated loss time also includes the time delay caused during braking before entering the reduced speed area. The time delay caused through the reduced speed area is deducted from the accumulated loss time during the time step the vehicle reaches the reduced speed area. So during this time step, the accumulated loss time is reduced (see "Showing delay measurements in lists" on page 1107).

6.10.4.2 Defining reduced speed areas

You can add reduced speed areas in links or connectors. At least one desired speed distribution must be defined for the definition of a reduced speed area (see "Using desired speed distributions" on page 237).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Reduced Speed Areas**.
2. Hold down the CTRL key and right-click on the desired position of the reduced speed area in the link or the connector.

   *A colored bar is added. The **Reduced Speed Area** window opens. You can define the desired speed distribution and the deceleration value for each vehicle class, whose speed should be changed.*
3. Edit the attributes (see "Attributes of reduced speed areas" on page 437).
4. Confirm with **OK**.

*The attributes are saved in the **Reduced Speed Areas** list.*

6.10.4.3 Attributes of reduced speed areas

The **Reduced Speed Area** window opens automatically when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Reduced Speed Areas list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and show-

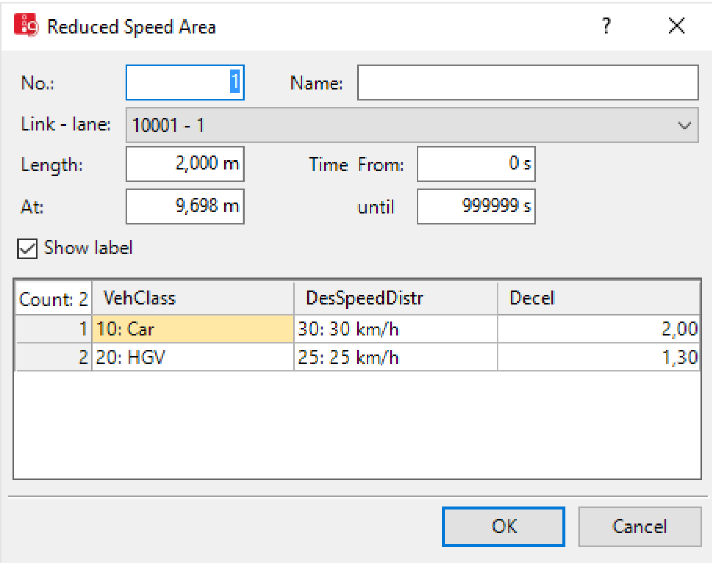
ing them in a list" on page 359).

- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	**Description**
No.	Unique number of the reduced speed area
Name	Designation of the reduced speed area
Lane	**Ln**: Number (**Index**) of lane on which the reduced speed area is placed.
Length	Length of the reduced speed area in the network in meters
At	**Position** (**Pos**): Distance from start of the link or connector

Element	Description
From time, To time	**TimeFrom**, **TimeTo**: Time interval in simulation seconds for which the reduced speed area is active
Label	☐ If the option is not selected, the label for the Reduced speed area is not displayed, even if the label for all reduced speed areas is selected.

The list contains, amongst others, the following attributes:

Element	Description
VehClass	**Vehicle Classes** for which the reduced speed area applies
DesSpeedDistr	**Desired Speed Distribution**: Speed in the reduced speed area
Decel	The maximum **deceleration** with which faster vehicles decelerate when they approach the reduced speed area. The lower this value is, the further away from the reduced speed area the vehicle begins to reduce its speed.

Notes:

- A reduced speed area only applies to vehicles which enter into the reduced speed area.
- The reduced speed areas may not overlap for a particular vehicle class. If reduced speed areas overlap, the driving behavior of the vehicle class is not defined.
- Do not insert a stop line for a signal control, a priority rule, or a stop sign, within a reduced speed area. Otherwise, not all of the vehicles may recognize the stop line. Always insert the start of reduced speed areas behind the stop line.
- The combination of vehicle classes, speed distribution and acceleration of the last reduced speed area which was edited is used as the preset value for the new reduced speed area.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Speed reductions**.

   Attributes of slow driving are displayed: Slow driving defined for different vehicle types in this reduced speed area.

The attributes are described further above.

3. Enter the desired data.

The data is allocated.

6.10.4.4 Using desired speed to modify desired speed decisions

You can permanently change desired speeds with desired speed decisions, for example on motorways, for traffic signs which limit the speed, or at the entry and exit of a town. Desired decisions are placed at the position in the network where the change is to begin. For multi-lane links, position a desired speed decision on each lane and define its attributes.

With desired speed decisions, a vehicle only reduces its speed once it has entered a desired speed decisions section. The vehicle is then assigned a new desired speed and changes its speed accordingly. To do so, it uses the desired acceleration or desired deceleration.

When entering the network, each vehicle is assigned a fractile value for speed distribution. This value remains unchanged during the entire simulation time. If this value is 40 %, the vehicle is always assigned a 40 percentile of the desired speed distribution for desired speed decisions. If the fractile is 100 %, the vehicle is always assigned the highest distributed speed.

Delay measurement for the desired speed decision

After the vehicle has traversed the desired speed decision, Vissim calculates a theoretical speed over several time steps until this speed is identical with the new desired speed of the vehicle. This theoretical speed starts at the original desired speed and ends at the new desired speed. During the time steps the vehicle needs to reach its new, desired speed, Vissim compares the current speed of the vehicle with the current theoretical speed calculated - not with the new desired speed the vehicle is supposed to reach - in order to create a basis for loss time calculation.

Note: You define speed limits over a short distance with reduced speed areas, e.g. for a curve or tuns at intersections (see "Using reduced speed areas to modify desired speed" on page 435). If you use a desired speed decision for this, you must define a second desired speed decision immediately after this, so that the vehicles are given their original desired speed.

6.10.4.5 Defining desired speed decisions

You can add Desired Speed Decisions in links or connectors.

At least one desired speed distribution must be defined for the definition of a Desired Speed Decision (see "Using desired speed distributions" on page 237).

Vehicles change their speed exactly at the position of the desired speed decision. Thus, the acceleration phase or the deceleration phase is only after covering this position. Depending on the current speed, the new desired speed is reached only at a certain distance.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Desired Speed Decisions**.
2. Hold down the CTRL key and right-click on the desired position of the desired speed decision in the link or the connector.

   *A colored bar is added. The **Add desired speed decision** window opens. You can define the distribution of the new desired speed for the relevant vehicle classes.*
3. Edit the attributes (see "Attributes of desired speed decisions" on page 441).
4. Confirm with **OK**.

*The attributes are saved in the **Desired Speed Decisions** list.*

6.10.4.6 Attributes of desired speed decisions

The **Desired Speed Decision** window opens automatically when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Desired Speed Decisions list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

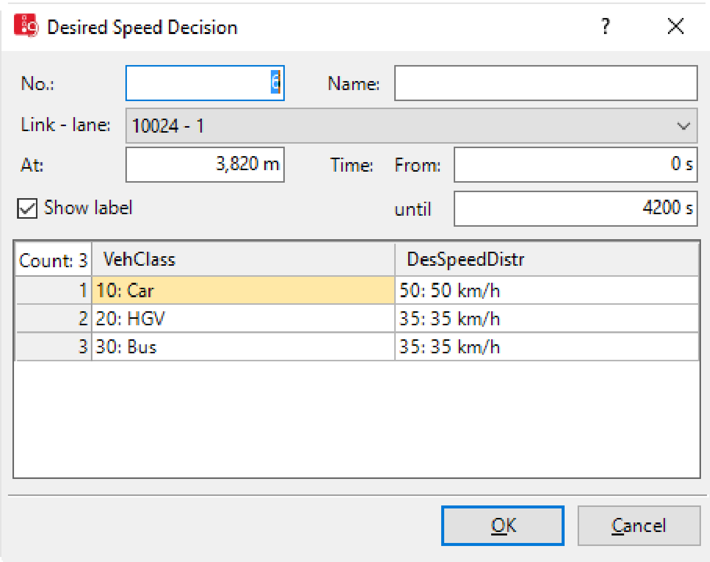
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number of the desired speed decision
Name	Name of desired speed decision
Lanes	**Ln**: Number of lane
At	**Position** (**Pos**): Distance from start of the link or connector
From time, To time	Time interval in simulation seconds for which the Desired speed decision is active
Label	If the option is not selected, the label for the Desired speed decision is not displayed, even if the label for all Desired speed decisions is selected.

The list in the tab contains, amongst others, the following attributes:

Element	Description
Vehicle class	**Vehicle classes** for which the desired speed decision applies
DesSpeedDistr	**Desired Speed Distribution**: Speed after the desired speed decision

Notes:

- The desired speed distribution does not change for vehicles in vehicle classes which are not selected.
- The combination of vehicle classes and speed distribution of the last desired speed decision which was edited is used as the preset value for the new desired speed decision.
- You define speed limits over a short distance with reduced speed areas, e.g. for a curve or tuns at intersections (see "Using reduced speed areas to modify desired speed" on page 435). If you use a desired speed decision for this, you must define a second desired speed decision immediately after this, so that the vehicles are given their original desired speed.
- If a speed change is only to apply for turning vehicles, define a reduced speed area directly on the turning link.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Desired Speed Distributions (DesSpeedDistr)**.

The attributes are described further above.

3. On the list toolbar, in the **Relations** list, click the desired entry.

4. Enter the desired data.

The data is allocated.

6.10.5 Modeling pavement markings

You can add pavement markings and therefore model crosswalk and arrow markers. With arrow markers you indicate the permissible movement directions in the lanes. Lanes can be marked with diamond shapes. Especially in the USA, diamond shapes are used to show lanes which may only be used by vehicles with a minimum number of occupants.

Note: Pavement markings do not affect the driving behavior and are not used to model turn relations. Turning movements are modeled with vehicle routes (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459).

6.10.5.1 Defining pavement markings

You enter pavement markings on links in Network Editor.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click > **Pavement Markings**.
2. Hold down the CTRL key and right-click the desired position of the Pavement Markings in the link or connector.
3. Release the mouse button.

   *An arrow marker is added. The **Pavement Marking** window opens.*
4. Edit the attributes of the pavement marking (see "Attributes of pavement markings" on page 444).
5. Confirm with **OK**.

*The attributes are saved in the list of **Pavement Markings**.*

6.10.5.2 Attributes of pavement markings

The **Pavement Marking** window opens automatically when you insert a network object and have selected to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Pavement Markings list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

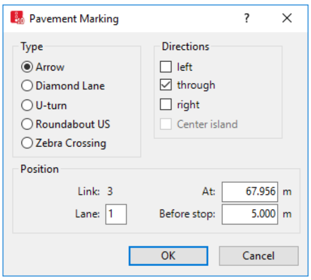
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Type</td><td>Defines the geometric symbol shown in Vissim the network:
Arrow direction: Can be Left, Straight, Right or a combination of the latter options.
Diamond: In the USA and Canada identifies lane for high occupancy vehicles (carpool lane)
U-rurn: Vissim displays a clockwise or anticlockwise arrow. This depends on the settings made under Network Settings > Vehicle Behavior > Traffic Regulations and the option selected Right-hand traffic or Left-hand traffic.
Roundabout US: The arrow direction can be Left, Straight, Right, Center island or a combination of these options. Center island must be combined with at least one other arrow option.
Zebra Crossing is used to mark pedestrian links that are defined as pedestrian crossings. The option is used to indicate the width of the link which is crossed. All traffic information must be defined for the links (see "Modeling links as pedestrian areas" on page 922).</td></tr>
<tr><td>Directions</td><td>Arrow directions (ArrowDir): This is only relevant for the Arrow type: This defines the geometric symbol shown in the Vissim network. If you select several directions, these are combined in the arrow.
Links
Straight
Right
Central Island: Only relevant for US roundabout. Must be combined with at least one other arrow option.</td></tr>
<tr><td>Position</td><td>Pos: Position in the link</td></tr>
<tr><td>Before stop:</td><td>Distance to the next downstream signal head on the same link or connector</td></tr>
</table>

6.10.6 Defining data collection points

Data collection points and data collection measurements based on them are similar to induction loops that are attached to roadway tracks for the recording of traffic volume.

In Vissim data collection points can be used, in particular, for monitoring the simulated number of vehicles. This data can be recorded in evaluations for specific vehicle classes (see "Evaluating data collection measurements" on page 1093).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Data Collection Points**.
2. Hold down the CTRL key and right-click on the desired position of the data collection point in the link or the connector.

   *A colored bar is added. The **Data Collection Points** list opens.*
3. Edit the attributes:

Element	Description
No	Unique identification of the data collection points
Name	Data collection point name
Lane	Lane
Pos	Position on the link

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Data collection measurements (see "Defining a data collection measurement in lists" on page 1011), (see "Generating data collection measurements in lists" on page 1011)

2. Edit the desired entries.

The data is allocated.

6.10.7 Defining vehicle travel time measurement

A vehicle travel time measurement consists of a From Section and a To Section. The mean travel time from traversing the From Section up to the traversing of the To Section is calculated, including the waiting time and/or stop time on all lanes (see "Evaluating vehicle travel time measurements" on page 1096).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network object toolbar, click **Vehicle Travel Times**.
2. Using the mouse pointer, point to the desired position of the From Section of the Vehicle travel time measurement in the link.
3. Hold down the CTRL key and right-click on this position.

   A colored bar is added.
4. Using the mouse pointer, point to the desired position of the To Section in the link.
5. Click.

   *A colored bar is added. The **Travel Time Measurement** window opens.*
6. Edit the attributes (see "Attributes of vehicle travel time measurement" on page 448):
7. Confirm with **OK**.

6.10.8 Attributes of vehicle travel time measurement

The **Travel Time Measurement** window opens automatically when you insert a network object and have selected to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Vehicle Travel Time Measurements list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Element	Description
No	Number of travel time measurement. You can enter a different number. Use a continuous numbering system for the Vissim network. This simplifies the evaluations.
Name	Name of travel time measurement
start section	- **Start link** (**StartLink**): Number of the link of From Section - for **Start position** (**StartPos**): Distance between From Section and beginning of link or connector - **Label**: If this option is not selected, the label for an individual vehicle travel time measurement is hidden if the label for all vehicle travel time measurements is selected.

Element	Description
destination section	• **End link** (**EndLink**: Number of link of destination section • for **End position** (**EndPos**): Distance between destination section and beginning of link or connector • **Label**: ☐ If this option is not selected, the label for an individual vehicle travel time measurement is hidden if the label for all vehicle travel time measurements is selected.
Distance	**Dist**: Length of the Vissim determined shortcut (shortest possible path) from the From Section to the To Section. If you change the value, the position of the To Section is automatically adjusted. If no distance is entered, no continuous link sequence exists between both markers. An important connector is probably missing or one of the sections does not lie on the desired link, rather, for example, in the opposite direction. Define a distance so large that the vehicles cannot traverse the From section and the To Section within the same time step. Vehicles that do not traverse the From Section and the To Section within the same time step are not accounted for in travel time management.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Delay measurements**.

   The delay measurements generated are displayed. All delay measurements can be newly generated (see "Defining delay measurement in lists" on page 1012), (see "Generating delay measurements in lists" on page 1013).

6.10.9 Modeling queue counters

Queue lengths can be determined with queue counters at any point in the Vissim network and evaluated for any time interval. This is facilitated by stop lines of signalized intersections on links and connectors. The following values are output (see "Showing results of queue counters in lists" on page 1105):

- Maximum queue length
- Average queue length
- Number of queue stops

Queues are measured from the upstream position of the queue counter up to the last vehicle that has entered the queue conditions. You define the queue conditions when configuring the evaluation of the queue counters. To evaluate queue lengths, you can take the adjacent lanes of previous links into account (see "Showing results of queue counters in lists" on page 1105).

If a queue has multiple ends, queue lengths are recorded at every time step on all arms of the queue which are upstream to the queue counter. The queue counter then returns the average queue length of the arm, for which the longest queue was measured. At this time step

The length of a queue is detected as long as the queue has been eliminated completely, even if vehicles have resumed driving between the queue counter and the queue end and no longer meet the queue condition. The queue length is measured up to the last vehicle remaining in the queue that meets the condition until the queue condition is no longer met by any of the vehicles.

Queue lengths are output in terms of units of length, not in terms of number of vehicles.

Queues are always tracked up to their original end, even if the first vehicles are no longer in queue.

The maximum queue length is as long as the distance to the next queue counter upstream or as long as specified in the queue definition for the attribute **Max. length**. For a queue counter that is automatically generated by Vissimfor node evaluation, the maximum queue length is the distance to the next queue counter, upstream in the node.

6.10.9.1 Defining queue counters

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Queue Counters**.
2. Hold down the CTRL key and right-click on the desired position of the queue counter in the link or the connector.

   *A colored bar is added. The **Queue Counters** list opens.*
3. Edit the attributes (see "Attributes of queue counters" on page 451).

6.10.9.2 Attributes of queue counters

The **Queue Counter** window opens when you insert a network object and have selected to automatically have a list opened after object creation (see "Right-click behavior and action after creating an object" on page 152).

1. From the **Lists** menu, choose **Measurements** > **Queue counters**.

   *The **Queue Counters** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Tip: To open coupled lists for a network object type, from the **Lists** menu, choose > **<Name network object type>**.

The list on the left may include the following attributes:

Element	Description
No.	Unique identification of the queue counter
Name	Name of queue counter
Link	Name of the link, on which the queue counter is defined
Pos	Position on the link

6.11 Modeling vehicular traffic

Vehicular traffic can be modeled with the following variants:

- **Static routes**: You define the routing decisions and therefore specify the paths which the vehicles travel in the network (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459), (see "Modeling parking lots" on page 493). After this you define the vehicle inputs. Vehicle inputs control the number of vehicles which are fed into the Vissim network. Then run the simulation.
- You use source-destination matrices with the add-on **Dynamic Assignment** module. For dynamic assignment, static routes or vehicle inputs are not required (see "Using the dynamic assignment add-on module" on page 692).

6.11.1 Modeling vehicle compositions

You must define vehicle compositions from the vehicle type, so that the vehicle can be generated. In general, there will be multiple vehicle compositions, e.g. **cars only**, **cars with 5% HGV proportion** or vehicle compositions for **pedestrians**, for which you specify relative volumes for the vehicle types **male** and **female**.

For vehicles, which only drive in PT routes, no vehicle compositions are necessary, for example, for public transportation by trams and city buses (see "Modeling short-range public transportation" on page 511).

A vehicle composition contains multiple vehicle types. You allocate a relative volume and desired speed distribution to each vehicle type (see "Using desired speed distributions" on page 237).

For pedestrian flows you define the pedestrian compositions. Do not define vehicle compositions for pedestrian flows, as these would then be bound to links and the model of the flow of traffic according to Wiedemann (see "Driving states in the traffic flow model according to Wiedemann" on page 285).

6.11.1.1 Defining vehicle compositions

You can define new vehicle compositions, assign the desired vehicle types and enter attribute values.

1. Select **Private Transport** > **Vehicle Compositions** in the **Lists** menu.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. In the list on the left, enter a number and name (see "Attributes of vehicle compositions" on page 453).

   In the next steps, assign the desired vehicle types to the chosen vehicle compositions. For each vehicle type you must add a row to the right hand list and you can define the attribute values.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

4. In the list on right, on the toolbar, click the **Add** button .
5. Select the desired entry.
6. Enter the desired attribute values in the right-hand list (see "Attributes of vehicle compositions" on page 453).

The data is allocated.

6.11.1.2 Attributes of vehicle compositions

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Tip: To open coupled lists for a network object type, from the **Lists** menu, choose > **<Name network object type>**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Unique number of the vehicle composition
Name	Name of vehicle composition
RelFlow	Relative volumes: Taken from the right list

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Relative flows**

Element	Description
Vehicle type	In the list box, select **Vehicle type**.
DesSpeedDistr	Reference to the **Desired speed distribution** for the vehicle type
RelFlow	**Relative flow**: Share of the vehicle type in the composition. The sum of the relative volumes must not necessarily yield 1.0. As in the route shares, the sum of all relative volumes always forms the basis of 100%; on the basis of Vissim the absolute shares are calculated.

6.11.2 Modeling vehicle inputs for private transportation

In the network, you can define time-dependent vehicle inputs for all vehicle types. For this, at least one vehicle composition must be defined (see "Modeling vehicle compositions" on page 452). Vehicle inputs may be positioned on links only. A vehicle input cannot be placed on a single lane of a link.

Time intervals for vehicle inputs

You can create time intervals for vehicle inputs and thus determine the volume for each time interval in a pedestrian input. You enter the volume for a link in vehicles per hour. When doing so, you can choose to use the exact number of vehicles or have Vissim select a number stochastically based on the volume entered. You must always enter the number of vehicles per hour, even if the time interval is shorter or does not end at a full hour, e.g. after 3.5 hours.

Vehicle entry times

The time when the vehicles enter a link in the Vissim network is defined by Vissim stochastically: An average time gap between two vehicles results from the hourly volume. This average time gap is used as an average value of a negative exponential distribution. Vissim obtains the time gaps from this distribution which relates to a Poisson distribution. In real life scenarios, the entry time may be subject to greater variability than in Vissim on the basis of the Poisson distribution.

If the vehicle input specified for the network during a simulation time interval is exceed by more than two vehicles, a corresponding message is displayed.

- The message and time interval per vehicle input are saved to the **.err file*.
- When you exit the simulation run, a window opens displaying the message. In this window, you can open the **Messages** window. It shows this and other messages, if there are any.
- The **.err* file does not contain any messages if vehicles are not input because the vehicle input interval is longer than the simulation time. The maximum possible traffic volume depends on the speed and the driving behavior parameters which have been set.

Lane selection when entering a vehicle into the Vissim network

The vehicle must be assigned to a vehicle type that is allowed in the lane of the link on which the vehicle input is placed (see "Attributes of links" on page 409).

If a link has multiple lanes in which the vehicle can be entered, Vissim will introduce the vehicle in the lane that provides the longest collision time for the vehicle. That is the lane in which the vehicle can drive the longest without reaching a preceding vehicle or network object that has an impact on its desired speed, e.g. an SC, a priority rule or a conflict area.

Vehicle speed used in network

When calculating the speed of a vehicle within the Vissim network, e.g. in an input flow or during dynamic assignment, on a parking lot of a zone connector, Vissim accounts for the distance between the vehicle and its preceding vehicle as well as for the maximum look ahead distance. The look ahead distance on the respective link applies for the driving behavior of the vehicle type the vehicle was assigned.

- If the distance is greater than the maximum look ahead distance, the vehicle moves at its desired speed.
- If the distance is less than the maximum look ahead distance, but greater than the safety distance, the following applies: *v = DesSpeed - (DesSpeed - vPrecedingVehicle) * (1 - dx / maximum look ahead distance)*.
- If the distance is less than the safety distance, the vehicle is not used.

Demand in origin-destination matrices for dynamic assignment

For dynamic assignment, you do not need to define any vehicle inputs, as the demand is contained in the origin-destination matrices (see "Using the dynamic assignment add-on module" on page 692).

6.11.2.1 Defining vehicle inputs

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. Make sure that you have defined the time intervals for which you want to enter volumes (see "Defining time intervals for a network object type" on page 326).
2. On the Network objects toolbar, click **Vehicle Inputs**.
3. Hold down the CTRL key and right-click the desired link or connector.

   *The **Vehicle input** window opens.*

   If there is no volume yet, only the column titles are displayed.
4. Right-click in the list.
5. From the shortcut menu, choose **Add**.

   *A colored bar is shown at the start of the link. The **Vehicle inputs** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152).*

   A new row with default data is inserted.

   *The default value for **Volume** (**Volume**) is 0. The default value for the **Vehicle composition** (**VehComp**) is **Default**. Both default values are applied for all time intervals defined for vehicle inputs.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

6. In the list on the left, enter the desired attribute values (see "Attributes of vehicle inputs" on page 457).

   In the next steps you assign the desired network objects to the selected vehicle input.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Link**: Attributes of the link (see "Attributes of links" on page 409)
- **Vehicle volumes by time interval**: if you have entered a volume or selected a vehicle composition and then select **Vehicle volumes by time interval**, the attributes of the

vehicle volumes for the first time interval are displayed (see "Defining time intervals for a network object type" on page 326).

7. On the list toolbar, in the **Relations** list, click the desired entry.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

8. Enter the desired attribute values in the right-hand list (see "Attributes of vehicle inputs" on page 457).

9. In the right-hand list, you can add more entries for additional time intervals for which you want to define vehicle volumes for the selected vehicle input.

The data is allocated.

6.11.2.2 Attributes of vehicle inputs

The **Vehicle Inputs** list opens automatically when you insert a network object and have selected to automatically have a listed opened after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Vehicle Inputs list is opened.

1. Select **Private transport** > **Inputs** in the **Lists** menu.

   *The **Vehicle inputs** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Tip: To open coupled lists for a network object type, from the **Lists** menu, choose > **<Name network object type>**.

The list on the left may include the following attributes:

Column	Description
No	Unique number of the vehicle input
Name	Name of vehicle input
Link	Name of the link to which the vehicle input was added
Volume	**Volume**: Number of vehicles per hour - not per time interval
VehComp	**Vehicle composition** per hour - not per time interval (see "Modeling vehicle compositions" on page 452)

2. If you want to change the time intervals for a vehicle input, right-click the desired entry in the **Vehicle Inputs** list.

3. In the context menu, select **Edit Time Intervals**.

*The **Time Intervals** list opens (see "Defining time intervals for a network object type" on page 326).*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

Vehicle Volumes By Time Interval list

1. Enter the desired values.

Element	Description
Cont	**Continued**: Adopts the volume of the previous interval, if several time intervals for vehicle inputs have been defined. The first time interval is therefore always deactivated. The cells are white and the values are valid for this time interval only. ▸ ☐ The option is not selected: The cells are white and are only valid for this interval. ▸ ☑ The option is selected: The cells **Volume**, **VehComp** and **VolType**are gray and are valid for the period of the combined intervals. Only the last cell, for which the **Cont** attribute is not selected, can be edited. When this option is selected, a change in volume is adopted for all the following cells.
TimeInt	**Time interval**: Start and end of the interval in simulation seconds (see "Defining time intervals for a network object type" on page 326).
Volume	**Volume** per time interval (number of vehicles per hour)
VolType	**Volume type**: ▸ **Stochastic**: Stochastic fluctuations of the traffic volume may occur. The cells are white. ▸ **Exact**: Exactly the specified number of vehicles are generated and used. The cells are yellow. If in addition, the **Continued** attribute is selected, the exact number of vehicles is generated for the entire period, which is made up of all time intervals. If the **Continued** attribute is not selected, the exact number of vehicles is generated in each time interval.
VehComp	**Vehicle Compositions**(see "Modeling vehicle compositions" on page 452)

2. If you want to allocate additional volumes to other intervals for the selected vehicle input, right-click on the **Vehicle Volumes By Time Interval** list.

3. From the shortcut menu, choose **Add**.

   The next defined time interval will be added.

4. Enter the desired values.

6.11.3 Modeling vehicle routes, partial vehicle routes, and routing decisions

A route is a fixed sequence of links and connectors:

- A route starts with a routing decision (From Section default color is purple)
- It ends at a so-called To Section (default color is turquoise).

A routing decision point normally leads to several "to sections". Thereby the individual routes branch out from the routing decision point to the individual "to sections".

A route may have any length. You can use a route to simply display a turning movement at a single junction or to connect multiple nodes throughout your network. In many cases, it is useful to deploy routes throughout the Vissim network.

A routing decision only applies to vehicles that have been assigned a vehicle class and that are without any routing information. A vehicle already on a route may only accept new routing information after it has passed the "to section", i.e. destination, of its route. However, this does not include vehicles on partial routes, PT partial routes and parking lot routes.

6.11.3.1 Types of routing decisions and routes

- **Static**: Guides vehicles from a start section (purple) to one of the defined destination sections (turquoise) of the vehicle routes. In this case, the **Route choice method** attribute allows you to select the basis on which vehicles are distributed across vehicle routes:
  - Based on the static share per vehicle route that you define in the **Relative volume** attribute of the vehicle route (see "Attributes of static vehicle routing decisions" on page 468).
  - Based on a user-defined formula. Using the formula, you calculate the share of vehicles for the vehicle route depending on the attributes and attribute values of the vehicles (see "Attributes of static vehicle routing decisions" on page 468). For example, you can use the route choice method **Formula** to distribute taxis across several queues.

  Static routing decisions do not apply to PT vehicles (see "Modeling short-range public transportation" on page 511), (see "Defining PT lines" on page 519).

- **Partial route**: Serves for local distribution of vehicles. Defines a section of one or multiple static routes. For this section, the routes of all relevant vehicles are newly assigned according to the partial route shares of this section. After leaving the partial route, vehicles continue with their original route.

  Partial routing decisions do not apply to PT vehicles.

- **Partial PT route**: Defines a section of one or multiple static routes. For this section, the routes of all relevant vehicles of the PT lines selected are newly assigned according to the partial route shares of this section. Use the **Route choice method** to specify the basis on which vehicles are distributed across the partial route share of the vehicle route (see "Attributes of partial vehicle routes" on page 488).After driving on the partial route, vehicles of PT lines continue driving on their original route.

Similar to when defining PT line stops, you can define PT stops for your PT partial route, including attributes for dwell time (see "Modeling PT lines" on page 518):

As long as a vehicle has not completed its original line route, its **Departure offset** is treated like an offset at a stop of the original route (see "Attributes of PT lines" on page 520). Once the PT vehicle has passed the "to section" of its original line route, the **Departure offset** specified for a PT partial route stop is interpreted as relative to the simulation time when the vehicle passes the respective routing decision point.

- **Parking Lot**, only for parking lots of the type **Real parking spaces**: Defines a routing decision point used to automatically generate routes leading to each of the respective parking lots and back to the network. You select parking lots instead of destination sections.
- **Managed Lanes**: Routes vehicles via two parallel routes (managed lanes), from the start section to a destination section. For a routing decision of the type **Managed Lanes**, you must define a managed lanes facility with a **toll pricing model** and **decision model**. Consequently, the following is taken into account:
  - occupation rate of vehicles with one, two or three or more persons
  - the time of day
  - the current traffic situation, including time savings and average speed

  If the current Managed Lanes route is replaced with a new route at such a **Managed Lanes** routing decision, it is possible that the travel times for previously begun Managed Lanes routes continue to be counted until the vehicle passes its chosen destination.

  The following conditions must be fulfilled for this to occur:

  - The vehicle maintains its decision for or against the toll.
  - The old destination is located on the new route or the new destination is on the old route.

  If the conditions are not fulfilled, the total travel time for the old route is proportionally estimated when more than 75% of the length has been completed.

  Like all other routing decisions types, only the vehicles of the selected vehicle classes will be taken into account. Vehicles of a type, in which the classes are not selected here, use neither the toll route nor the toll-free route. Thus, for example, HGVs can be excluded. Note the effects of routing decisions of the type managed lanes facilities (see "Mode of action of routing decisions of the type Managed Lanes" on page 463).

6.11.3.2 Routing decisions and routes for dynamic assignment

- **Dynamic**: Defines a routing decision point, at which traffic is re-routed. For the route, you must have defined a condition or strategy (see "Defining dynamic routing decisions" on page 762), (see "Modeling parking lots and zones" on page 698).

- **Closure**: Defines a route as a link sequence that will not be available for dynamic assignment (see "Influencing the path search by using cost surcharges or blocks" on page 787). You need not assign time intervals to routes of the type **Closure**. You do not have to enter a relative volume.

6.11.3.3 Placing the routing decision and the mode of action in the simulation

Mode of action of routing decisions for routes

If no route is assigned to a vehicle in a simulation run, the vehicle is assigned its route as soon as a routing decision marker traverses it.

One of several routes at a routing decision point is selected based on the route choice method, which is either the Monte Carlo method, i.e. proportional to the relative volumes on each vehicle route, or a user-defined formula. Using the formula, you calculate the share of vehicles for the vehicle route depending on the attributes and attribute values of the vehicles.

On links with multiple lanes, a vehicle driving on a route attempts to independently choose a lane for the relevant connector according to the **Lane change** attribute value set (**Lane change distance** set (default 200 m), so that the connector can be reached without further lane changes. Long advance links allow the vehicles a timely classification (pre- sort). In unfavorable cases, for example, when the advance link which is selected is too short, it is possible that many unrealistic lane changes lead to traffic disruptions, which do not exist in reality. Most of the time such cases are preventable using suitable modeling.

In 2D animation, a current change of lanes, as well as the desire to change lanes is visualized via a small red line to the right or left of the vehicle (representing the indicator), from the defined **Lane change distance** on. This is also the case for lane changes on connectors.

In 3D animation, a current lane change and the desire to change lanes is shown via an indicator, if this is defined for the 3D model of the vehicle.

Vehicles in the adjacent lanes decelerate partially cooperatively in order to allow the blinking vehicle to merge (see "Applications and driving behavior parameters of lane changing" on page 300).

Mode of action of routing decisions for partial routes

Partial routes can be used, for example, for variable message signs to model multiple alternative routes, without having to change every single route that leads to the position of the variable message sign. If two alternative routes are possible, only one of the partial routing decisions with two routes must be defined, which is assigned to the total volume for each desired share.

Positioning of routing decisions

- If you define routes for links with multiple lanes, you must position the routing decision adequately far enough from the point in which the routes separate. This will prevent unrealistic jams, which occurs because the decision marker assigns all vehicles a route and not just a portion of the vehicles. This allows more lane changes to take place in the simulation than are possible in reality.

- Position the routing decision section so that it is located before the end of the longest known jam in this section.
- When you insert numerous routing decisions, for example, in order to model turns per junction separately, a vehicle with an active routing decision transverses and ignores these routing decisions until it has reached the destination section of its route. In order for a vehicle to switch from route to route successfully and thus follow each route consecutively, the end of the first route must be in the movement direction; only afterwards can the second route begin. To do this, position all destination sections (turquoise) of a route on the first connector or at the corresponding position of a link behind the last decision marker of this route. Once you have positioned all start sections (purple) on a link behind the junction and at the end of all connectors, ensure that all preceding routes end before the start of a new route.

Note: Routing decisions are, like all other decision markers, relevant for a vehicle only after the subsequent time step. Therefore the distance between the decision marker and the subsequent link or connector must be defined at least large enough so that the length of the path corresponds to the vehicle with the highest possible desired speed within a time step. If this is not ensured, it is possible that some of the vehicles will not be influenced by the routing decision.

Note: If a vehicle on a route at the last possible position (emergency stop distance) is waiting for an opportunity to change lanes, but this cannot occur within 60 seconds, this vehicle is removed from the network. Otherwise unrealistic interferences and backups will arise. In reality, one can assume that vehicles waiting to change lanes will be compelled to "squeeze in" after a short period of time. You can adjust the standard value of 60 s in the driving behavior parameter **Diffusion time** of the lane change (see "Editing the driving behavior parameter Lane change behavior" on page 300).

Notes:

- If Vissim finds no route between the start section (purple) and the destination section (turquoise), either a connector is missing or the position of the connector is disadvantageous. Check the link sequence.
- For partial routes or toll routes, Vissim checks at the start of the simulation run whether all destination section, which go out from the decision section, are at a collective position on the collective destination link. If this is not the case, the destination link number and the position **at [m]** of the route with the lowest number, which goes out from this starting marker, is taken over for all further partial routes or Managed lanes, which begin with this starting marker.

Mode of action of routing decisions of the type Managed Lanes

In the simulation, routing decisions of the type **Managed Lanes** (Managed Lanes Routing Decisions) only influence the path selection behavior of the vehicles which are already on a static route or on one of the routes, which begins on a routing decision of the type **Managed Lanes**. If a managed lane route traverses a connector blocked for all vehicle classes, the corresponding managed lanes routing decision does not apply.

This also influences vehicles on the paths of a dynamic assignment. If a managed lane route traverses a blocked edge of dynamic assignment, the corresponding managed lanes routing decision does not apply.

Positioning routing decisions of the type Managed Lanes

You can define routing decisions of the type **Managed Lanes** by section or add multiple routing decisions before the managed lane starts. This choice depends on whether the driver shall decide on how many sections to traverse before reaching the first section or whether he shall make that decision each time before reaching one of the sections.

Positioning routing decisions of the type Managed Lanes by section

This option is useful if the toll is actually displayed at the routing decision for the next section and the driver is only then to decide whether to use the toll route:

- Place a routing decision on the toll-free part of the highway, before each access from the toll-free part of the highway to the parallel-running toll part of the highway.
- For each of these routing decisions, place the destination behind the next possible exit from the toll part to the toll-free part of the highway.
- Place a routing decision on the toll road of the highway, before each access from the toll part to the toll-free part of the highway.
- On each of the toll-free parts of the highway, for each of these routing decisions, place the destination behind the next possible exit from the toll road to the toll-free part of the highway.

Inserting routing decisions of the type Managed Lanes at the beginning of managed lane

This option is useful when the toll for individual sections of the total toll distance is displayed before the first routing decision, nothing changes at the end of the first section and no other toll is displayed there.

Using multiple, successive routing decisions, you can model various toll route options for a driver. If you e.g. want the vehicle to be able to use one, two or three managed lane segments, position the most expensive routing decision option on the toll-free route, so that the vehicle has to traverse it first, then drives downstream to the next expensive routing decision option and last to the least expensive routing decision option. Place these types of routing decisions on toll-free routes only.

For routing decisions of the type **Managed Lanes** that follow upon each other at a distance of less than10 m, the following applies: If the vehicle has selected a managed lane based on one of these routing decisions, you cannot use another routing decision of this group that lies further downstream to assign the vehicle a different toll lane. As soon as the vehicle has selected a managed lane, it ignores any other managed lane routing decisions of this group that lie further downstream.

Probability of switching to a toll lane

At the first passing of a routing decision of the type **Managed Lanes** each vehicle receives a random number for the probability that it will change to the toll lane. This random number is then used for all further routing decisions of the Type **Managed Lanes** . This ensures that the vehicle will only change its original decision when a completely different traffic-related state ensues.

Two routing decisions of the type **Managed Lanes** should only use the same Managed lanes facility when the following conditions are in place:

- The characteristics of the toll-free routes of both routing decisions are identical for the most part.
- The course of the toll lanes of both routing decisions are identical for the most part (see "Saving managed lane data to a file" on page 1084).

This may be the case, for example, if you model a high occupancy toll lane for certain vehicles on a highway parallel to toll-free lanes.

Modeling the toll part of a highway

The toll part of a highway can be a structurally separate road parallel to the toll-free lanes of the highway. You use links and connectors to define both parts of the highway and the ramps for entries and exits. You use routing decisions of the type **Managed lanes**, the respective toll route and toll-free route to model the use of these parallel lanes per section. To enable the vehicle to decide whether to use the high occupancy toll lane or the toll-free part of the highway, place the Managed lanes routing decision before the junction between the toll route and the toll-free route. Then have one of these two routes run along a different link sequence that starts at the same Managed lanes routing decision and ends at a common destination cross section. (see "Modeling a separate route course for the toll route and toll-free route" on page 480).

The toll part can also be a high occupancy toll lane (HOT lane):

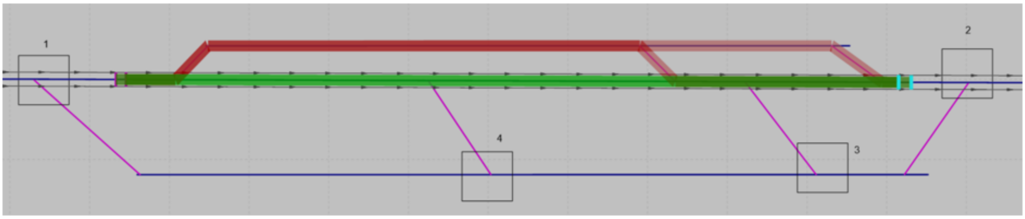
- On a high occupancy toll lane, vehicles whose number of passengers falls below a specified value are subject to toll.
- Vehicles on the HOT lane are toll-free if the number of passengers exceeds a specified value, for example buses, taxis and HOV3+ vehicles (vehicles with more than three passengers, for example).

Use occupancy distribution to assign the number of occupants for the vehicles of a vehicle type (see "Using occupation distributions" on page 255). The occupancy rate of the vehicle is taken into account in the toll pricing calculation model (see "Defining managed lane facilities" on page 327), (see "Defining toll pricing calculation models" on page 331). Using the toll pricing calculation model, you can take current traffic conditions into account by defining the influence of travel time savings and average speed on toll costs. The decision model defines the probability of using the toll route depending on the vehicle class to which a vehicle is assigned (see "Defining decision model for managed lane facilities" on page 329). You group the vehicle types that can use the HOT lane toll-free into one vehicle class. These vehicles always use the toll-free route.

For path selection in the Vissim network, dynamic assignment takes link sequences without toll routes into account as well as link sequences on which managed lane facilities and toll routes have been defined (see "Defining a vehicle route of the type managed lane" on page 476).

Determining the number of toll routes based on possible paths - examples

In the Vissim network depicted in the following figure, the highway branches downstream of node **1** into a toll-bearing part (upper link sequence, red) and a toll-free part (green). In the further parallel course of the link, a ramp enables the vehicles to change from the toll-bearing part to the toll-free part. The link sequences of the toll-bearing and toll-free parts are reunited upstream of node **2**. The lower link sequence via nodes **4** and **3** allows vehicles to bypass the highway.



For comparable use cases in which the vehicle can optionally continue on or leave the toll-bearing part to switch to the toll-free part, you define a managed lanes routing decision with a toll-free route and a toll route for each possible path and adjust the route course:

1. Define a managed lanes routing decision upstream of the junction of the toll and toll-free parts of the highway (see "Defining a vehicle route of the type managed lane" on page 476).
2. Add the common destination section of the respective toll route and toll-free route downstream of the junction of the toll and toll-free parts of the highway.
3. Move the destination section and separate the course of the toll route from that of the toll-free route (see "Modeling a separate route course for the toll route and toll-free route" on page 480).
4. Repeat these steps for a managed lanes routing decision, whose toll route leads via the toll part of the highway and then over the ramp and the toll-free part of the highway to the destination section. Again, separate the toll route from the toll-free route.

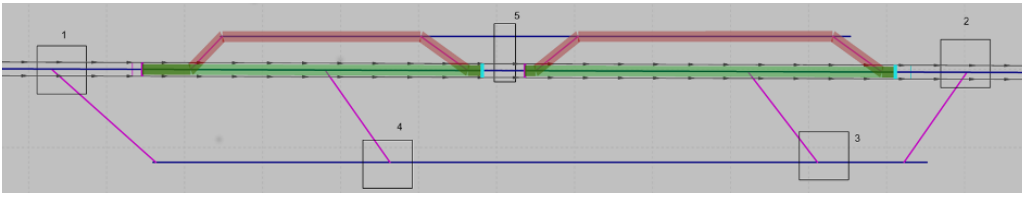
If you model several ramps, the vehicle can decide in sections whether to use a ramp to switch from the toll lane to the toll-free lane. After traversing a managed lanes routing decision, the vehicle ignores all other managed lanes routing decisions for the following 10 m.

Toll routes In dynamic assignment

Dynamic assignment does not decide at the start of the journey whether the vehicle uses the toll route or the toll-free route. If the use case in the above figure is simulated using dynamic assignment, the latter decides when the vehicle starts driving whether it will take the lower path via nodes **4** and **3** or the upper path via the toll routes. Only after the vehicle has driven straight through node **1** and has reached the managed lanes routing decision downstream of node **1** does it decide whether to take the toll-free route straight ahead or the upper route that is subject to toll (see "Calculating toll using dynamic assignment:" on page 798).

Requirements of dynamic assignment for modeling separate toll routes

In addition to changing from the toll part of the highway via a ramp to the toll-free part, you can also use a ramp to model the change from the toll-free lane to the toll- lane in sections:



- Insert a node upstream of each ramp that connects the toll-free link sequence with the toll-bearing link sequence. The node must extend over both link sequences (node **5**).
- The destination section of the toll route and the toll-free route of the section must be placed in front of the node.
- In addition to the toll route and the toll-free route for each section, you can define a toll route and toll-free route that runs along the entire link sequence. To do so, place the managed lanes routing decision downstream of node **1** and upstream of the junction of the toll-free route and the toll route, whose destination section you need to place downstream of the final combination of the toll-free route and the toll route, upstream of node **2**. Move the destination section and separate the course of the toll route from that of the toll-free route (see "Modeling a separate route course for the toll route and toll-free route" on page 480).

6.11.3.4 Defining static vehicle routes

To define a static vehicle route, insert a routing decision on a link and a destination section on a destination link. You may also define multiple destination links or connectors for a routing decision. The routing decision or the destination section may also lie on a connector.

You can assign the interval limits via the time intervals (see "Defining time intervals for a network object type" on page 326) , (see "Calling time intervals from an attributes list" on page 327). If a distribution on a percentage basis of the traffic volume to the vehicle routes of a routing decision varies temporally, you must define multiple time intervals which do not overlap.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Static**.

Tip: Alternatively to the following steps, to insert a routing decision, from the shortcut menu, choose **Static vehicle routing decision**.

4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.

5. Release the keys.

   By default, a purple bar is inserted. If for this start section you want to insert multiple destination sections, carry out the following steps accordingly. Thereby you can insert a destination section and subsequently define each of its attributes.

   If you would like to insert a destination section for this start section, execute the next steps only once.

6. On the desired link, point the mouse pointer to the desired position of the first destination section.

   If Vissim does not find a valid link sequence, neither a yellow band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

   If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is displayed as a yellow band by default.

On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.

7. Right-click this position.

   The context menu opens.

8. If you do not wish to insert additional destination sections, from the shortcut menu, choose **Create static vehicle route: Define end**.

9. To insert addition destination sections, click the desired positions.

10. If you do not want to add any additional destination sections, in the Network editor, click in an empty area.

    *A turquoise bar is added for the destination section by default. The **Static vehicle routing decision** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*

11. Edit the attributes (see "Attributes of static vehicle routing decisions" on page 468), (see "Attributes of static vehicle routes" on page 470).

Tip: You may assign a routing decision to a destination section later on:

1. On the Network objects toolbar, click **Vehicle Routes**.
2. Hold down the CTRL key.
3. In the Network Editor, right-click the routing decision.
4. Release the keys.
5. Right-click the position on the link where you want to add the destination section.
6. From the shortcut menu, choose **Create static vehicle route: Define end**.

6.11.3.5 Attributes of static vehicle routing decisions

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Static routing decisions**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number of static vehicle routing decision
Name	Name of static vehicle routing decision
Link	Number of the link on which the static routing decision is located
Pos	**Location**: Distance to the beginning of link or connector
AllVehTypes	☑ If the option is selected, all vehicle types follow the static vehicle routing decision. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	Valid **vehicle classes**
RouteChoiceMeth	Route choice method for static vehicle routing decision: - **Static** (default value): Route choice is made based on the **Relative volume** attribute of the vehicle route. - **Formula**: Route choice is based on a user-defined formula. Using the formula, you calculate the share of vehicles for the vehicle route depending on the attributes and attribute values of the vehicles.
CombineStaRoutDec	**Combine static routing decisions**: Combines static routes which follow one other into one route. You therefore prevent vehicles from recognizing an imminent required change of lanes only when passing the next routing decision and therefore artificially causing queues. Vissim checks at the start of the simulation for all vehicle routes, whether on the previous link of the vehicle route there is still a further routing decision downstream of the end of the route, for which the option **Consider subsequent static routing decisions** is selected (see "Editing the driving behavior parameter Lane change behavior" on page 300). If node routes are e.g. imported from PTV Vistro, ANM Import automatically selects this attribute.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Static vehicle routes (see "Attributes of static vehicle routes" on page 470)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.11.3.6 Attributes of static vehicle routes

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Static routing**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of static vehicle routing decision of vehicle route
Formula	Enter a formula with the attribute and the attribute value that determines the share of vehicles on this vehicle route. Only active if in the **Static vehicle routing decision** list, the route choice method **Formula** is selected. The **Formula** attribute is independent from time intervals.
No	Number of static vehicle route
Name	Designation
DestLink	**Destination link**: Number and name of link on which the static vehicle route ends
DestPos	**Destination position**: Distance between destination section and beginning of link or connector
RelFlow	**Relative volume** in time interval = absolute volume in time interval: Sum of the volumes of all time intervals . Only active if in the **Static vehicle routing decision** list, the route choice method **Static** is selected.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Link sequence**.

Attributes of link sequence are displayed: Numbers of links and connectors the static vehicle route traverses (see "Attributes of links" on page 409):

3. On the list toolbar, in the **Relations** list, click the desired entry.

4. Enter the desired data.

Notes:

- Relative volumes: Relative volumes of a route must correspond to their absolute volumes. However, for example, numerous turn volume calculations can deviate at subsequent junctions. Therefore vehicles in the network are neither inserted nor removed automatically. You must secure consistent data for modeling real scenarios.
- Instead of absolute values, Vissim uses relative shares in order to determine the traffic volume for destination sections of a routing decision. Thereby real volume values and percentage shares can be entered. Internally, Vissim totals the relative volume and calculates the absolute share of each stream automatically.

6.11.3.7 Combining static routes

You can connect a static vehicle route with another static vehicle route.

Look ahead distance:

- The routing decision of the static vehicle route must lie on the same route as the destination section of the other static vehicle route further upstream.
- In the **Vehicle classes** attribute of the two static vehicles, the same vehicle classes must be assigned.
- Only one routing decision may be selected.

1. In the Network Editor, right-click the routing decision of the downstream static vehicle route.
2. From the context menu, choose **Combined static routes**.

The link sequence of each vehicle route of the selected routing decision is appended to each vehicle route with a destination section further upstream on the link of the selected routing decision. These destination sections and the selected routing decision are deleted in the Network editor. The link sequences are connected and then displayed as a yellow band by default.

*The selected static vehicle routing decision is deleted from the **Static vehicle routing decisions** list.*

*The **Static vehicle routes** list on the right is updated for the static vehicle routing decision, to which the link sequence is appended.*

Tip: Alternatively, in the **Static vehicle routing decisions** list, right-click the routing decision of the downstream static vehicle route. Then from the shortcut menu, choose **Combine routes**.

6.11.3.8 Defining parking routes

If you want a vehicle to use a parking lot, define a vehicle route of the type **Parking lot** that leads to the desired parking lot. To define a parking route, insert a routing decision on a link and on a destination section located on the parking lot of your choice. For a routing decision, you may also define multiple destination sections located on different parking lots. Your routing decision may lie on a connector. The destination section may also lie on a connector, if the parking lot is located on a connector.

You can assign the interval limits via the time intervals (see "Defining time intervals for a network object type" on page 326) , (see "Calling time intervals from an attributes list" on page 327). If a distribution on a percentage basis of the traffic volume to the routes of a routing decision varies temporally, you must define multiple time intervals which do not overlap each other.

For parking routing decisions, time intervals assigned in the **Select Attributes** window, in the list on the left, are taken into account for the subattributes **Parking duration** and **Parking rate**. They can be selected and displayed in the **Parking Routing Decisions** list (see "Selecting attributes and subattributes for columns of a list" on page 112).

If you define a parking route to a parking lot in which the vehicle can pull out of the parking space backwards, Vissim will create the parking route internally, using several sections: The shortest path search defines the section of the routing decision leading to the parking lot. For driving backwards onto the original route, Vissim defines the following sections:

- A section extends up to the point where the vehicle stops driving backwards, comes to a standstill and then continues its route driving forwards.
- A section extends up to the point where a connector begins that leads to the parking lot.
- A section extends from the beginning of this connector up to the next section of the original route.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Parking lot**.

> Tip: Alternatively to the following steps, to insert a routing decision, from the shortcut menu, choose **Parking routing decision**.

4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.
5. Release the keys.

   By default, a purple bar is inserted. If for this start section you want to insert multiple destination sections, carry out the following steps accordingly. Thereby you can insert a destination section into a parking lot and subsequently define each of its attributes.

   If you would like to insert a destination section for this start section, execute the next steps only once.

6. Move the cursor into the parking spaces of the desired parking lot.

   If Vissim does not find a valid link sequence, neither a blue band nor a turquoise bar are displayed, or the band might be interrupted. Select another parking lot for the destination section or correct the Vissim network, for example if a link is not connected properly to a connector.

   If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is displayed as a blue band by default.

   On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar within the parking lot indicates the possible position of the destination section.

7. Click this position.
8. To insert additional destination sections, in respective parking lots, click the desired position.
9. If you do not want to add an additional destination section, in the Network editor, click in an empty area.

   *A turquoise bar is added for the destination section by default. For vehicle routes of the type **Parking Lot**, the route ends at the beginning of the destination parking lot. The **Parking routing decisions** list opens, if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*

10. Edit the attributes (see "Attributes of parking routing decisions" on page 474), (see "Attributes of parking routes" on page 476).

Tip: You may assign a routing decision to a destination section later on:

1. On the Network objects toolbar, click **Vehicle Routes** > **Parking lot**.
2. Hold down the CTRL key.
3. In the Network Editor, right-click the routing decision.
4. Release the keys.
5. In the parking lot of your choice, click the desired position of the destination section.
6. If you do not want to add an additional destination section, in the Network editor, click in an empty area.

6.11.3.9 Attributes of parking routing decisions

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Parking Routing Decisions**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number of parking routing decision
Name	Name of parking routing decision
Link	Number of the link, on which the parking routing decision is located
Pos	Distance of the parking routing decision to the beginning of link or connector
AllVehTypes	☑ If the option is selected, all vehicle types follow the parking routing decision. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	Valid **vehicle classes**

Column	Description
ParkRate	**Parking rate**: Share of vehicles belonging to the allocated vehicle classes. A vehicle, which has been assigned a parking space via a parking routing decision, carries this out on an automatically generated route and parks for as long as the dwell time distribution specifies. This dwell time distribution is allocated in the Parking Routing Decisions. After the dwell time expires, the vehicle leaves the parking lot and begins on an automatically generated route, which brings the vehicle via the shortest path back to its original route behind the position of the routing decision.
Parking duration	Parking time
GenBy	**Generated by**: Indicates whether the parking routing decision is user-defined or has been generated by Vissim. - **User** The parking routing decision was defined by a Vissim user. - **Dynamic assignment**: only for parking routing decisions of dynamic assignment: For real parking lots, Vissim automatically creates a parking routing decision 50 m from a parking lot and in the attribute **GenBy** box, enters **Dynamic assignment**. You can change the distance between the parking routing decision and the parking lot in the attribute **Routing decision distance** of the parking lot. Use this distance to ensure that the parking routing decision lies at the beginning of the last edge before the parking lot. After simulation has been completed, the list no longer shows automatically generated parking routing decisions of dynamic assignment.
FullOccupBehav	**Full occupancy behavior**: Waiting behavior of vehicles traversing the parking routing decision: - **Waiting**: If there is no parking space available, the vehicle drives to the next parking space that will become available (a vehicle is currently still parked there) and waits. - **Drive on**: If no parking space is available, the vehicle ignores the parking routing decision. By default, the attribute is not displayed in the list.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Parking rate by time interval: The attribute **Parking rate** is described above.
- Vehicle classes (see "Using vehicle classes" on page 280)
- Parking routes (see "Attributes of parking routes" on page 476)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.11.3.10 Attributes of parking routes

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Parking Routes**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of the parking routing decision of the parking route
No	Number of parking route
Name	Description
ParkLot	**Parking lot**: Name of parking lot Additional attributes (see "Modeling parking lots" on page 493)

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list box, click > **Link sequence**.

   Attributes of link sequence are displayed: Numbers of links and connectors via which the route leads. (see "Attributes of links" on page 409)

3. Enter the desired data.

6.11.3.11 Defining a vehicle route of the type managed lane

To define a vehicle route of the type **Managed lanes**, insert a managed lanes decision on a link and insert a destination section on a destination link. The managed lanes routing decision or the destination section may also each lie on a connector.

A routing decision of the type **Managed lanes** has a maximum of two routes: the managed lane route and the general purpose route. After you have positioned the managed lanes routing decision, add the destination section of the toll route (red by default). Subsequently, at the destination section, insert the toll-free route (green by default). By default, the toll-free route runs along the same link sequence as the toll route from the managed lanes routing decision to the destination section. Both routes automatically have the same destination section. You can have one of these two routes run via a different link sequence (see "Modeling a separate route course for the toll route and toll-free route" on page 480).

A routing decision of the type **Managed lanes** is only taken into account in the simulation when it is complete: This means a route of the type **Managed lanes** and a general purpose route must be defined. Then a managed lanes facility with a user-defined toll price model and decision model must be assigned (see "Defining decision model for managed lane facilities" on page 329), (see "Defining toll pricing calculation models" on page 331). Note the effects of routing decisions of the type managed lanes facilities (see "Mode of action of routing decisions of the type Managed Lanes" on page 463).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button ▾ is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Managed lanes**.
4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.
5. Release the keys.

   The link or connector is displayed in red. A turquoise bar is added for the destination section by default. It superimposes the purple bar of the managed lanes routing decision.

6. On the desired link, point the mouse pointer to the desired position of the destination section.

On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. The turquoise bar shows the possible position of the destination section.

7. Right-click the position.

   The context menu opens.

8. Then select **Create managed lane route: Define end**.

   *The managed lane route continues to be displayed in red. The turquoise bar for the destination section is inserted. The **Managed lanes routing decisions** list opens, if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists. In the **Managed Lane Routes** list on the right, a new row with a vehicle routing decision of the type **Toll** is created. Next, in the Network editor, add the respective vehicle routing decision type **General purpose**.*

9. Right-click the turquoise bar of the destination section.

   The context menu opens.

10. Select **Add vehicle route to existing destination**.

    *The toll-free route on the toll route is highlighted in green. In the **Managed Lane Routes** list, a new row, with a vehicle routing decision of the type **Toll**, is inserted.*

    By default, the course of the toll route and the toll-free route is identical up to their destination section. You can have one of these two routes run along a different link sequence that starts at the same managed lanes routing decision and ends at the common destination section. This allows you, for example, to model the section of a highway that has a toll lane parallel to toll-free lanes (see "Modeling a separate route course for the toll route and toll-free route" on page 480).

11. Edit the attributes (see "Attributes of managed lanes routing decisions" on page 478), (see "Attributes of managed lane routes" on page 479).

6.11.3.12 Attributes of managed lanes routing decisions

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Managed lanes routing decisions**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number of managed lane routing decision
Name	Name of managed lane routing decision
Link	Number and name of link on which the managed lane routing decision lies
Pos	**Location**: Distance to the beginning of link or connector
AllVehTypes	☑ If the option is selected, all vehicle types follow the managed lane routing decision. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	Valid **vehicle classes**
Managed lanes facility	Name of assigned managed lane facility (see "Defining managed lane facilities" on page 327)
Managed lanes data	The result attribute lists: travel time savings / average speed / current toll for managed lane route.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Using vehicle classes" on page 280)
- Managed lane routes (see "Attributes of managed lane routes" on page 479)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.11.3.13 Attributes of managed lane routes

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Managed Lanes Routes**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of managed lanes routing decision of the managed lane route
No	Number of managed lane route
Name	Desription
Type	**Toll**: toll route, highlighted in red by default **General purpose**: toll-free route, highlighted in green by default
DestLink	**Destination link**: Number and name of link on which the managed lane route ends
DestPos	**Destination position**: Distance between destination section and beginning of link or connector

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Link sequence**.

   Attributes of link sequence are displayed: Numbers of links and connectors affected by the managed lane route (see "Attributes of links" on page 409):

3. Enter the desired data.

6.11.3.14 Modeling a separate route course for the toll route and toll-free route

Once the toll route (green) and the toll-free route (red) have been inserted, their route courses are identical by default up to the destination section. You can let one of these two routes run along a different link sequence that starts at the same managed lanes routing decision and ends at the common destination section. This allows you, for example, to model the section of a highway that has a toll lane parallel to a toll-free lane.

1. Ensure that the managed lanes routing decision is located upstream of the branching of the desired toll route and toll-free route.
2. Ensure that the destination section is located downstream of where the link sequences of the toll-free and toll-bearing routes are merged.

   That can also be on a ramp or downstream of a ramp where the vehicle can switch from the toll route to the toll-free route.

3. Click a destination section, and keep the mouse button pressed.
4. Hold down the mouse button and move the destination section to the link sequence via which the route is to run.

   The new route course is highlighted in color.

5. Release the mouse button.

   The following window opens: ***Shall route 1 be adjusted to decision <Number> as shown?***.

6. Select the desired entry.

Button	Description
Yes	Only route **1** (by default the toll route) of the two routes of the managed lanes routing decision **Decision <Number>** is moved to the new link sequence. If this route is the toll route, the toll-free route continues to run on the original link sequence.
Yes (all)	Toll-free route and toll route will be moved to the new link sequence.
No	Only route **2** (by default the toll-free route) of the two routes of the managed lanes routing decision **Decision <Number>** is moved to the new link sequence. If this route is the toll-free route, the toll route continues to run on the original link sequence.
No (all)	Both the toll-free route and toll route remain on their original course.

6.11.3.15 Defining a vehicle route of the type closure

To define a vehicle route of the type **Closure**, insert a routing decision on a link and a destination section on a destination link. You may also define multiple destination links or connectors for a routing decision. The routing decision or destination section may also lie on connectors.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Closure**.

Tip: Alternatively to the following steps, to insert a routing decision, from the shortcut menu, choose **Vehicle route closure decision**.

4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.
5. Release the keys.

   By default, a purple bar is inserted. If for this start section you want to insert multiple destination sections, carry out the following steps accordingly. Thereby you can insert a destination section and subsequently define each of its attributes.

   If you would like to insert a destination section for this start section, execute the next steps only once.
6. On the desired link, point the mouse pointer to the desired position of the first destination section.

   If Vissim does not find a valid link sequence, neither a red band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

   If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is displayed as a red band by default.

   On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.
7. Right-click this position.

   The context menu opens.
8. If you do not wish to insert additional destination sections, from the shortcut menu, choose **Create vehicle route closure: Define end**.
9. To insert addition destination sections, click the desired positions.
10. If you do not want to add any additional destination sections, in the Network editor, click in an empty area.

    *A turquoise bar is added for the destination section by default. The **Vehicle route closure decision** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*
11. Edit the attributes (see "Attributes of route closure decisions" on page 483), (see "Attributes of route closures" on page 484).

Tip: You may assign a routing decision to a destination section later on:

1. On the Network objects toolbar, click **Vehicle Routes**.
2. Hold down the CTRL key.
3. In the Network Editor, right-click the routing decision.
4. Release the keys.
5. Right-click the position on the link where you want to add the destination section.
6. From the shortcut menu, choose **Create vehicle route: Define end**.

6.11.3.16 Attributes of route closure decisions

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Route closure decisions**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number of route closure decision
Name	Name of route closure decision
Link	Number and name of link on which the route closure decision lies
Pos	**Position**: Distance between route closure decision and beginning of link or connector

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list box, click > **Vehicle route closures** (see "Attributes of route closures" on page 484).
3. Enter the desired data.

6.11.3.17 Attributes of route closures

- From the **Lists** menu, choose > **Private Transport** > **Routes** > **Route Closures**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of vehicle routing decision of route closure
No	Number of route closure
Name	Description
DestLink	**Destination link**: Number and name of link on which route closure ends
DestPos	**Destination position**: Distance between destination section and beginning of link or connector

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list box, click > **Link sequence**.

   Attributes of link sequence are displayed: Numbers of links and connectors affected by the route closure (see "Attributes of links" on page 409):

3. Enter the desired data.

6.11.3.18 Defining partial vehicle routes

To define a partial vehicle route, insert a routing decision on a link and a destination section on a destination link. The partial routing decision or the destination section may also lie on a connector.

You can assign the interval limits via the time intervals (see "Defining time intervals for a network object type" on page 326) , (see "Calling time intervals from an attributes list" on page 327). If a distribution on a percentage basis of the traffic volume to the routes of a routing decision varies temporally, you must define multiple time intervals which do not overlap each other.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Partial route**.

Tip: Alternatively to the following steps, to insert a routing decision, from the shortcut menu, choose **Vehicle partial routing decision**.

4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.

5. Release the keys.

   By default, a purple bar is inserted.

6. On the desired link, point the mouse pointer to the desired position of the destination section.

   If Vissim does not find a valid link sequence, neither a yellow band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

   If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is displayed as a yellow band by default.

   On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.

7. Right-click this position.

The context menu opens.

8. Then select **Create vehicle partial route: Define end**.

   A turquoise bar is added for the destination section by default. The ***Vehicle partial routing decision*** *list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*

9. Edit the attributes (see "Attributes of partial vehicle routing decisions" on page 487), (see "Attributes of partial vehicle routes" on page 488).

6.11.3.19 Defining a partial route based on an existing partial route

If you have defined a partial route, PuT partial route, or a route of the type **Managed Lanes** for a link, you can select the destination section and then insert a new partial route that starts at the existing routing decision and ends at the current destination section.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button ▾ *is shown.*

2. Click **Vehicle Routes** again.

   A list box opens.

3. Select the desired entry.
   - **Partial**
   - **Partial PT**
   - **Managed Lanes**

4. In the Network editor, right-click the destination section of the partial route you want to use as the basis of your new partial route.

5. From the context menu, choose **Add Vehicle Route**.

6. On the desired link, point the mouse pointer to the desired position of the new destination section.

If Vissim does not find a valid link sequence, neither a colored band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

You must select a different destination link or a new position for the destination section or correct the Vissim network.

If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is by default displayed as a colored band.

On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A colored bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.

7. Hold down the CTRL key and right-click on this position.
8. Release the keys.

   *Per default, a colored bar is inserted for the destination section. The list **<Variant> Vehicle Routes** opens.*

9. Edit the attributes (see "Attributes of partial vehicle routing decisions" on page 487), (see "Attributes of partial vehicle routes" on page 488)

   The attributes are saved in the list.

10. In the Network editor, click into an empty section.

The attributes are saved in the lists of the partial route and the routing decision.

6.11.3.20 Attributes of partial vehicle routing decisions

1. From the **Lists** menu, choose > **Private Transport** > **Routes** > **Partial routing decisions**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Column	Description
No	Unique **Number** of vehicle partial routing decision
Name	Name of partial vehicle routing decision
Link	Number of the link on which the vehicle partial routing decision lies

Column	Description
Pos	**Location**: Distance to the beginning of link or connector
RouteChoiceMeth	Route choice method for partial vehicle routing decision: - **Static** (default value): Route choice is made based on the **Relative volume** attribute of the vehicle route. - **Formula**: Route choice is based on a user-defined formula. Using the formula, you calculate the share of vehicles for the vehicle route depending on the attributes and attribute values of the vehicles.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Partial vehicle routes (see "Attributes of static vehicle routes" on page 470)
- Link (see "Attributes of links" on page 409)
- Destination link (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.11.3.21 Attributes of partial vehicle routes

1. From the **Lists** menu, choose > **Private Transport** > **Routes** > **Partial routes**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of partial routing decision of partial vehicle route (see "Attributes of partial vehicle routing decisions" on page 487)
Formula	Enter a formula with an attribute and attribute value that determines the share of vehicles on this vehicle route. Only active if in the **Partial vehicle routing decisions** list, the route choice method **Formula** is selected. The **Formula** attribute is independent from time intervals.
No	Unique **Number** of partial vehicle route

Column	Description
Name	Name of partial vehicle route
Destination link	Number and name of link on which partial vehicle route ends
DestPos	**Destination position**: Distance between destination section and beginning of link or connector
RelFlow	**Relative volume** in time interval = absolute volume in time interval: Sum of the volumes of all time intervals . Only active if in the **Partial vehicle routing decisions** list, the route choice method **Static** is selected.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle routing decision (see "Attributes of partial vehicle routing decisions" on page 487)
- Link sequence (see "Attributes of links" on page 409)
- Destination link (see "Attributes of partial vehicle routing decisions" on page 487)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.11.3.22 Inserting intermediate points into a vehicle route

You may insert intermediate points into a vehicle route. Subsequently, you can change the routing process (see "Changing routing procedure via intermediate points" on page 490). When you add a destination section, it is highlighted by default and a colored band shows the course of the link. This identifies the following possibilities:

- You can add additional destination sections for the routing decisions (see "Defining parking routes" on page 472).

When you select a destination section, it is highlighted by default and a colored band indicates the course of the road. This identifies the following possibilities:

- You may move the destination section.
- You can add intermediate points on the vehicle route before the marked destination section.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, select the network object type **Vehicle Routes**.
2. Click the destination section of the vehicle route into which you want to insert the intermediate points.

   The link sequence is shown as a colored band.
3. Keep the CTRL key pressed and right-click the desired positions for intermediate points on the colored band of the vehicle route.

   The intermediate points are added.
4. If you do not wish to add any additional intermediate points or destination sections, click on the free space in the Network Editor.

6.11.3.23 Changing routing procedure via intermediate points

You can add temporary intermediate points in a vehicle route and move them to different links. Vissim calculates a new link sequence or partial link sequence from the next upstream lying intermediate point via the current new point up to the next intermediate point downstream. In the case that no further intermediate point was set, the entire route is recalculated. You can add intermediate points already during the definition of vehicle routes (see "Inserting intermediate points into a vehicle route" on page 489).

1. On the Network objects toolbar, click **Vehicle Routes**.
2. Click in the Network Editor on the destination section of the desired route.

   The link sequence is shown as a colored band.
3. Hold down the CTRL key and right-click on the desired positions of the intermediate points.
4. Release the keys.
5. Click the first intermediate point you want to move and keep the left mouse button held down. Then drag the intermediate point to the desired position on another other link.

   The new link sequence is shown as a colored band.
6. Release the keys.
7. If desired, move additional intermediate points.

8. If you would like to confirm the new link sequence and hide the yellow band, in the Network Editor, click an empty area.

6.11.3.24 Defining a vehicle route based on an existing vehicle route

If you have defined a vehicle route of the type Static, Closure or Parking Lot for a link, you can select the destination section and then insert a new vehicle route that starts at the existing routing decision and ends at a new destination section.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button  *is shown.*

2. Click **Vehicle Routes** again.

   A list box opens.

3. Select the desired entry.

   - **Static**
   - **Closure**
   - **Parking Lot**

4. In the Network editor, right-click the destination section of the vehicle route you want to use as the basis for your new vehicle route.

5. From the context menu, choose **Add Vehicle Route**.

6. On the desired link, point the mouse pointer to the desired position of the new destination section.

   If Vissim does not find a valid link sequence, neither a colored band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

   If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is by default displayed as a colored band.

On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.

7. Right-click this position.

   The context menu opens.

8. To insert addition destination sections, click the desired positions.

9. If you do not wish to insert additional destination sections, from the shortcut menu, choose **Create vehicle route: Define end**.

   *Per default, a colored bar is inserted for the destination section. The **<Scenario> Vehicle routing decisions** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*

10. Edit the attributes (see "Attributes of static vehicle routes" on page 470), (see "Attributes of route closures" on page 484), (see "Attributes of parking routes" on page 476).

11. Confirm with **OK**.

The attributes are saved in the lists of the route and routing decision.

6.11.3.25 Moving routing decisions or destination sections

1. Click the header of the Network Editor.

2. Click the desired routing decision or destination section and keep the mouse button pressed.

   When you move the destination section, by default the route is displayed as a yellow band.

3. Move the start section or destination section to the desired position on a link or connector.

4. Release the mouse button.

6.11.3.26 Deleting vehicle routes, partial vehicle routes and routing decisions

1. In the Network Editor, drag the by default purple From section out of the link by holding down the mouse button.

2. Release the mouse button.

The route, partial route or routing decision is deleted. Once you have deleted a routing decision, all of the corresponding routes are deleted.

When you remove a To Section from the link, the From Section remains intact.

Tip: Alternatively, you may also delete vehicle routes, partial vehicle routes and routing decisions in the respective attribute list.

6.11.3.27 Checking and repairing routes

When you split or delete links and connectors that run across routes, these routes are disconnected. Even if you then insert new links or connectors there, these routes remain disconnected. You can have disconnected routes repaired by Vissim.

1. On the Network objects toolbar, click **Vehicle Routes**.
2. Right click into the Network editor.
3. From the context menu, choose **Check and repair all routes**.

Vissim will find new routes for disconnected routes. These may run via the added or other links and connectors in the network.

6.11.4 Modeling parking lots

The network object **Parking Lot** is deployed for the following purposes:

- **Real parking spaces**: With static routes and dynamic assignment, for modeling roadside parking and parking on pick-up/drop-off parking spaces (see "Modeling parking and stopping on the roadside" on page 493), (see "Defining parking lots" on page 499)
- **Abstract parking lot**: For dynamic assignment to model multiple, real parking lots, e.g. in an underground garage or parking garage (see "Modeling parking lots and zones" on page 698)
- **Zone connector**: For dynamic assignment to model the origins and destinations of all trips (see "Modeling parking lots and zones" on page 698)

6.11.4.1 Modeling parking and stopping on the roadside

For parking or stopping on the roadside define:

- a parking lot of the type **Real parking spaces**. Depending on the attributes **Length** and **Length of each space:** the parking lot may have several parking spaces. The parking spaces are aligned one after the other towards the lane.
- a vehicle routing decision of the type **Parking lot**, with one or several vehicle routes. Each vehicle route leads from the vehicle routing decision to a parking lot (see "Defining parking routes" on page 472).

Vehicle routing decisions of the type **Parking Lot** work similarly to vehicle partial routes (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459). However, instead of routes, the desired number of parking lots is assigned. In the attributes of the vehicle routing decision type **Parking lot** you can specify (see "Attributes of parking lots" on page 500):

- **Parking duration**: the desired time distribution, how long you want the vehicle to park
- **Parking rate**: The percentage of vehicles that per time interval you want to assign a parking space of the parking lots assigned.

Positioning a parking lot and vehicle routing decision of the type Parking lot

A parking lot is created in the Network editor on a lane (see "Defining parking lots" on page 499). If the lane does not continue downstream of the parking lot, the emergency stop position of the first downstream connector must not be tangent to the parking lot (see "Attributes of connectors" on page 422).

The beginning of a parking lot must be positioned in movement direction and at a sufficient distance further downstream of the corresponding vehicle routing decision of the type **Parking lot**. This allows vehicles enough time to slow down and safely reach the first parking space. Otherwise, it might not be possible to allocate a vehicle to the first parking space(s), or the vehicle might miss its assigned parking space and block the traffic flow.

If the parking lot is located on a single-lane link, vehicles parked there might hinder free traffic flow on this link. With each time step, vehicles heading for a parking space in this parking lot check whether there is another parking space available. The vehicle thus drives far downstream in order to move up if possible and if the value of the **Attraction** attribute has been set so high that the parking space downstream is more attractive than the parking spaces upstream.

Vehicle routing decision assigns vehicle parking space

The parking space is assigned when the vehicle crosses a vehicle routing decision of the type **Parking lot**. Vissim assigns a parking space to the percentage of vehicles specified in the **Parking rate** attribute of the vehicle routing decision. In doing so, the vehicle classes selected, including PT vehicles, are taken into account. The vehicles Vissim has already assigned a parking space are not considered.

Vissim selects the parking lot and parking space depending on the following attributes (see "Attributes of parking lots" on page 500):

- **Opening hours**
- **Maximum parking time**
- Available parking spaces for the respective vehicle length, depending on the **Length per parking space** attribute
- **Attraction**

A parking space is only assigned if the following conditions are met:

- The parking lot is open (**Opening hours** attribute).
- The time distribution period selected in the **Parking duration** attribute of the vehicle routing decision type **Parking lot** is shorter than the time period specified in the **Maximum parking duration** attribute of the parking lot.
- There is enough parking capacity. The parking capacity of real parking spaces is based on the length of the parking lot and the length of each parking space. To change the parking capacity, in the Network editor, change the length of the parking lot.

In the conditions are met, the vehicle is assigned the best suitable parking space when it traverses the vehicle routing decision of the type **Parking lot**. The best suitable parking space is determined based on the attribute **Attraction**. If multiple parking spaces have the same attraction, Vissim choses one of them based on uniform distribution.

Vissim finds suitable parking space

If a parking lot only consists of a single parking space, a vehicle may park there if it is maximally as long as the parking space.

If the vehicle is longer than a parking space, Vissim checks whether two or more adjacent spaces are available to accommodate the vehicle. If a vehicle takes up all the parking spaces of a parking lot, the vehicle may park there if it is maximally as long as the total length of the parking spaces. If a vehicle does not take up all the parking spaces of a parking lot, the vehicle may park there if the total length of the parking spaces is at least 0.5 m longer than the vehicle. This ensures that there is sufficient space for pulling into and out of the parking space and the other adjacent parking spaces.

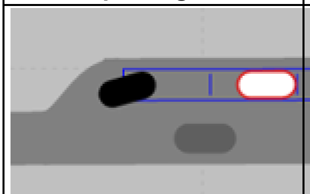
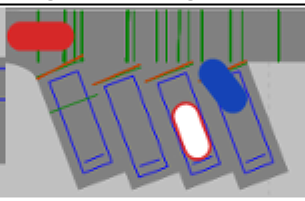
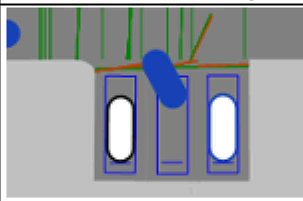
Parking spaces that are only partly used by a vehicle will not be assigned to another vehicle and are considered occupied. A vehicle always come to a standstill at the beginning of a parking space. If there are not enough adjacent parking spaces available for the long vehicle, it does not park, but continues on its route.

Vehicle routes to parking space

Vissim automatically generates vehicle routes that lead from the vehicle routing decision of the type **Parking lot** to each parking space of the corresponding parking lots. You cannot change these internal vehicle routes.

Displaying parking and parked vehicles

During the simulation, vehicle movements are displayed realistically for vehicles pulling into and out and in reverse of a parallel, diagonal or perpendicular parking lot. Once the vehicle has reached its final parking position in the parking space, in 2D mode, the vehicle inside is highlighted in white (by default). Examples:

Parallel parking lot	**Diagonal parking lot**	**Perpendicular parking lot**
		

Dwell time distribution defines parking duration

For the **Parking duration** attribute of the parking routing decision, you can select the desired time distribution. The parking duration begins when the vehicle is parked in the parking space in the direction of travel and the traffic on the adjacent lane, previously blocked, can flow again.

Assigning the vehicle a free parking space

If a parking space has not been assigned to a vehicle and no vehicle is parked in it, it is considered free. Vissim can then assign it to a vehicle. This is also the case, if the parking space can currently not be reached, as it is blocked by a vehicle, e.g. because of a traffic jam.

Occupying and blocking parking spaces

A parking space is considered occupied and is not assigned to another vehicle, if:

- It has already been assigned to a vehicle. The vehicle follows its route until it comes to a stop in the parking space. From this point on, the parked vehicle blocks the parking space.
- If because of its length, a vehicle is assigned several adjacent spaces, these are considered occupied. The vehicle follows its route until it comes to a stop in the parking spaces. From this point on, the parked vehicle blocks the parking spaces.
- The parking space is considered occupied and remains assigned to a vehicle, even if the vehicle has not yet reached it or access to it is blocked by another vehicle:
  - because the vehicle blocking the space is in a traffic jam
  - because the vehicle blocking the space is in a traffic jam adjacent to the parking space
- as soon as a parking space is reserved for a waiting vehicle, as all parking spaces that can be assigned according to routing decision type **Parking lot** are still occupied.

Leaving the parking space

After the dwell time, the vehicle leaves the parking space via an automatically generated route that guides it back to its original route.

> Note: Vissim determines the shortest path, in terms of time, to a position downstream of the parking lot routing decision. For generation of the route that takes the vehicle from the parking lot back to the route network, Vissim accounts for the entire route as well as for the current position of the vehicle on the parking lot it wants to leave. Thus, a vehicle might travel part of its original route twice or skip part of it.

Parking direction when pulling into and out of a parking space

The vehicle drives forward into the parking space. The vehicle cannot back into a parking space. The **Parking direction** attribute of the parking space defines whether the vehicle pull outs of a parking space forward or in reverse (see "Attributes of parking lots" on page 500). This allows the vehicle to pull into and out a parking space in the following parking directions:

- Pull into and out of parking space forward: For modeling parking in a parking lot that the vehicle can leave driving forward, e.g. a parallel parking lot, with several parking spaces parallel to the link.
- Pull into parking space forward, pull out in reverse: For modeling parking in a diagonal or latitudinal parking lot.. A connector leads to the link with the parking lot. The link ends downstream of the parking lot. Downstream there is no connector.

If the vehicle pulls out of the parking space in reverse, the following applies:

- The car following model can be interaction-free, as when driving forward, Wiedemann 74 model or Wiedemann 99.
- Visibility is based on the driving behavior parameters for forward driving. This allows Vissim to determine the distance and number of perceived vehicles. The distance is calculated starting from the rear edge of the vehicle. The vehicle pulling out of the parking space in reverse notices other vehicles on its route. If there is an oncoming vehicle, both vehicles can brake.
- The DesSpeed of the vehicle is based on the DesSpeed specified in the **Speed (reversing)** attribute of the parking lot.
- The vehicle does not change lanes.
- At the end of the parking duration, the vehicle pulls out of the parking lot in reverse and drives via the first upstream connector back onto its starting link. This may be a different link than the one the vehicle originally came from. The vehicle drives backwards until it has reached the link on which its route lies and on which it can continue driving forwards. It then comes to a standstill. Once the waiting time has passed, the vehicle continues driving forwards on its route that has been defined in the parking lot attribute **Direction change duration distribution** (see "Attributes of parking lots" on page 500). When driving forward, the vehicle drives at the DesSpeed valid before parking.
- Vissim Internally calculates the position at which the vehicle comes to a standstill, so that the vehicle can continue driving forward on its route from the next time step. This calculation is also based on the standstill distance.
- The **Travel direction** attribute of the vehicle is set to **Reverse**, e.g. in the **Vehicles in Network** list.
- The **Driving state** attribute of the vehicle is set to **Reversing**, e.g. in the **Vehicles in Network** list.
- The vehicle follows a route. The route uses the front edge of the vehicle as a reference point.
- Conflict areas do not respond to the vehicle, but to priority rules (see "Modeling priority rules" on page 541).
- At the start of a simulation run, Vissim generates priority rules for vehicles pulling out of a parking space in reverse:
  - To make vehicles pulling out of a parking space wait for a gap in traffic flow
  - To ensure that vehicles that wish to pull into a parking space wait to let vehicles pulling out of the parking space proceed. Vehicles that want to park in a parking space on the adjacent lane wait in their lane, approximately level to the center of the parking space, until a lane change becomes possible. This gives the vehicle that is still parked in the parking space more room to pull out.

Parking behavior on links with several lanes and parking lots

A vehicle only selects a parking space that is not occupied or blocked by another parking vehicle.

In the following cases, an available parking space is blocked by another vehicle that is either parked in the space or on a lane next to an available space:

- To the left and/or right of the parking space, there is a lane without a parking lot, on which a vehicle is parked next to the available parking space.
- To the left and/or right of the parking space, there is a lane with parking lots and another lane without parking lots. A vehicle is parked in the parking space between the available parking space and the lane without a parking lot.
- To the left and/or right of the parking space, there is a lane with parking lots and another lane without parking lots. There are two available parking spaces next to each other, but a vehicle is parked on the lane next to the two spaces.

As soon as a vehicle traverses the section of a vehicle routing decision of the type **Parking lot**, Vissim checks its vehicle routes and the corresponding parking lots. Vissim then finds and occupies a parking space for the vehicle. With every time step, Vissim checks for all vehicles that do not have the status **Waiting**, whether the occupied parking space is still accessible. If the parking space is still accessible and lies on a link with a single lane, or if the parking space is no longer accessible, Vissim will look for an accessible parking space that is available. This parking space must be located on a parking lot downstream of the current parking route of the vehicle.

If the parking space chosen for a vehicle is still available when the vehicle traverses the decision point, but is occupied right before it arrives there, the vehicle will choose another parking space. Precondition: There is still a parking space available further downstream, on the route of the parking routing decision.

If an attractive parking space becomes available after the vehicle has traversed the decision section, the vehicle drives towards the originally selected parking space as long as it is still accessible.

If the vehicle cannot park on any of the parking spaces because there are not enough spaces available and accessible, the vehicle routing decision of the type **Parking lot** is ignored. The vehicle continues driving on its original route.

Parking behavior with one lane and at least one occupied parking space

If on a lane with multiple parking spaces in a row, one or several of the spaces are occupied or reserved, the driver of a vehicle will choose a parking space further upstream that is located before the other available parking spaces. This is also true if the **Attraction** attribute suggests a parking space further downstream. With each time step, the driver of the vehicle checks whether in the meantime a more attractive parking space has become available and changes its destination accordingly.

Vehicle reserves parking space and waits until parking space is free

If a vehicle passes a vehicle routing decision of the type **Parking lot** and there is no free parking space, you can set the attribute **Full occupancy behavior** to specify what you want the vehicle to do in this case (see "Attributes of parking routing decisions" on page 474):

- continue driving
- drive to the next parking space that can be reserved, wait there until the parking space frees up and then park there

Requirements for a parking space to be reserved:

- The parking space is occupied, a vehicle is parked in it.
- The parking space has not been reserved.
- The length of the parking space is large enough to fit the length of the vehicle. If the vehicle is longer than the parking space and multiple adjacent spaces cannot be reserved, the vehicle continues driving.

On its way to a reserved parking space, the vehicle does not look for another free parking space.

The vehicle continues driving in the following cases:

- There is no free parking space available.
- No parking space can be reserved.
- All parking spaces are reserved.
- For the parking route decision, in the attribute **Full occupancy behavior**, **Drive On** is selected.
- There is no parking space that is long enough for the vehicle.
- Vissim Is unable to calculate the parking duration, as all parking spaces are occupied, but no vehicles are parked there yet, e.g. because they have not yet reached their assigned parking space or cannot reach it because another vehicle, stuck in a traffic jam, is blocking it.

6.11.4.2 Defining parking lots

You can define parking lots in a network editor on a link or connector:

- Parking lots with the attribute **Real parking spaces**, for parking or stopping on a lane (see "Modeling parking and stopping on the roadside" on page 493).
- Parking lots for dynamic assignment with the **Abstract parking lots** or **Zone connector** attribute (see "Modeling parking lots and zones" on page 698), (see "Defining parking lots for dynamic assignment" on page 700)

A parking lot can be modeled as a parallel, diagonal or perpendicular parking lot.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Parking Lots**.
2. With the mouse pointer, point to the position in the link at which the parking lot is to begin.

3. Hold down the CTRL key and the right mouse button, and drag the pointer to the desired end position.
4. Release the keys.

   *The parking lot is inserted. The **Parking Lot** window opens.*

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

5. Edit the attributes (see "Attributes of parking lots" on page 500).
6. Confirm with **OK**.

*The attributes are saved in the **Parking Lots** list.*

For individual, adjacent parallel parking lots, accordingly define long, adjacent links and connectors on which you can place individual parking lots of the desired length.

Note: To block traffic in a lane with parking vehicles, select the option **Keep lateral distance to vehicles on next lane(s)** in the driving behavior parameter set (see "Editing the driving behavior parameter Lateral behavior" on page 308).

6.11.4.3 Attributes of parking lots

The **Parking Lot** window opens when you insert a network object and have selected to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Parking Lots list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Basic attributes of parking lots

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

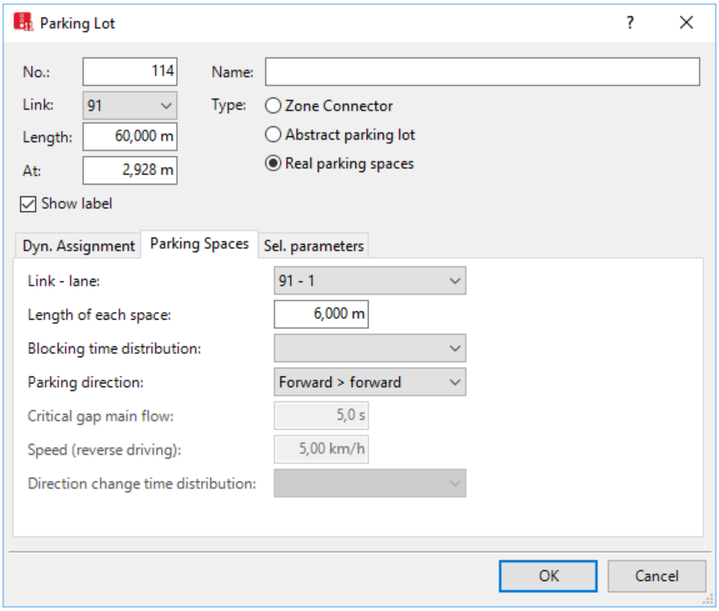
1. Make the desired changes:

Element	Description
No.	Unique number
Name	Designation
At	**Position** (**Pos**): Distance from start of the link or connector
Length	Length of the parking lot
Type	■ **Zone Connector**: only relevant for dynamic assignment (see "Modeling parking lots and zones" on page 698). Automatically creates a zone in the **Zones** list, if no zone has been defined. Automatically creates an origin zone and a destination zone in the **OD Pairs** list, if neither have been defined yet. ■ **Abstract parking lot**: only relevant for dynamic assignment ■ **Real parking spaces** relevant for simulation with and without dynamic assignment: modeling parking capacity in movement direction on one lane. Combined with vehicle routes of the type **Parking Lot**, you can realistically model parking maneuvers and stops at the roadside.
Showing label	☐ If the option is not selected, the label for the parking lot is not displayed, even if you selected labeling for all parking lots. For parking lot labels, the following are available: ■ Number ■ Name ■ Zone No. ■ Group No. ■ Occupancy ■ Current parking availability

Dyn. Assignment tab

These attributes are only relevant for parking lots of dynamic assignment (see "Defining parking lots for dynamic assignment" on page 700).

Parking Spaces tab



These attributes are only relevant for **Real parking spaces**.

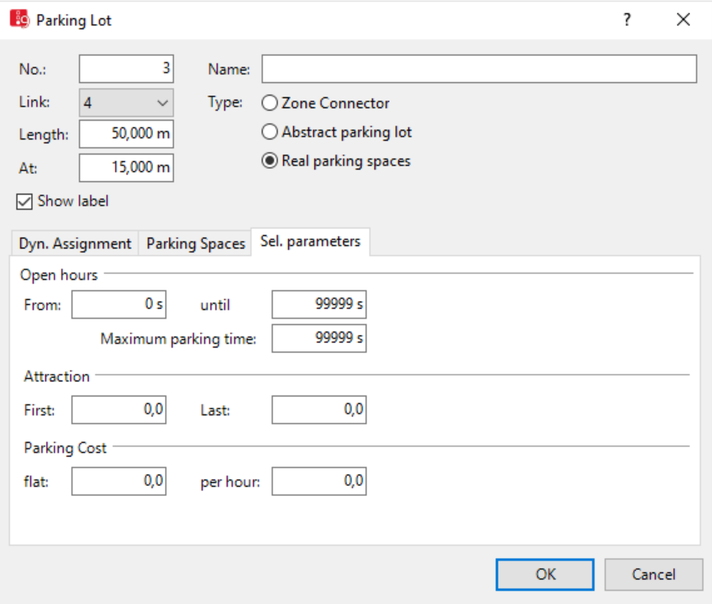
Element	Short name	Description
Link - lane		Number of the link and lane (**Ln**) on which the parking lot is located
Length per space	**LenPerSpc**	Length of a parking space The maximum length per space must not exceed the length of the parking lot. If the total length is not a multiple of the parking lot length, the remaining length is added to the end of the parking lot, but is not used by any vehicle.
Blocking time distribution	**BlockTmDistr**	Optional blocking time distribution: period during which a parallel-parking vehicle blocks the lane until it has completed its parking maneuver. **0: None**: Blocking time is not considered for simulation.
Parking direction	**ParkDir**	Only for real parking spaces: Direction in which the vehicle is driving into and out of the parking space.

Element	Short name	Description
Minimum gap time major flow	**MinGapTmMajFl**	Minimum gap time between two vehicles of the major flow, so that a vehicle may leave the parking space
Speed (reversing)	**SpeedRvs**	Desired speed of the vehicle for backing out of a parking space. Default value 5 km/h, value range 0.001 to 9999 km/h.
Direction change duration distribution	**DirChgDurDist**	Only relevant if the attribute parking direction **Forward > reverse** is selected: period during which the vehicle remains in standstill after it has pulled out of parking space in reverse until it drives forwards. Default: time distribution 5 s (see "Using time distributions" on page 246).. If no value is specified, the vehicle remains in standstill for the duration of a time step.

The blocking time begins when the parking vehicle first stops while heading into a parking space with the rear of the vehicle sticking out of the parking space, and it ends when the vehicle finally comes to a stop in the parallel-parked position.

- Blocking time distribution is considered for parking lots on the adjacent lane and thus includes parking maneuvers in parallel to the direction of travel.
- The parking lot must be longer than one length per parking space.
- The time distribution may be a normal or an empirical distribution (see "Defining time distributions" on page 246).
- If the parking lot is precisely as long as the parking space, the vehicle remains in parking position during the blocking time. In this case, blocking time distribution is ignored.

Sel. parameters tab



Element	Description
Open hours	**From** (**OpenFrom**), **to** (**OpenUntil**): Time span during which vehicles can enter the parking lot. Vehicles will not drive to the parking lot outside these hours.

Maximum parking time	**ParkTmMax**:
	- for **Real parking spaces**: Vehicles with a longer parking time are not assigned a parking space on this parking lot.
	- for **Zone connector** and **Abstract parking lot**:
	  - Only relevant when using a trip chain file: time span a vehicle may use this parking lot. If the maximum parking time is shorter than the minimum dwell time, the parking lot is not approached by the vehicle.
	  - Vehicles that are assigned a route via COM and vehicles moving based on an origin-destination matrix during dynamic assignment may select any destination parking lot open at the time of their departure, regardless of the attribute value **Maximum parking time:**. For these vehicles, for selection of a destination parking lot, a parking time of 1 s is assumed.

Element	Description
Attraction	**Attrac**: The higher the value, the more attractive the parking lot or parking space. This allows you to account for features of the parking lot that are not explicitly available as an attribute. For **Real parking spaces**, you can create a linear change in the attractiveness across the parking spaces by entering different values for **First** and **Last**. If you want to create a parking lot with attractive spaces in the middle or on the very right or left, create two symmetrical parking lots of the type **Real parking spaces**, with inverse **Attraction** values.
Parking Cost	**ParkFee**: only relevant for **Zone Connector** and **Abstract parking lot**: - **flat**: fee for one-time use of the parking lot, irrespective of the dwell time. - **per hour**: parking costs depending on the parking time. If a trip chain file is used, the minimum dwell time is considered. Without a trip chain file, an hour parking time is assumed for all parking.

The network object has additional attributes that you can show in the Attributes list. Among them are the following for example:

Short name	Long name	Description
DetBlock	**Detect blockage**	For real parking spaces only: - ☑ Select this option, if you want Vissim to check whether a parking space is blocked during the simulation. Vissim detects a vehicle blocking a parking space, without parking in it, e.g. when the vehicle parks in a space further upstream that lies on the same single lane link. The blocked parking space cannot be assigned to a vehicle traversing the parking routing decision. - ☐ If this option is not selected, Vissim will not check whether a parking space is blocked during the simulation. Vissim does then not recognize vehicles blocking a parking space, unless they are parking in it. The blocked parking space can be assigned to a vehicle traversing the parking routing decision. This results in fewer vehicles ignoring the routing decision due to a lack of available parking spaces. The option is selected by default.
Zone	**Zone**	Allocation of the zone number in the OD matrix to the parking lot. Multiple parking lots can belong to a zone.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Dyn assign des speed distributions (see "Defining parking lots for dynamic assignment" on page 700)
- Vehicles (parking): Attributes of the vehicles that are currently parked in the parking lot
- Paths: Paths of path file from dynamic assignment If no paths are shown and you have performed dynamic assignment, you can use its path file to read in the paths. To do so, from the List shortcut menu, choose the respective command (see "Attributes of paths" on page 752).

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.11.5 Using vehicle attribute decisions

You can use a vehicle attribute decision to set a vehicle attribute to the value of your choice once the vehicle traverses the cross-section of the vehicle attribute decision on a link or connector. The attribute value may also be based on a distribution defined in Vissim. You can transfer the attribute value set to a vehicle route. This allows you to select routes based on the attribute value (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459). You can restrict use of the attribute value to vehicle classes of your choice.

6.11.5.1 Defining vehicle attribute decisions

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the network objects toolbar, click  **Vehicle Attribute Decisions**.
2. Hold down the CTRL key and right-click the desired position of the vehicle attribute decision in the link or connector.

   *A dark red bar is inserted. The **Vehicle Attribute Decisions** list opens.*

3. Edit the attributes:

Short name	Long name	Description
No.	**Number**	Unique number of vehicle attribute decision
Name	**Name**	Name of the vehicle attribute decision
AllVehTypes	**All vehicle types**	☑ If the option is selected, all vehicle types account for the vehicle attribute decision. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	**Vehicle classes**	Vehicle classes for which the vehicle attribute decision applies
Attr	**Attribute**	Select the attribute of the vehicle for which you want to set a value or adopt a value from a distribution defined in Vissim. **Attribute** is independent from time intervals or vehicle classes. **Attribute** can be user-defined (see "Using user-defined attributes" on page 210).
DecTyp	**Decision type**	➤ **Value**: Activates the **Value** box, deactivates the **Distribution** box **(Distr**). ➤ **Distribution**: Activates the **Distribution** box **(Distr**), deactivates the **Value** box.
Value	**Value**	Value to which the **attribute** is set. Only active if in the **Decision Type** attribute. **Value** is selected.
Distr	**Distribution**	Distribution defined in Vissim to which the **attribute** is set. Only active if in the **Decision type** attribute, **Distribution** is selected.
FromTime	**From time**	Start of time interval in seconds for which the vehicle attribute decision is valid
ToTime	**To time**	End of time interval in seconds for which the vehicle attribute decision is valid
Link	**Link**	Number of the link on which the vehicle attribute decision has been positioned
Pos	**Position**	Distance of vehicle attribute decision from beginning of link

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Vehicle classes**

2. On the list toolbar, in the **Relations** list, click the desired entry.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

3. Enter the desired attribute values in the right-hand list (see "Defining the vehicle class" on page 280).

The data is allocated.

6.11.6 Modeling overtaking maneuvers on the lane of oncoming traffic

You can model overtaking maneuvers during which the overtaking vehicles use the lane of oncoming traffic. This for instance allows you to simulate the overtaking of PuT vehicles temporarily stopping in an urban area, on a link with a single lane only, or the overtaking of slower vehicles outside of the city. You can model multiple vehicles overtaking and multiple vehicles being overtaken. The overtaking vehicle shall only then perform the overtaking maneuver, if there is no oncoming traffic which excludes the option of overtaking.

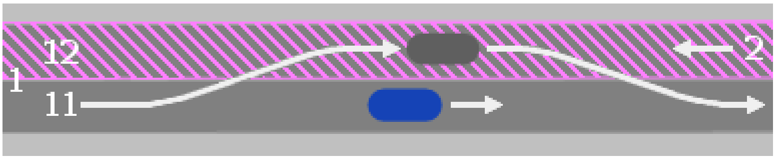
In Vissim a vehicle wants to overtake when it's desired speed is considerably higher than that of vehicle preceding it (0 m/s to 4 m/s, normally distributed, around mean value 2 m/s). The vehicle then checks the following requirements in order to be able to perform the overtaking maneuver:

- At overtaking speed, it would take the vehicle at most 4 seconds to drive into its preceding vehicle. The overtake speed is the desired speed plus the attribute **overtaking speed factor** (**OvtSpeedFact**).
- Its link has a passing lane (see "Attributes of links" on page 409).
- The passing lane must overlap with the inner lane of the oncoming lane. The width of the overlapping area is not relevant. The z value of the oncoming lane and of the passing lane must not deviate more than 0.5 m from each other.
- The overtaking area, which consists of links and connectors that overlap with the oncoming lane, must be at least long enough for a vehicle to be able to perform an overtaking maneuver.
- Downstream of the last vehicle that is to be overtaken, there must be a large enough gap visible, so that the passing vehicle is able to change back into its previous lane after completing the overtaking maneuver. The largest gap must be at least one passing vehicle length, plus double the safety distance from the front and rear of the vehicle, as required during standard car following behavior.
- The overtaking maneuver must be able to be completed safely, without endangering oncoming traffic. The vehicle recognizes oncoming traffic on the opposite lane from a certain distance. This distance is defined for the link of the opposite lane in its attribute **Look ahead distance for overtaking** (**LookAheadDistOvt**). The vehicle that wants to overtake safely assumes that anywhere beyond this distance there will be oncoming traffic. To assess the course of the trip of oncoming traffic, Vissim uses the attribute **Assumed speed of oncoming traffic** (**AssumSpeedOncom**) of the oncoming lane (see "Attributes of links"

on page 409). Vissim does not account for any additional vehicles following the oncoming vehicle.

The figure below shows the following in 2D mode:

- a section of a link **1** that runs from left to right via a lane **11** and a passing lane **12** (hatched) with right-hand traffic running from left to right. Vissim displays passing lanes with hatched lines as soon as its link has at least two lanes and the attribute **Has passing lane** is selected.
- a section of a link **2** that has only one lane in right-hand traffic, running from right to left. Link **2** was created as an oncoming lane of link **1** and fully overlaps with passing lane **12**.
- For overtaking maneuvers on link **1**, vehicles driving from the left to the right use passing lane **12**.
- Vehicles on link **2**, driving from the right to the left, cannot be overtaken, as link **2** does not have a passing lane.
- The gray vehicle that is overtaking on passing lane **12** recognizes that there is no vehicle in the opposite lane within the distance defined in the attribute **Look ahead distance for overtaking** (**LookAheadDistOvt**) of link **2**.



The overtaking vehicle reduces its speed to the desired speed, as soon as it leaves the passing lane.

An overtaking vehicle may cancel its overtaking maneuver, if downstream of the vehicle it is overtaking, the gap in its previous lane is not yet large enough to change back into it. This may be the case when there is suddenly unexpected oncoming traffic or the vehicle that is being overtaken accelerates. When a vehicle cancels an overtaking maneuver and wants to change back into its lane, the vehicles behind it can reduce their speed to allow for the lane change.

The driving behavior of oncoming traffic is not influenced through the overtaking maneuver.

When the length of the overtaking areas is checked, this is done along the current route or path of the vehicle. This way, vehicles without a current vehicle route or path cannot overtake on the oncoming lane.

The vehicle width is not considered a criterion for the option of an overtaking maneuver. It is generally assumed that there is not enough space on the oncoming lane for oncoming traffic and a passing vehicle to drive next to each other .

Before an overtaking maneuver takes place, Vissim does not account for network objects on the oncoming lane, e.g. reduced speed areas, SCs or stop signs. However, as soon as the overtaking maneuver starts, the passing vehicle reacts to these network objects.

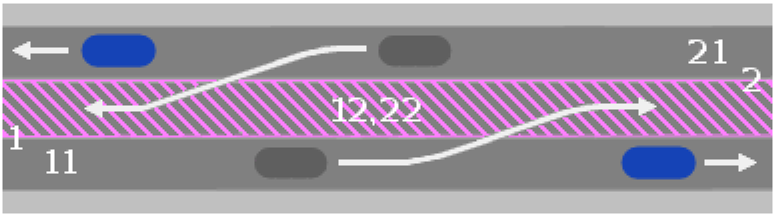
When a vehicle changes from a link with a passing lane to a connector without a passing lane, the connector then has one lane less than the link with a passing lane. In this case, the distance defined in the attribute **Lane change distance** (**LnChgDist**) also specifies the minimum distance to the connector at which an overtaking maneuver may be started. Passing on the oncoming lane is not possible within this lane change distance.

Passing on oncoming lane for both movement directions

You can model overtaking maneuvers for both movement directions. In this case, an additional passing lane is placed on top of the inner oncoming lane. This means the passing vehicles of both movement directions use the oncoming lane and need to account for any oncoming traffic.

The figure below shows the following in 2D mode:

- a section of a link **1** that runs from left to right via a lane **11** and a passing lane **12** (hatched).
- a section of a link **2** that runs from left to right via a lane **21** and a passing lane **22** (hatched).
- Links **1** and **2** are placed so that their passing lanes **12** and **22** each overlap with the lane of oncoming traffic, so that vehicles can pass in both movement directions.



6.11.6.1 Defining network objects and attributes for passing on oncoming lane

1. Make sure that the desired speed and acceleration you have defined for vehicles allows for passing.
2. In the Network Editor, for the vehicles that shall be overtaken, define at least one link with normal lanes, but without a passing lane (see "Defining links" on page 407).
3. Generate a link for the opposite direction (see "Generating an opposite lane" on page 419).
4. For the first link, specify the following attributes (see "Attributes of links" on page 409):
5. Increase the value for the attribute **Number of lanes** (**NumLanes**) by 1.

   The number of lanes includes the passing lane.
6. Select ☑ **Has passing lane** (**HasOvtLn**).
7. Make the desired settings (see "Attributes of links" on page 409).
   - **Overtake speed factor** (**OvtSpeedFact**)
   - **Look ahead distance for passing** (**LookAheadDistOvt**)

- **Assumed speed of oncoming traffic** (**AssumSpeedOncom**)
- **Overtake only PT** (**NurÖVÜberh**): must be deactivated

8. Deactivate the attribute **No lane change left – All vehicle types** (**NoLnChLAllVehTypes**) for the lane the vehicle will change back to after the overtaking maneuver.
9. Make sure that for the passing lane, the attribute **Blocked vehicle classes** (**BlockedVehClasses**) does not contain vehicle classes that are meant to do the passing.
10. Specify the following attributes for the overlapping link(s) (see "Attributes of links" on page 409):
    - **Look ahead distance for passing** (**LookAheadDistOvt**)
    - **Assumed speed of oncoming traffic** (**AssumSpeedOncom**)
11. Define vehicle inputs or create vehicles with the add-on module Dynamic Assignment (see "Defining vehicle inputs" on page 456), (see "Using the dynamic assignment add-on module" on page 692).

    For overtaking on the lane of oncoming traffic, the vehicle must be on a vehicle route or a path of dynamic assignment.
12. If you have defined vehicle inputs, define vehicle routes that lead via links on which vehicles shall be overtaken (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459).

    If you use dynamic assignment, Vissim will distribute the vehicles across the paths. Vissim Saves the paths to the path file (see "Using the dynamic assignment add-on module" on page 692), (see "Files of dynamic assignment" on page 1195).

6.12 Modeling short-range public transportation

Vehicles of short-range public transportation (PT) can use the links of the private transportation or separate links, for example, lanes for buses or tracks for trains.

First define the stops and then the PT line that you assign a route, operated stops, PT vehicles, and a timetable.

6.12.1 Modeling PT stops

You can construct public transport stops on a link or next to a lane.

Types of PT stops

- **PT stop**: Stop of a PT vehicle on a lane of a link. Select the lane.
- **Public transport stop bay**: Stop of a PT vehicle on a specific link in the movement direction to the right, next to the lane.

Behavior of vehicles on multi-lane links

For multiple-lane links, following vehicles try to overtake PT vehicles, which are stopped at a public transport stop.

Behavior of vehicles on single-lane links

If a PT vehicle stops at a stop on a single-lane link, the vehicles behind it will wait until the PT vehicle continues its journey. With regard to exiting the public transport stop bay, the PT vehicle has the right of way. This is in accordance with the German Traffic Code (StVO) and the rules in road traffic of other countries.

Behavior of PT vehicles also depends on the length of the public transport stop

While a PT vehicle is parked at a PT stop waiting for passengers to alight and board, it can only be overtaken by another PT vehicle, moving downstream, looking for a free parking space at the same PT stop to also allow passengers to alight and board. The PT stop must be sufficiently long.

Emergency stop position at PT stop

The emergency stop position depends on the number of the lane the PT stop is on:

- Even lane number: 1.6 m before the end of the PT stop
- Odd lane number: 2.1 m before the end of the PT stop

6.12.2 Defining PT stops

Before modeling, you must define whether or not the passengers at the public transport stop should be taken into consideration in the simulation:

- Without passengers: Select a pre-defined dwell time distribution per public transport stop and PT line.
- Boarding passenger profile: Allocate **Volumes** as hourly values to the lines on the public transport stop. Select the **PT parameters** per vehicle type.
- Viswalk for microscopic pedestrian simulations: Define at least one pedestrian area with the attributes **PT usage** - **Waiting area** or **PT usage** - **Platform edge** at the stop.

Parameterize the public transport stops according to the method of your choice (see "Calculating the public transport dwell time for PT lines and partial PT routes" on page 531). In the network you can combine public transport stops of these three methods with each other. Then the parameterization of a public transport stop applies to all lines which operate at this public transport stop.

- For each defined public transport stop, you can set specific PT line stop parameters for each PT line and PT partial route (see "Modeling PT lines" on page 518).
- For the microscopic pedestrian simulation with Viswalk, the length of the public transport stop must correspond with at least the length of the longest PT vehicle, which operates at this public transport stop. Doors of the vehicle, which at the time of the stop are not positioned at the platform edge, are not used by the passengers.
- You can also define a public transport stop, in which boarding and alighting by multiple PT vehicles takes place at the same time. For this to occur, the length of the public transport stop must be correspondingly defined: at least the sum of all vehicle lengths must

have enough space, behind and between the vehicles, which operate simultaneously at the public transport stop.

- On a multiple lane link, the PT vehicles can enter or exit when there is a large enough gap.
- On a single lane link, for example for bus bays, a following vehicle can only exit once the preceding vehicle has exited.
- If you move PT stops that a PT line uses, these PT stops are shown as passive, in green.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Public transport stops**.
2. With the mouse pointer, point to the position in the link at which the public transport stop is to begin.
3. Hold down the CTRL key and the right mouse button, and drag the mouse pointer to the desired end position.
4. Release the keys.

   *The public transport stop is inserted. The **PT Stop** window opens.*

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

5. Edit the attributes (see "Attributes of PT stops" on page 513).
6. Confirm with **OK**.

*The attributes are saved in the list **Public transport stops**.*

6.12.3 Attributes of PT stops

The **PT Stops** window opens when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the PT stops list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.

- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

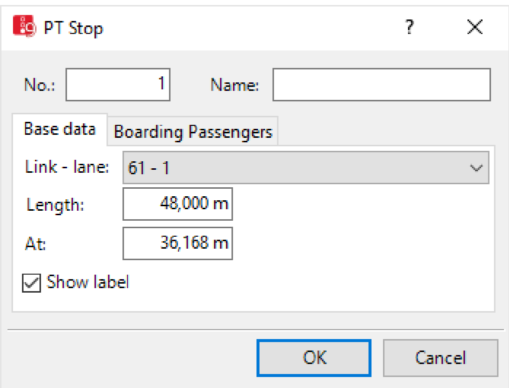
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



6.12.3.1 Basic attributes of PT stops

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number
Name	Description

6.12.3.2 Base data tab

Element	Description
Length	**Length**: Length of the public transport stop [m]
Lane	**Ln**: Lane of the link or connector on which the public transport stop is located.
At	**Position** (**Pos**): Start of the public transport stop on the link or the connector
Label	☐ If the option is not selected, the label for an individual public transport stop is not displayed when the label for all public transport stops is selected.

6.12.3.3 Boarding Passengers tab

You can set boarding passenger profiles for a public transport stop. The data is used for:

- calculation of stop dwell times
- proportional distribution of pedestrians on public transport lines in Viswalk

1. Right-click in the list.
2. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

The list contains, amongst others, the following attributes:

Element	Description
Volume	**Volume**: Passenger volume of the category pedestrians per hour for all or selected PT lines
TimeFrom	**Time from**: Time from the start of the time interval for which this number of boarding passengers is valid in simulation seconds.
TimeTo	**Time to**: Time till the end of the time interval for which this number of boarding passengers is valid in simulation seconds.
PTLines	**PT lines** which may be used by passengers of this boarding passenger profile
AllPTLines	☑ **All PT lines**: If the option is selected, all PT lines take the PT stop into consideration.

Note:

- For a PT stop with **Platform edge**, enter the **relative volume** of the public transport stop instead of the **volume** in [P/h].
- For passengers, who are Viswalk pedestrians, defaults are generated in two cases:
  - For each automatically generated platform edge.
  - If a pedestrian area (either a **Waiting area** or a **Platform edge**) of a public transport stop is allocated, for which no vehicle volume has been defined.
- With regard to the defaults, each pedestrian, who reaches the waiting area in the time interval of 0 - 99,999, boards each PT vehicle of a preferred PT line.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Boarding volumes: The attributes are described further above.
- Vehicles: Vehicles in network (see "Displaying vehicles in the network in a list" on page 847)
- Areas (PT usage) (see "Attributes of areas" on page 898)
- Public transport lines (active) (see "Attributes of PT lines" on page 520)
- Public transport line stops (see "Editing a PT line stop" on page 526)
- Partial PT routes (active) (see "Attributes of partial PT routes" on page 540)
- Partial PT route line stops (see "Editing a PT line stop" on page 526)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.12.4 Generating platform edges

Notes:

- You require the add-on module Viswalk for microscopic pedestrian simulation.
- Platform edges are not automatically moved along with the link or the public transport stop nor are they adjusted to changes in length. Therefore, generate the platform edges once the links and PT stops will no longer be changed. In this way you prevent a transition type between the network and the PuT vehicle from being guaranteed.

You can define areas of the type **Polygon** and **Rectangle** as a **Platform edge**. For a PT stop, you can have an area of the type **Rectangle** automatically generated as a **Platform edge**. An automatically generated **Platform edge** follows the user-defined course of the road. Platform edges are created with a width of 2 m immediately next to the link.

If boarding passengers are to board and/or alight from both sides, add platform edges to the left and the right.

Notes:

- When generating platform edges, the volume **[Pers./h]** is changed in the attribute **Boarding passengers** in a **relative volume** and existing values are taken over. Every public transport stop with an area for **Public transport usage** is used by the passengers, who are generated as pedestrians. These passengers are proportionately distributed among the PT lines.
- If there are no pedestrian areas with the attribute **Public transport usage**, the passenger volume is distributed over the PT stops exactly according to the absolute values.

- The **Public transport usage** attribute can be used to define a pedestrian area as a **Platform edge** or a **Waiting area** for one or more selected stops (see "Modeling construction elements" on page 880). Based on its type, you determine the shape of the platform edge using the shape of a polygon or the rectangle of the pedestrian area.
- A pedestrian area with the attribute **Public transport usage** defined as a **Platform edge** must fulfill the following conditions:
  - The platform edge and the lane with the public transport stop must be directly adjacent or overlap so that alighting passengers may alight on the platform edge and boarding passengers may board the PT vehicle from the platform edge. The overlapping must be smaller than half of the width of the pedestrian area. This means that the center line of the platform edge, which is parallel to the lane and thus to the public transport stop, must lie outside the lane.
  - The length of the pedestrian area parallel to the lane must be at least the length of the public transport stop, so that when the public transport vehicle stops, no vehicle door lies outside of the platform edge.
  - If the area for a platform edge is not automatically generated and you do not edit it later on in the Network editor, ensure that the maximum distance between the lane on

which the PT vehicle stops and the area of the platform edge is 2 m. Otherwise, the PT vehicle will not open its doors.

Note: When two platform edges with different parameters overlap, a warning is displayed during the check. Even so, Vissim assigns the platform edges to the right doors and the simulation is not interrupted.

1. On the Network objects toolbar, click **Public transport stops**.
2. In the Network editor, right-click the desired PT stop.
3. In the context menu, select the entry **Add platform edge left** or **Add platform edge right**.

   The platform edge is displayed in the Network editor next to the link. Default color: pink. All relevant area attributes are set automatically, e.g. for public transport usage ***Platform edge*** *and* ***for public transport stop(s)****, the number of the PT stop selected. The name of the pedestrian area includes numbers and sides of the PT stop, e.g.* ***platform edge PT stop 1 (right)****. The name is not automatically adjusted when the area of the platform edge is allocated a later point in time of another public transport stop.*

6.12.5 Generating a public transport stop bay

Note: You require the add-on module Viswalk for the microscopic pedestrian simulation.

1. On the Network objects toolbar, click **Public Transport Stops**.
2. In the Network editor, right-click the desired PT stop.
3. In the context menu, select the entry **Create lay-by stop**.

   The lay-by stop is displayed in the Network editor, next to the PT stop in the movement direction. Connectors and conflict areas are added to the modeling for the right of way of buses (see "Modeling PT stops" on page 511). By default, all vehicle types may use the public transport stop bay.

6.12.6 Modeling PT lines

You can define PT lines for buses or trains, which operate in a fixed public transport stop order. For this, they must have a timetable with departure times at the first public transport stop, and offset times according to the time table between public transport stops and dwell times at public transport stops.

Public transport stop dwell times are either obtained from the dwell time distribution or are calculated on the basis of boarding passenger profiles. They may also be determined via microscopic pedestrian simulation. Before modeling, decide whether and to what extent the passengers in the simulation should be taken into consideration and parameterize the public transport stops accordingly (see "Calculating the public transport dwell time for PT lines and partial PT routes" on page 531). In the network model, you can combine public transport stops of these three models with each other. However, the selection per public transport stop applies to all lines which operate at this public transport stop (see "Modeling PT stops" on page 511).

For each public transport stop, you can enter specific PT stop parameters for each PT line and partial PT route (see "Attributes of PT stops" on page 513).

A public transport line in Vissim always has a fixed route. In the case that a real PT line should drive on different routes within the Vissim network, then multiple, separate PT lines must be modeled in the Vissim network.

Modeling PT lines may be compared to the modeling of static routes. However, PT lines do not distribute incoming vehicles; instead they generate them. Start points of PT lines cannot be moved.

Note: PT vehicles drive the route, which is specified by the PT line. By default, they stay within the Vissim network afterwards. In order that PT vehicles do not stay within the network and therefore do not move within the network without routes, model the PT lines in a way that they must move out of the network at the end of the link.

6.12.6.1 Defining PT lines

Define all public transport stops for the line before defining a PT line (see "Modeling PT stops" on page 511).

Note: For every PT line, define a link, which will only be used from this PT line.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Public Transport Lines**.
2. Hold down the CTRL key and right-click the desired link on which the vehicles of the line are to drive into the network.

   A blue decision marker is set by default at the beginning of the link, on which the line begins.
3. In the desired link, point with the mouse pointer at the desired position of the destination section, at which the public transport line should end.

   If no connection between the start section and the destination section exists, Vissim still displays the line path. In this case, you must either correct the destination link and/or the destination section or the Vissimnetwork.

If there is a connector is between a start section and a destination section, it is displayed as a colored band in the PT line fill color over a continuous link curvature. A turquoise bar (default color) shows the possible position of the destination section.

4. Click this position.

   *The **PT Line** window opens. The public transport line is marked in the Network Editor.*

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

5. Edit the attributes (see "Attributes of PT lines" on page 520).
6. Confirm with **OK**.

*The attributes are saved in the list **Public Transport Lines**.*

If in the **Public Transport Lines** list, on the toolbar, you click the **Synchronization** button, then in the list, click a public transport line, this line is by default displayed as a colored band in the PT line fill color, with the active stops in red and passive stops in green (default color setting). By default, all PT public transport stops located directly on the line path are highlighted as active in red.

The line path of a new PT line does not automatically include public transport stop bays. You can change the line path of a PT line, so that it does include a public transport stop bay (see "Entering a public transport stop bay in a PT line path" on page 525). If you modify the line path, PT stops on the new line path are highlighted as passive, in green. Even if you move these PT stops, they are still highlighted as passive, in green.

6.12.6.2 Attributes of PT lines

The **PT Line** window opens automatically when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Public Transport Lines list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

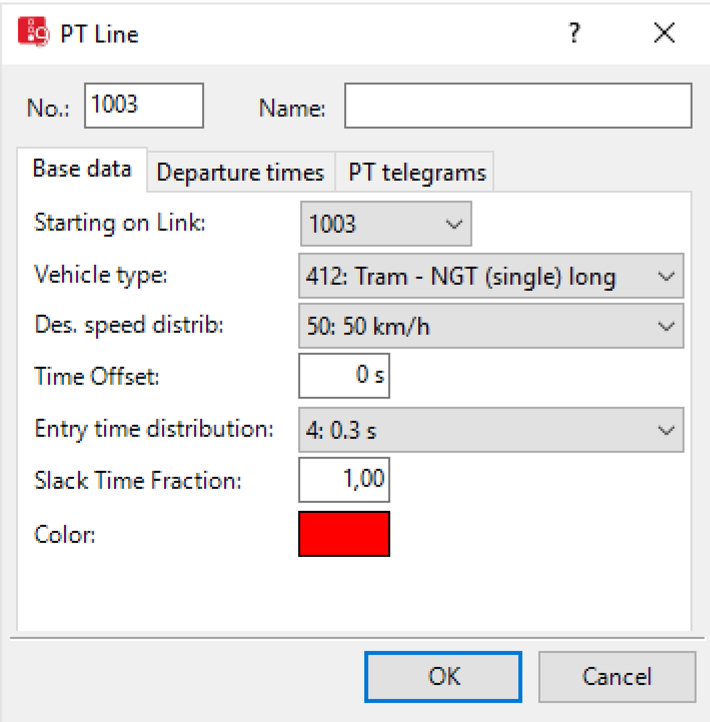
- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).

- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



Basic attributes of PT lines

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number 1 to 4294967295
Name	Description

Base data tab

Element	Description
Starting on Link	**Entry link EntryLink**: Link on which the PT line begins
Vehicle type	**VehType**: Vehicle type of PT line
Desired speed distribution	**DesSpeedDistr**: Initial speed of PT vehicle
Time Offset	**Entry time offset** (**EntTmOffset**): Time PT vehicles need to enter the network before their scheduled departure time in order to depart punctually at the departure times scheduled, from the first stop serviced. This means you enter the departure times of your timetable as departure times of the PT line. The time offset is the sum of the time the vehicle requires to get to its first stop in the network and the average passenger interchange time at this stop. The resulting network entry time is always set to zero, if the departure time is smaller than the time offset. If the time offset is 0 s, the vehicles of the PT line enter the Vissim network precisely at their defined departure time.
Start time distribution	**Entry time distribution** (**EntryTmDistr**):Time distribution for variation of departure time at which the vehicle enters the network (see "Defining time distributions" on page 246). This value is added as a delay to the calculated entry time. $\textit{Entry time} = \textit{departure time} - \textit{time offset}$
Slack Time Fraction	**SlackTmFrac**: only relevant for stops with specified departure time: factor for wait time of PT vehicle as part of the remaining time until scheduled departure. Value range 0.00 to 1.00. • Slack time fraction = 1: Earliest departure time is according to the timetable (see "Attributes of PT lines" on page 520) • Slack time fraction < 1: Departure time may be earlier than fixed in the timetable. Departure time is based solely on arrival time and dwell time. This allows you to model earliness as well.
Color	Define the color of PT vehicle of this line

Departure times tab

You may define individual trips of the line as courses. When you enter departure times, individual trips (**New**) and departures based on service frequency rate (**Rate**) can be mixed. In both cases, you may in addition to the time also specify a course number and occupancy rate.

The list contains, amongst others, the following attributes:

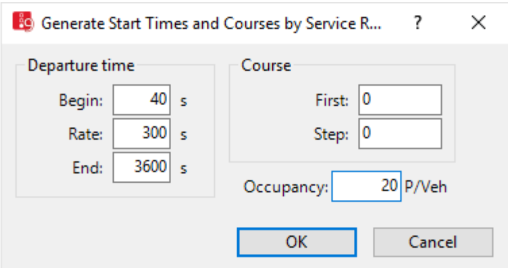
Element	Description
Dep	Departure time of a public transport vehicle relative to the start time of the simulation
TeleCour	**PT telegram - course**: Optional course number. If PT calling points are defined for the network, the course number serves to perform evaluations of serial telegrams in terms of the course number.
Occup	**Occupancy**: Number of passengers

Generate start times

You may define several departure times simultaneously.

1. In the **Departure Times** tab, right-click into the table.
2. From the shortcut menu, choose **Departure Times**.

   The window ***Generate Start Times and Courses by Service Rate*** *opens.*



3. Make the desired changes:

Element	Description
Departure time	First departure in timetable of service frequency ▸ **Begin**: first departure ▸ **Rate**: service frequency rate. Vissim generates all departures defined this way as individual trips. You may also define multiple service frequency rates in a row. ▸ **End**: last departure
Course	Optional course ▸ **First**: first course number. Is displayed in the **Departure Times** tab. ▸ **Step**: increment for all following course numbers. The departure times are sorted in chronological order.

Element	Description
Occupancy	Number of passenger in PT vehicle when entering the Vissim network. Is displayed in the **Departure Times** tab.

PT Telegrams tab

You can define data that is transmitted via PT telegrams to control procedures, when vehicles pass PT calling points (see "Using detectors" on page 593).

Element	Description
Line sends PT telegrams	**Send PT telegrams** (**SendTele**): ☑ Select this option if you want vehicles of this PT line to be recorded by PT calling points.
Line	**PT Telegrams - Line** (**TeleLine**): Number of PT line, max. 999 999 999
Route	**PT Telegrams - Route** (**TeleRout**): Number of PT line path, max. 999 999 999
Priority	**PT telegram - priority** (**TelePrio**): Priority of PT vehicle [1 to 7]
Tram Length	**PT telegram - tram length** (**TeleVehLen**): Length of PT vehicle [1 to 7]
Manual Direction	**PT telegram - manual direction** (**TeleManDir**): Direction from which the vehicle is coming, if the PT calling point cannot clearly identify this via line or route number.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Departure times: The attributes are described further above.
- Line stops (see "Editing a PT line stop" on page 526)
- Link sequence: Numbers of links and connectors via which the PT line leads (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.12.6.3 Checking and repairing PT line courses

When you split or delete links and connectors that run across PT lines, these PT lines are disconnected. Even if you then insert new links or connectors there, these PT lines remain disconnected. You can have disconnected PT lines repaired by Vissim.

1. On the Network objects toolbar, click **Public Transport Lines**.
2. Right click into the Network editor.
3. From the context menu, choose **Check and repair all PT line courses**.

Vissim will find new PT lines for disconnected PT lines. These may run via the added or other links and connectors in the network.

6.12.7 Entering a public transport stop bay in a PT line path

1. Enter a public transport stop bay in the link (see "Generating a public transport stop bay" on page 518).
2. Select **Public Transport** > **PT Lines** from the **Lists** menu.
3. Select the desired entry.

   In the Network Editor, the selected PT line is shown as a yellow band; active public transport stops red; passive public transport stops green. By default, all PT public transport stops located directly on the line path are highlighted as active in red. Public transport stop bays are not automatically in the line path of a new PT line.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

4. Press the CTRL key, and in the Network Editor right-click next to the stop bay in the yellow PT line.

   A point is added to the PT line.

5. Click on the point, hold the mouse button down, and drag the point into the stop bay.

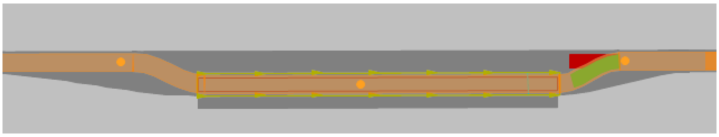
   The line path runs over the stop.

Moving a specific PT line path section

- In the section you wish to change, add three points at the following positions:
  - Position of the first point from which the new PT line path shall be recalculated

- Position of the middle point you want to move to the public transport stop bay
- Position of the third point up to which the new PT line path shall be recalculated

When you move the middle point, only the path between the two outer points is recalculated, and the PT line path is only changed for this area.



6.12.8 Editing a PT line stop

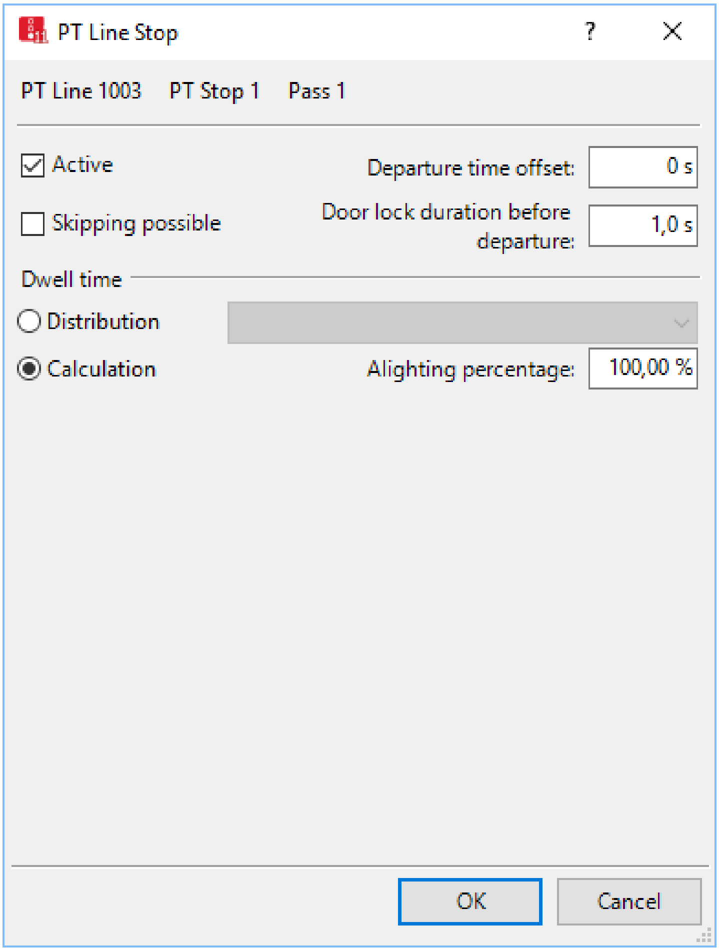
You can edit specific attributes of PT stops for a PT line or partial PT route in the **Edit PT Line Stop** window.

Note: Alternatively to in the **PT Line Stop** window, you may also show and edit PT line stop attributes in the PT line stops list (as a relation of a PT stop attribute list).

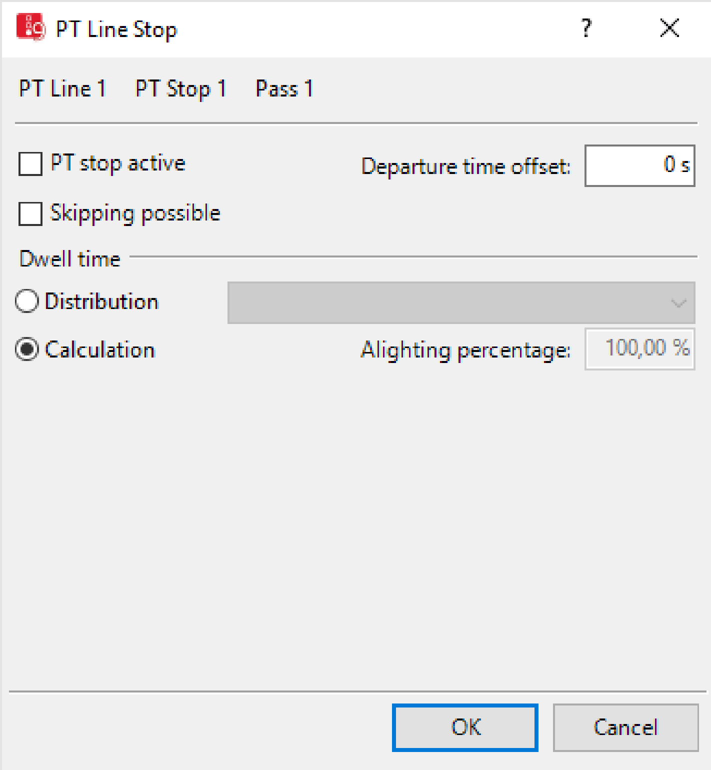
1. On the Network objects toolbar, click **Public Transport Stops** or **Public Transport Lines**.
2. In the network editor, right-click the PT stop.
3. If a public transport stop is serviced by one PT line only, from the shortcut menu, choose Edit **PT Line <No. > PT Stop <No.> Pass<No.>**.
4. If a public transport stop is serviced by multiple PT lines, from the shortcut menu, choose **Edit PT Line Stop**. Then select the desired line stop > **PT Line <No. > PT stop <No.> Pass <No.>**.

   *A **PT Line Stop** window opens with attributes that depend on the network object the PT line stop refers to:*

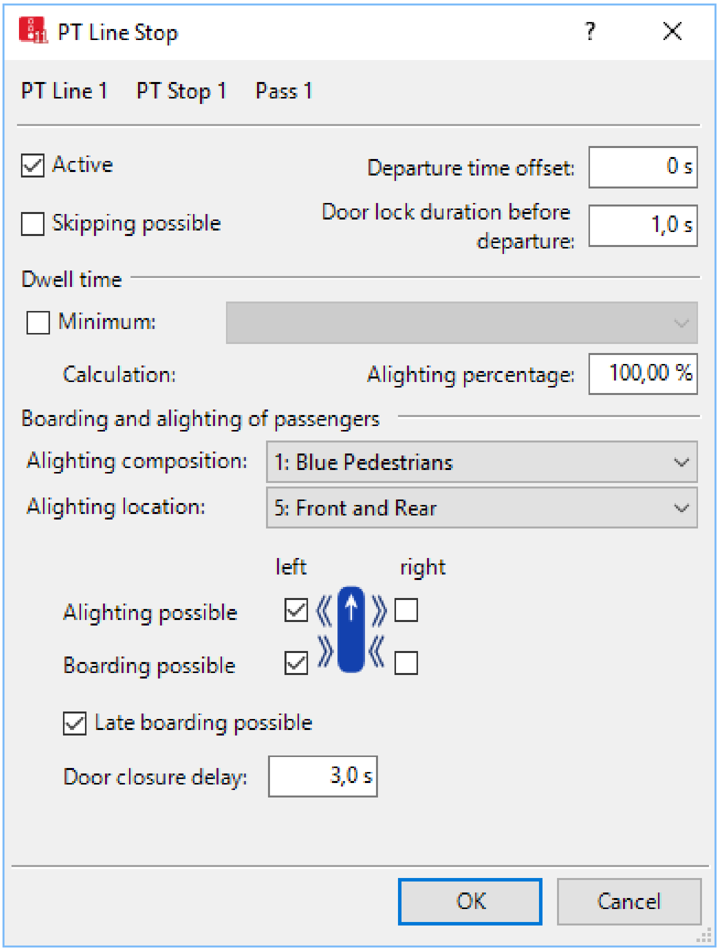
- For PT stops of a PT line, the number of the line is shown:



- For the PT stops of a partial PT route, the number of the routing decision and the number of the route are shown:



- For a PT stop in Viswalk, additional attributes are displayed. These settings allow yo to simulate passengers boarding and alighting at this stop with Viswalk:
  - The PT stop must be selected in the **For PT stop(s)** attribute of the area you want to use as a waiting area or platform edge.
  - In the **Public Transport Usage** attribute of the area, **Platform edge** or **Waiting area** must be selected.
  - The attribute **PT stop active** must be selected.



Determining public transport dwell time

The public transport dwell time for PT lines and PT routes can be determined via the following methods:

- Method **Dwell time distribution**: (see "Defining dwell time according to dwell time distribution" on page 532)
- Method **Advanced passenger model**: (see "Calculating dwell time according to the advanced passenger model" on page 533)
- Method **VISWALK: microscopic pedestrian simulation**: (see "Calculating dwell time with PTV Viswalk" on page 534)

You can use different methods at different PT stops. However, only one method is used for PT lines or PT partial routes at a PT stop.

Showing PT line stop attributes as a relation of a PT stop

1. From the **Lists** menu, choose > **Public Transport** > **Public transport stops**.

   The list with the attributes is displayed as the left list of two coupled lists (see "Using coupled lists" on page 119).

2. On the list toolbar, in the **Relations** list, click **Public transport line stops**.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

Column	Description
PTLine	Name of PT line that services the PT line stop.
Public transport stop	Number and name of public transport stop of the PT line stop
Active	☑ If this option is selected, the public transport stop is a PT line stop of the PT line. A public transport stop created in the Vissim network according to the definition of PT lines is not automatically contained in a PT line path, even when placed on a link that is traversed by a PT line. Non-serviced PT stops are displayed in green and the **Active** attribute is disabled. You can enable the **Active** attribute to include the public transport stop into the PT line path. If a PT stop is not to be serviced by a PT line, disable its **Active** attribute.
SkipPoss	**Skipping possible**: ☑ If this option is selected, the PT stop is not serviced, depending on the method used for calculating the public transport dwell time (see "Defining dwell time according to dwell time distribution" on page 532), (see "Calculating dwell time according to the advanced passenger model" on page 533), (see "Calculating dwell time with PTV Viswalk" on page 534)
DepOffset	**Departure time offset:**: Define departure time according to timetable. The time offset is used additionally for boarding and alighting time (boarding/alighting). The resulting departure time is calculated as follows: *Simulation second of arrival + dwell time + max (0, ((departure time of PT line + departure offset) - (simulation second of arrival + dwell time)) • slack time fraction of PT line)* If the departure time, according to the time table, is later than the point in time, which is the sum of the arrival time and dwell time, the PT vehicle waits until the departure time, according to the time table if the Slack time fraction equals 1. For slack time fractions < 1, the vehicle correspondingly departs earlier, value range 0.00 to 1.00. If the **Slack time fraction** of the line is 0 or the **Departure time offset** of the public transport stop equals 0 seconds, the time table is not considered. In this case, the settings in the range dwell time are taken into consideration for the calculation of the **Dwell time**.

Column	Description
	For line stops in PT partial routes, the following applies: - As long as a PT vehicle has not completed its original line route, its **Departure offset** is treated like an offset at a line stop of the original route. - Once the PT vehicle has passed the "to section" of its original line route, the **Departure offset** specified for a PT partial route stop is interpreted as relative to the simulation time when the vehicle passes the respective routing decision point.
PedsAsPass	**Pedestrians as passengers**: Pedestrians of an area are used as passengers for public transport. The area is a waiting area or a platform edge. The area is assigned at least one PT stop.
DwellTmDef	**Dwell time definition** depends on the method used for calculating public transport dwell time (see "Defining dwell time according to dwell time distribution" on page 532), (see "Calculating dwell time according to the advanced passenger model" on page 533), (see "Calculating dwell time with PTV Viswalk" on page 534)

6.12.9 Calculating the public transport dwell time for PT lines and partial PT routes

The following methods allow you to model stop dwell times for PT vehicles

Method	Description
Dwell time distribution	The dwell time is based on the dwell time distribution used (see "Defining dwell time according to dwell time distribution" on page 532). Define all desired dwell time distributions (see "Using time distributions" on page 246). You then assign the desired dwell time distribution to each stop serviced by a PT line or partial PT route.
Advanced passenger model	The dwell time and number of passengers boarding and alighting are calculated using the advanced passenger model (see "Calculating dwell time according to the advanced passenger model" on page 533). To model stop dwell times with the number of passengers boarding and alighting instead of with dwell time distributions, define PT parameters for the respective vehicle type (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275).
Viswalk (microscopic pedestrian simulation)	Dwell time and number of passengers boarding/alighting are calculated during the simulation (see "Calculating dwell time with PTV Viswalk" on page 534), (see "Modeling pedestrians as PT passengers" on page 984) and (see "Quick start: defining pedestrians as PT passengers" on page 987).

Without Viswalk, you can define the **Dwell time distribution** method faster than the **Advanced passenger model** in Vissim. The **Advanced passenger model** method, however, allows you to model the behavior at stops more precisely, e.g. the cumulation of vehicles of a PT line at a stop caused by a delay.

6.12.9.1 Defining dwell time according to dwell time distribution

1. Define the PT line, the PT stop and at least one dwell time distribution (see "Using time distributions" on page 246).
2. On the Network objects toolbar, click **Public Transport Stops** or **Public Transport Lines**.
3. In the network editor, right-click the PT stop.
4. From the context menu, select **Edit PT Line <No. > PT Stop <No.> PT Pass <No.>**.

   *The **PT Line Stop** window opens.*
5. In the **Dwell time** section, select **Distribution**.
6. Make the desired changes:

Element	Description
Active	If the option is selected, the public transport stop for the current PT line or PT partial route is activated. If the option is not selected, the public transport stop is not operated and is shown in green.
Skipping possible	**SkipPoss**: If the option is selected, the public transport stop is not used if the resulting, random dwell time is < 0.1 seconds. Skipped public transport stops are reported in the Vehicle Records file with its identifier and recorded as having a public transport stop dwell time of 0 seconds.
Departure time offset	**DepOffset**: Departure time defined according to timetable (see "Editing a PT line stop" on page 526)
Door lock duration before departure	**DoorLockDurBefDep**: This dwell time of the PTV vehicle is the time until departure of the vehicle after the doors have been fully closed. Default 1.0 s
Dwell time	**Dwell time definition** (**DwellTmDef**). All settings for the dwell time apply for the line stops of a PT partial route. **Distribution** and **Calculation** are always available if the public transport stop is not allocated a pedestrian area with **Public transport usage**. - **Distribution**: The stop time is obtained from the selected dwell time distribution. - **Calculation**: Alighting percentage as percentage indication (see "Calculating dwell time according to the advanced passenger model" on page 533). - **Alighting percentage**: Percent of the passengers who alight on this public transport stop. This value serves the volume-dependent calculation of the stop time.
You can also show further attributes in the attribute list **Public Transport Line Stops**, e.g.:	
Door closure delay	**DoorClosDel**: Time after which the last pedestrian has walked through doors until the doors begin to close. Default 3.0 s

7. Confirm with **OK**.

6.12.9.2 Calculating dwell time according to the advanced passenger model

1. Ensure that the following parameters are defined:
   - Boarding passenger profile per PT stop and assignment to desired PT lines (see "Attributes of PT stops" on page 513)
   - Specific PT parameters per vehicle type (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275)
   - Occupancy rate per PT line (see "Attributes of PT lines" on page 520), (see "Editing functions and distributions of a vehicle type" on page 271)
2. On the Network objects toolbar, click **Public Transport Stops** or **Public Transport Lines**.
3. In the network editor, right-click the PT stop.
4. From the context menu, select **Edit PT Line <No. > PT Stop <No.> PT Pass <No.>**.

   *The **PT Line Stop** window opens.*
5. In the **Dwell time** section, select **Calculation**.
6. Make the desired changes:

Element	Description
PT stop active	☑ If the option is selected, the public transport stop for the current PT line or PT partial route is activated. ☐ If the option is not selected, the public transport stop is not operated and is shown in green.
Skipping possible	☑ If the option is selected, the public transport stop is not operated if the vehicle passes the 50 m mark before the public transport stop and no passengers would like to board or alight. A public transport stop bay can only be fully skipped if both of these are directly connected to the link, from which the PT vehicle is coming. If the network structure is more complex or if the PT vehicle has already reached the public transport stop bay or the connector, when the 50 m mark is passed, the bay is passed without making a stop. Skipped public transport stops are reported in the Vehicle Records file with its identifier and recorded as having a public transport stop dwell time of 0 seconds.

Element	Description
Departure time offset	Departure time defined according to timetable (see "Editing a PT line stop" on page 526)
Dwell time	All settings for the dwell time apply for the line stops of a PT partial route. **Distribution** and **Calculation** are always available if the public transport stop is not allocated a pedestrian area with **Public transport usage**. - **Distribution**: The stop time is obtained from the selected dwell time distribution. - **Calculation**: The number of boarding passengers is determined on the basis of the boarding passenger profile at the public transport stop. The time required for the boarding and alighting is calculated on the basis of the PT parameters of the vehicle type. - **Alighting percentage**: Enter percent of passengers who alight at this public transport stop. This value serves the volume-dependent calculation of the stop time.

7. Confirm with **OK**.

After you have defined the PT line, PT parameters per vehicle type, and the attributes for the PT line stop, Vissim calculates the stop dwell times of a PT vehicle as follows:

- *Number of alighting passengers = Number of passengers x percentage of alighting passengers*
- *Number of boarding passengers = Number of all passengers waiting* (who want or are allowed to take the line) If the number of waiting passengers exceeds the free capacity of the PT vehicle, the number of alighting passengers is limited to the maximum capacity of the vehicle.
- *Alighting time = Number of alighting passengers x average alighting time* (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275)
- *Boarding time = Number of boarding passengers x average boarding time*
- *Passenger service time = Clearance time + alighting time + boarding time*

After completion of passenger service, the PT vehicle remains at the stop until the scheduled departure time that accounts for the respective slack time fraction.

6.12.9.3 Calculating dwell time with PTV Viswalk

You can define parameters for the calculation of the number of boarding passengers and the dwell time, which occurs during the microscopic pedestrian simulation. Example (see "Modeling pedestrians as PT passengers" on page 984)

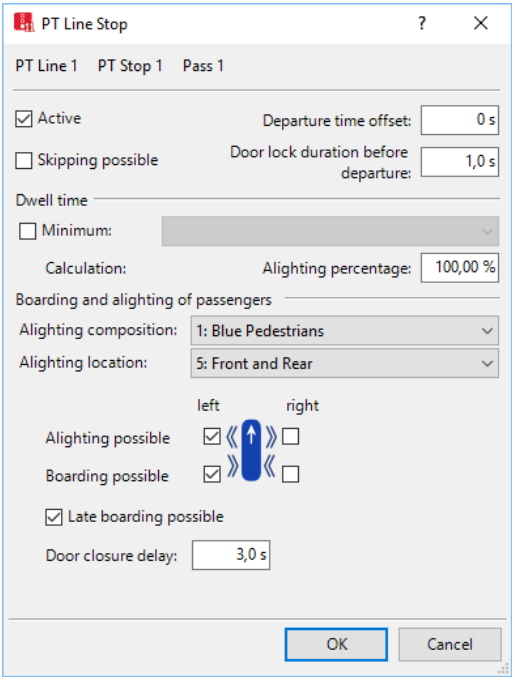
Note: For this, you require PTV Viswalk.

1. Ensure that the following requirements are met:

- The PT stop must have at least one pedestrian area with the **Public transport usage** attribute allocated of the **Waiting area** or **Platform edge** type (see "Attributes of areas" on page 898).
- At least one location distribution must be defined for alighting passengers (see "Using location distributions for boarding and alighting passengers in PT" on page 249).
- At least one pedestrian composition has to be defined (see "Defining pedestrian compositions" on page 931).

2. On the Network objects toolbar, click **Public Transport Stops** or **Public Transport Lines**.
3. In the network editor, right-click the PT stop.
4. From the context menu, choose **Edit PT Line <No. > PT Stop <No.> PT Pass <No.>**.

   *The **PT Line Stop** window opens.*



5. Make the desired changes:

Element	Description
PT stop active	☑ If the option is selected, the public transport stop for the current PT line or PT partial route is activated. ☐ If the option is not selected, the public transport stop is not operated and is shown in green.

Element	Description
Skipping possible	☑ If the option is selected, the public transport stop is not operated if the vehicle passes the 50 m mark before the public transport stop and no passengers would like to board or alight. A public transport stop bay can only be fully skipped if both of these are directly connected to the link, from which the PT vehicle is coming. If the network structure is more complex or if the PT vehicle has already reached the public transport stop bay or the connector, when the 50 m mark is passed, the bay is passed without making a stop. Skipped public transport stops are reported in the Vehicle Records file with its identifier and recorded as having a public transport stop dwell time of 0 seconds.
Departure offset	Departure time defined according to timetable (see "Editing a PT line stop" on page 526)
Dwell time section:All dwell time settings apply for the line stop of a PT partial route.	
Minimum	- ☑ If the option is selected, the minimum dwell time on the basis of the selected time distribution is determined. - ☐ If the option is not selected, the minimum dwell time = 0 s. If you have neither selected **Minimum** nor used a timetable, you must select **Late boarding possible**. PT vehicles depart once the minimum dwell time runs out. PT vehicles also depart if the minimum dwell time = 0 seconds when all alighting passengers have alighted. In addition, you can select for each line stop the option **Late boarding possible**
Calculation	**Calculation**: Alighting percentage as percentage indication when the public transport stop is allocated a pedestrian area as **Platform edge** or **Waiting area**. Viswalk pedestrians who board or alight at a public transport stop are modeled. The calculated public transport stop dwell time depends on the time necessary for boarding/alighting passengers. The option **Late boarding possible** is taken into consideration for the calculation of the public transport stop dwell time.
Alighting percentage	Percentage of the passengers, which will alight at this public transport stop. This value serves the volume-dependent calculation of the stop time.
Boarding and alighting of passengers section	
Alighting composition	Composition of the PT-passengers (see "Defining pedestrian compositions" on page 931). Within the PT vehicle, the Pedestrian types, corresponding to the selected composition, are generated and the given percentage set is used.
Alighting location	**Alighting location**: Distribution of the alighting passengers at the doors of the PT vehicle at this public transport stop (see "Using location distributions for boarding and alighting passengers in PT" on page 249). Within the PT vehicle, the alighting passengers at this public transport stop cor-

Element	Description
	responding to the selected composition are distributed to the vehicle doors.
Alighting possible, **Boarding possible**	Allow or disallow boarding and/or alighting on the right and left. An arrow on the layout of the PT vehicle shows the movement direction.
Late boarding possible	☑ If this option is selected: - the minimum dwell time is kept - the vehicle only departs after all passengers have boarded or vehicle capacity has been reached (see "Attributes of vehicle types" on page 269) - The doors close after three seconds have passed in which no passengers have wanted to board/alight. If you have neither selected the option **Minimum** nor used a time table, you must select the option **Late boarding possible**. ☐ If the option is not selected, the minimum dwell time = maximum dwell time. - If the Slack time fraction = 0, the PT vehicle departs immediately after the minimum dwell time has been reached. - If the Slack time fraction > 0, the corresponding share of the remaining time to the departure according to the time table adds to the dwell time if the departure time has not yet been reached. To ensure that the PT vehicle departs exactly at the time calculated, in case the option is not selected, no passengers can board the PT vehicle as soon as the doors begin to close. The closing of the doors always begins 3 seconds before departure. The doors also close when a passenger boards immediately before the doors begin to close.

6. Confirm with **OK**.

Parameters for micro-simulation of pedestrians of previous versions

- If you would like to reproduce the results of earlier Vissim versions, select the following parameters:

Option	Relevance	Settings
Minimum dwell time	PT line stop parameters	Nothing selected
Late boarding possible	PT line stop parameters	selected
Slack Time Fraction	Public transport line	no change
Departure offset	PT line stop parameters	no change

6.12.10 Defining partial PT routes

To define a partial PT route, insert a routing decision on a link and a destination section on a destination link. The partial routing decision or the destination section may also lie on a connector.

You can assign the interval limits via the time intervals (see "Defining time intervals for a network object type" on page 326) , (see "Calling time intervals from an attributes list" on page 327). If a distribution on a percentage basis of the traffic volume to the routes of a routing decision varies temporally, you must define multiple time intervals which do not overlap each other.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Vehicle Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Vehicle Routes** again.

   A list box opens.

3. Then select **Partial PT route**.

Tip: Alternatively to the following steps, to insert a routing decision, from the shortcut menu, choose **PT partial routing decision**.

4. Hold down the CTRL key and in the Network Editor right-click the desired link or connector on the desired position of the routing decision cross section.
5. Release the keys.

   By default, a purple bar is inserted.

6. On the desired link, point the mouse pointer to the desired position of the destination section.

   If Vissim does not find a valid link sequence, neither a yellow band nor a turquoise bar are displayed, or the band might be interrupted. Select another destination link or a new position for the destination section or correct the Vissim network, for example if a link is not connected properly with a connector.

If Vissim finds a valid connection via a link sequence, between the start section and the position the mouse pointer is pointing to, the link sequence is displayed as a yellow band by default.

On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A turquoise bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.

7. Right-click this position.

   The context menu opens.

8. Then select **Create partial PT route: Define end**.

   *A turquoise bar is added for the destination section by default. The **PT partial routing decision** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.*

9. Edit the attributes (see "Attributes of PT partial routing decisions" on page 539), (see "Attributes of partial PT routes" on page 540).

6.12.11 Attributes of PT partial routing decisions

1. From the **Lists** menu, choose > **Public transport** > **PT partial routing decisions**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Column	Description
No	Unique **Number** of PT partial routing decision
Name	Name of PT partial routing decision
Link	Number and name of link on which the PT partial routing decision lies
Pos	**Location**: Distance to the beginning of link or connector

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Public transport lines (see "Attributes of PT lines" on page 520)
- Partial PT routes (see "Attributes of partial PT routes" on page 540)

- Link (see "Attributes of links" on page 409)
- Destination link of partial PT route (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.12.12 Attributes of partial PT routes

1. From the **Lists** menu, choose > **Public Transport** > **PT Partial Routes**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Column	Description
VehRoutDec	**Vehicle routing decision**: Number and name of PT partial routing decision in partial PT route
No	Unique **Number** of partial PT route
Name	Name of partial PT route
Destination link	Number and name of link on which partial PT route ends
DestPos	**Destination position**: Distance between destination section and beginning of link or connector
RelFlow	**Relative volume** in time interval = absolute volume in time interval: Sum of the volumes of all time intervals If the relative load in a time interval = 0, no public transport partial route is selected.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Entry link (see "Attributes of PT lines" on page 520)
- Vehicle routing decision (see "Attributes of PT partial routing decisions" on page 539)
- Link sequence (see "Attributes of links" on page 409)
- Partial line stops: Attributes of line stops in partial PT route (see "Editing a PT line stop" on page 526)
- Destination link of partial PT route (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

6.13 Modeling right-of-way without SC

You can model the right-of-way without SC for non-signalized intersections, fork links, and merging links. Depending on your use case, you can use priority rules, conflict areas or stop signs in your model.

6.13.1 Modeling priority rules

You can model the priority rules for conflicting traffic flows which are not controlled by signals by means of priority rules.

Note: Model the standard priority rules for conflicting traffic flows which are not controlled by signals by means of conflict areas (see "Using conflict areas" on page 560). Only use priority rules, if conflict areas do not produce the desired results and if you have sufficient experience in modeling with priority rules.

Priority rules for conflicting traffic flows which are not controlled by signals are required in situations in which vehicles in different links or connectors need to consider each other. You can also use priority rules to model keeping intersections clear.

Add a priority rule to the marker at which a vehicle or vehicles on another link have to wait.

Vehicles on the same link mutually observe each other. This also applies to links with several lanes. Therefore you do not require any priority rules.

During simulation, Vissim automatically adds priority rules to parking lots with real parking spaces.

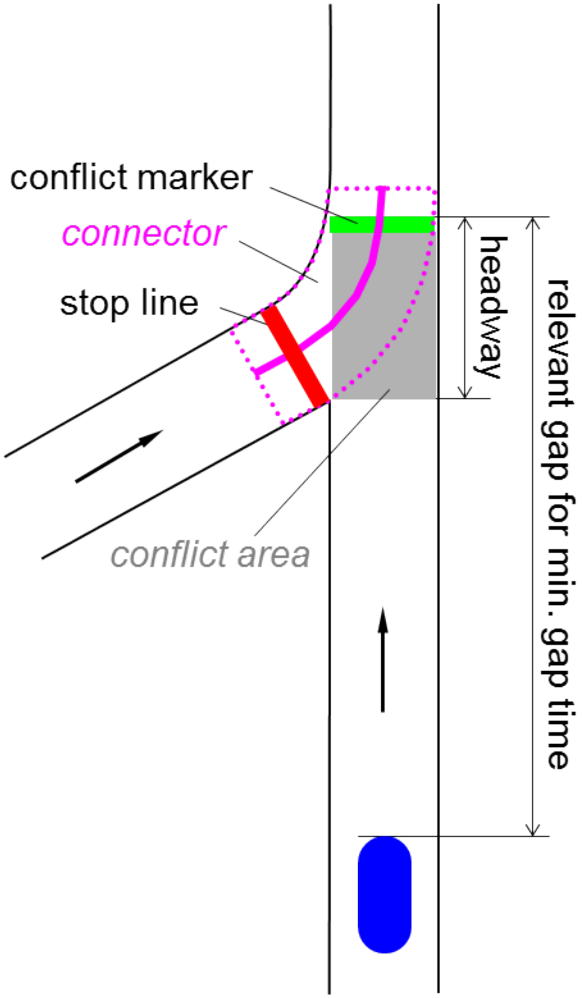
Tip: You can also use priority rules in Viswalk:

- For conflicting flows of pedestrians (see "Modeling priority rules for pedestrians" on page 929)
- For the interaction between vehicular traffic and pedestrian traffic (see "Modeling links as pedestrian areas" on page 922)

6.13.1.1 Creating priority rules

A priority rule always consists of at least two elements:

- Red bar: Stop line of the traffic which must wait, and therefore the conflicting markers, in the image below
- Green bar: One or more conflicting markers, in the top-right of the image

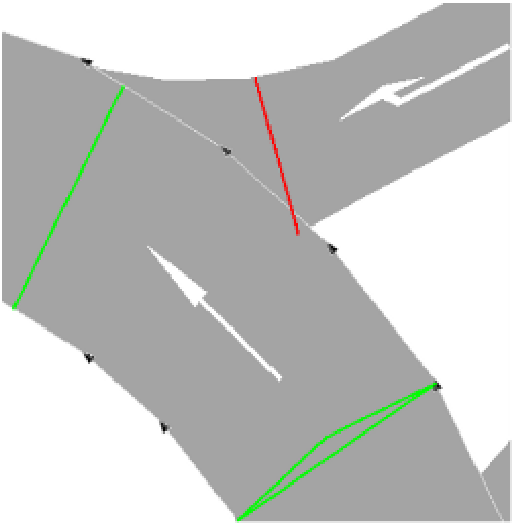


Min. Headway: The headway states the distance from the conflicting marker (green line) against the movement direction up to the first vehicle which is moving towards the conflicting marker. If a vehicle is still within the conflicting marker, the headway = 0.

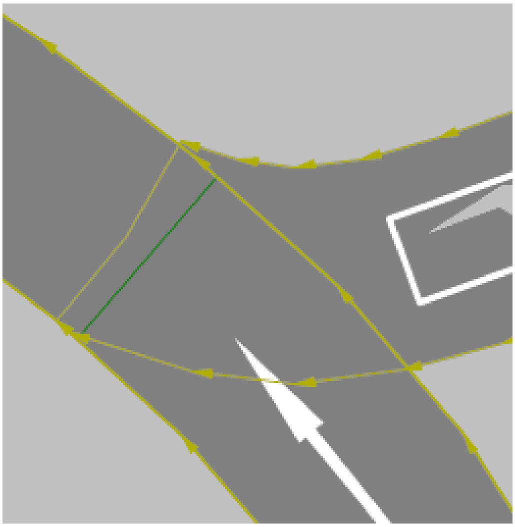
If a vehicle travels to a stop line, Vissim checks whether the prescribed value for the minimum headway and the minimum time gap upstream of the conflicting markers are present.

If the prescribed values are not present, the vehicle waits until both gaps are sufficiently long.

In selected priority rules, green triangles in movement direction indicate a minimum headway > 0 as distance between the conflict marker and the green triangle:



The conflict marker (in figures the top green bar on the left) also detects vehicles on all connectors, which lead upstream of the green bar onto the link. This behavior causes problems if the waiting vehicle is also detected by the conflict marker, for example if it is in the area of the headway of the green bar. To avoid this, always position the green bar on a link upstream of the end points of the relevant connectors to the link.



Min. Gap Time: The available time gap is the time that the first upstream vehicle will require in order to reach the green bar of the conflicting marker with its present speed. A vehicle which is already on the green bar is not taken into account. In a priority rule, the limiting time gap is specified: The vehicle must wait if the current time gap is less than the value which has been entered.

Depending on the situation which is to be modeled, either the headway or the limiting time gap is more important.

- Primarily, vehicles in a flow which has to wait in order to enter a flow which has priority, or which wish to cross such a flow, are oriented to the time gap.
- The headway is used if it has to be established whether a conflicting vehicle has already reached a certain location.

As well as this, the relevance depends on the ease of flow of the traffic in the conflicting marker:

- For a normal traffic flow, it is mainly the time gap which is relevant.
- In the case of slow-moving traffic and congestion, the headway is relevant.

In order that a vehicle does not need to stop and wait at a stop line, the conditions for all of the associated conflicting marker must be fulfilled.

For each red line (conflicting marker) Vissim takes one or several green bars (conflict markers) into account. Because of this, several different rules may apply for a stop line (red bar).

In the attributes, you may e.g. enter the following data:

- the vehicle classes of vehicles at the stop line
- The vehicle classes of the conflicting marker of the vehicle
- The maximum speed which a vehicle in the priority flow may still have in order for it to be recognized as a conflicting vehicle

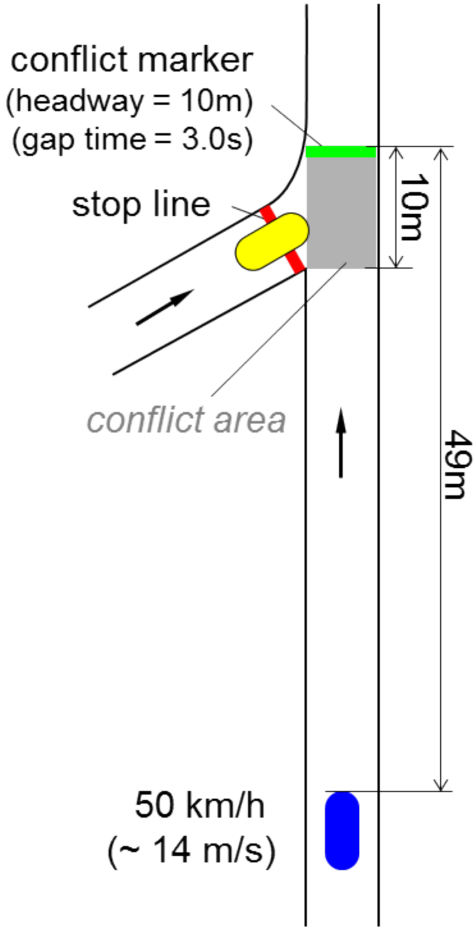
Red and green bars for conflicting and conflict markers can be specific to the route or to the traffic lane.

In order to simplify the modeling, both red and green bars may apply for **All lanes**. With this, it is sufficient to insert a single priority rule instead of several priority rules. If you have to use different attribute values, which are specific to different traffic lanes, you must define the appropriate number of green bars (conflict markers).

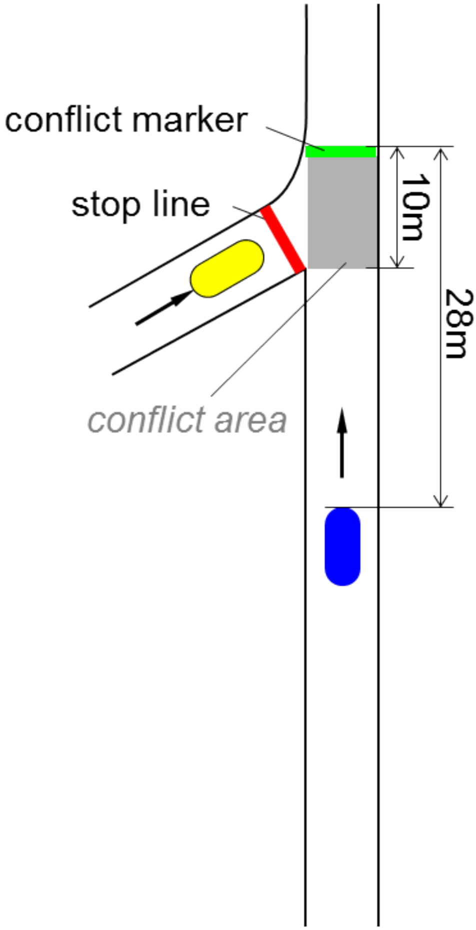
Note: If it appears that vehicles ignore the priority rules, this may be due to the fact that the priority rules are so defined that vehicles have to wait for themselves or have to wait for each other. Vissim resolves this deadlock. The vehicle with the longest waiting time may drive off first.

Examples for behavior at a junction with priority rule

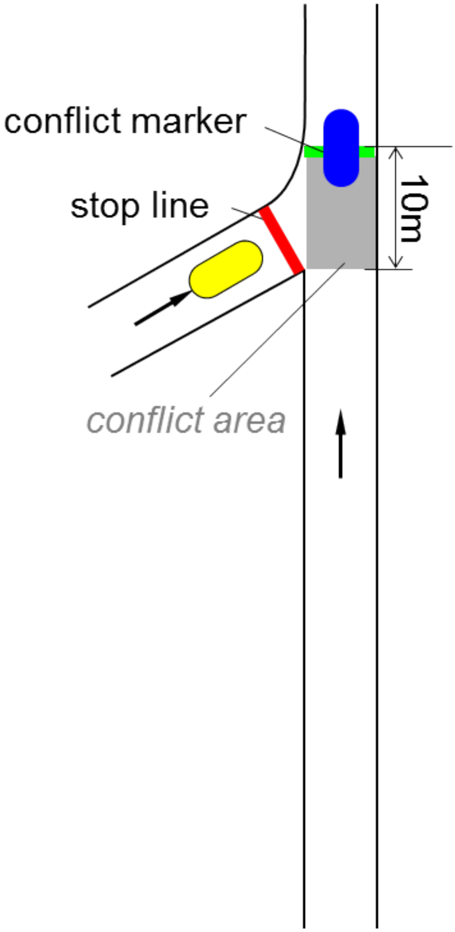
In the following figure, the blue vehicle on the left of the main road is traveling at a speed of 50 km/h (approx.14 m/s) and is 49 m upstream of the conflict marker. The present time gap is $49\ m / 14\ m/s = 3.5\ sec$. As the minimum time gap is 3.0 sec, the yellow vehicle at the bottom can enter from the side road.



In the following figure, the blue vehicle is still only 28 m from the conflicting marker. The present time gap is $28\ m\ /\ 14\ m/s\ =\ 2\ s$. As the minimum time gap is 3.0 sec, the yellow vehicle must wait:



In the following figure, the front end of the blue vehicle has just traversed the conflict marker. Therefore, the present time gap is 0 sec. However, the yellow vehicle must wait until the rear edge of the blue vehicle has completely cleared the conflict area, as the headway is greater than 0 m.



6.13.1.2 Defining priority rules

Add a start section for a priority rule and one or more destination sections on the links concerned:

- Start section: a priority rule on the conflict link, on which vehicles are required to wait.
- One or more destination sections (conflict markers) on the conflict link, on which vehicles have the right of way. By default, select the position 1 to 2 m before the end of the conflict area. The headway and time gaps must be present at the position opposite to the movement direction.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click > **Priority Rules**.
2. Hold down the CTRL key and right-click on the desired position of the stop line on the conflict link, on which vehicles should wait.
3. Release the keys.

   By default, a purple bar is inserted. If for this start section you want to insert multiple destination sections, carry out the following steps accordingly. Thereby you can insert a destination section and subsequently define its attributes.

   If you would like to insert a destination section for this start section, execute the next steps only once.
4. On the desired link, point the mouse pointer to the desired position of the first destination section.

   On the edge of the link, which you are pointing to with the mouse pointer, a black arrow is shown in the direction of travel. A colored bar shows the possible position of the destination section. Thereby you can select links from the different types of links, which are added in the next step of the destination section.
5. Hold down the CTRL key and right-click on this position.
6. Release the keys.

   *A green bar is added for the destination section by default. The **Priority Rule** window opens.*
7. Edit the attributes (see "Attributes of priority rules" on page 549).
8. Confirm with **OK**.

   *The window closes. The attributes are saved in the list **Priority Rules**. You can add additional destination sections in links or stop the addition.*
9. If you would like to add additional destination sections, on the desired link, point the mouse pointer to the desired position of the next destination section, and repeat the steps.
10. If you do not want to add any additional destination sections, in the Network editor, click in an empty area.

6.13.1.3 Attributes of priority rules

The **Priority Rule** window opens when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Priority Rules list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

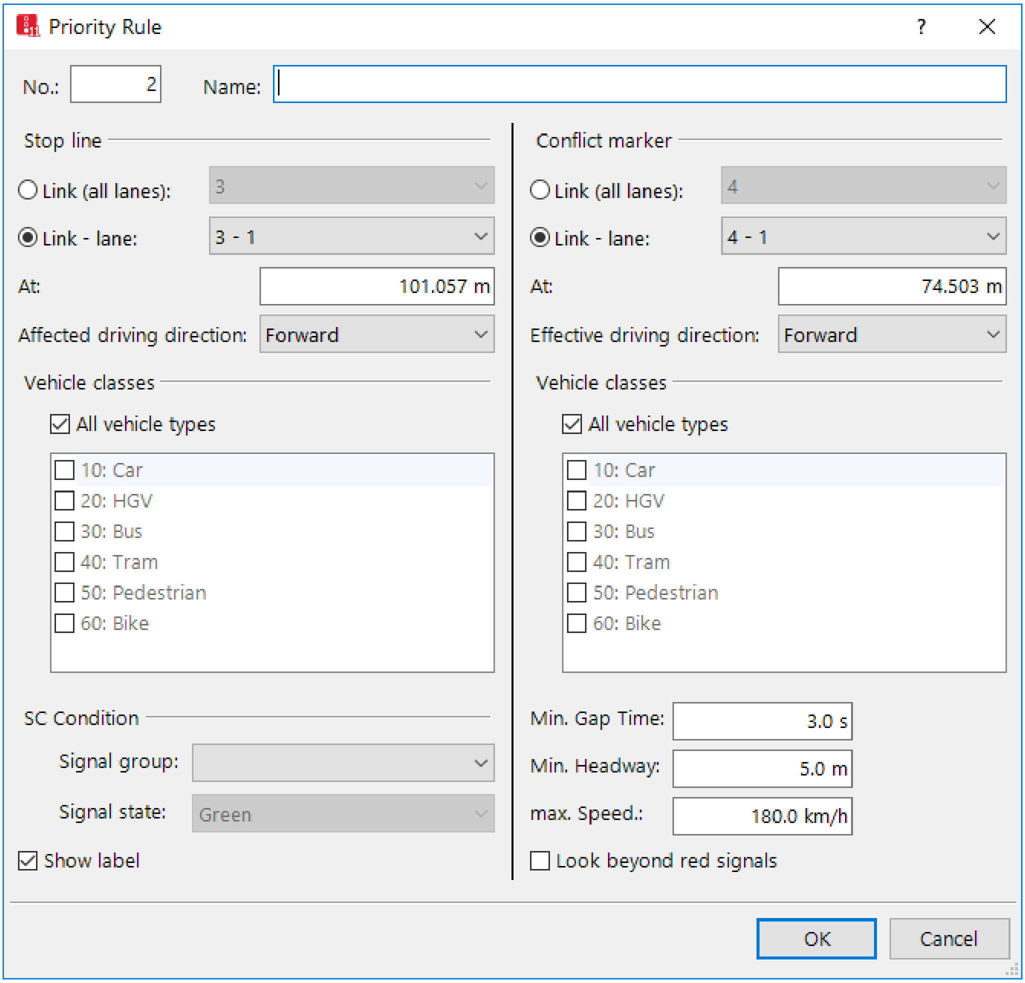
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



A network object may have the following attributes: These can be shown in the attributes list.

Element	**Description**
No.	Unique number
Name	Description
Elements under the sections Stop line and Conflict marker	
Link (all lanes)	Number of the link, in which the marker is located The marker extends over all lanes of the link.
Link - lane	**Ln**: Number of link and number of lane on which the marker is located. The marker extends over one lanes of the link.

Element	Description
At	Coordinate of the position of the priority rule: distance from the bus line to the beginning of the link or connector
Affected driving direction	**AffectDrivDir**: If one of the conflict markers of the priority rule recognizes a conflict, the stop line only causes vehicles to stop that are driving in the selected direction. ➤ **Forward** (default): Only vehicles moving forwards are recorded from the stop line on. ➤ **Reverse**: Only vehicles moving backwards are recorded from the stop line on. These are vehicles that are pulling out of a parking space in reverse (see "Modeling parking and stopping on the roadside" on page 493).
Vehicle classes	**VehClasses**: Vehicle classes for which the marker applies. The configuration of the vehicle class of a stop line (red bar) affects all of the associated conflict markers. In order to define a stop line for other vehicle classes, a new (separate) priority rule must be added, of which the stop line is at the same position.
Elements that are exclusively shown under Conflict marker.	
Stop only if	**Use signal controller condition** (**UseSCCond**): ☐ When the option is selected, the stop line is only active if the corresponding signal state of the chosen signal group is active. This is useful for example when all vehicles required to wait should not observe the vehicles located behind the stop line of a red SC. The other conditions, e.g. **Gap time**, **Headway** are also taken into account.
Label	☑ If the option is not selected, the label for individual Priority Rules is hidden, even when the label for all Priority Rules is selected.
Elements that are exclusively shown under Conflict marker.	
Gap time	Minimum gap time (**MinGapTime**) (in seconds) between the conflict marker and the next vehicle driving towards it.
Headway	Minimal headway (**MinHdwy**) (distance) between the conflict marker and the next vehicle upstream.
Max. Speed	Vehicles, which are traveling towards the conflict marker, are only considered for the headway condition when their $speed\ is \leq max.\ speed$.
Look beyond red signals	**LookBeyRedSig**: ☑ If this option is selected, vehicles traveling upstream of a red signal are also observed by the conflict marker. ☐ If the option is not selected, the time gaps and headways are only checked up to the Red SC.
Effective driving direction	Effective driving direction (**EffectDrivDir**): Only vehicles driving into the chosen direction are recognized by this conflict marker.

1. Confirm with **OK**.

The network object has additional attributes that you can show in the Attributes list. Among them are the following for example:

Element	Description
Slow down distance	**SlowDownDist**: Distance from stop line at which pedestrians start to reduce their speed in order to stop at the stop line. Default 3 m.

GenBy	**Generated by**: Indicates whether the priority rule is user-defined or has been generated by Vissim.
	- Default value **User**: The priority rule has been defined by another Vissim user.
	- **Parking lot**: Only during the simulation run: The priority rule has been defined by Vissim. A conflict marker may have the following values:
	  - **<Number of priority rule>: Parking lot <number>: Vehicle pulling out minds major flow**: Priority rule that causes the vehicle pulling out of the parking lot to stop if there is major flow traffic.
	  - **<Number of priority rule>: Parking lot <number>: Major flow minds vehicle pulling out**: Priority rule that causes major flow traffic to stop, if a vehicle is pulling out of the parking lot.
	  - **<Number of priority rule>: Parking lot <number>: Vehicle pulling in minds vehicle pulling out**: Priority rule that causes the vehicle pulling into the parking to stop and let the vehicle pulling out finish pulling out.
	  - **<Number of priority rule>: Parking lot <number x>: Vehicle pulling out minds vehicle pulling out of parking lot <number y>**: Priority rule that causes the vehicle pulling out of parking lot **x** to stop, while a vehicle is pulling out of parking lot **y**.

In the **Priority rules** list, double-click a priority rule to open the **Priority rule** window:

- Only during a simulation run: For priority rules that Vissim has automatically generated, the attributes of the stop line are displayed.
- For user-defined priority rules that Vissim has automatically generated, the section on the left displays the attributes of the stop line. The section on the right displays the attributes of the conflict marker.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

2. On the list toolbar, in the **Relations** list, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Pedestrian Classes (see "Attributes of pedestrian classes" on page 879)
- Conflict markers: Attributes of conflict markers, e.g. vehicle classes, gap time, headway (see "Attributes of links" on page 409). The attributes are described further above.

3. Enter the desired data.

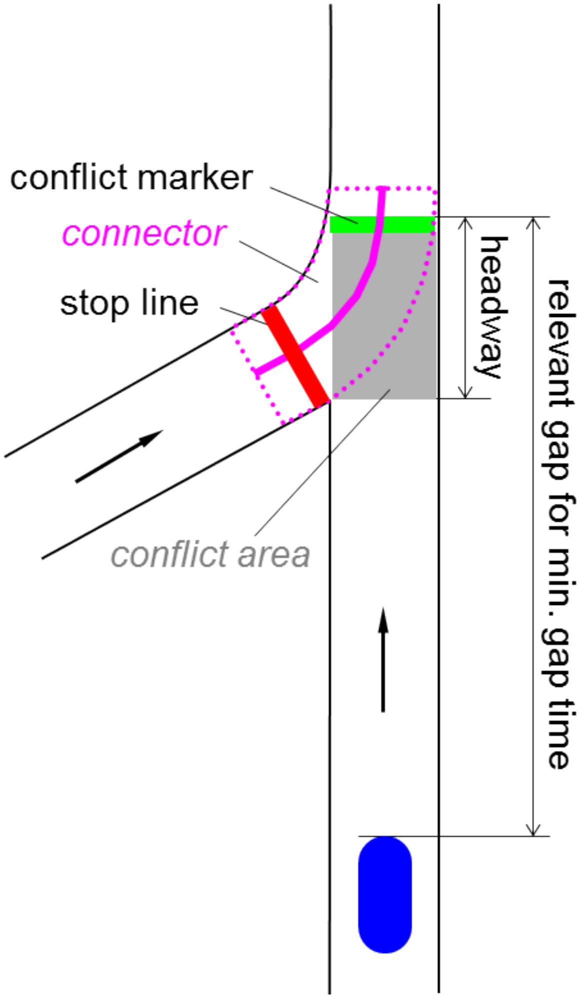
The data is allocated.

6.13.1.4 Examples of priority rules

Examples of using priority rules:

- Intersection with a main road, where vehicles have to wait (see "Priority rule Example 1: Minor yielding road leading into straight main road" on page 554)
- Avoiding tailbacks at a junction (see "Priority rule Example 2: Avoiding tailbacks at a junction" on page 555)
- Dual-lane roundabout with dual-lane entry (see "Priority rule Example 3: Dual-lane roundabout with dual-lane entry" on page 556)

Priority rule Example 1: Minor yielding road leading into straight main road



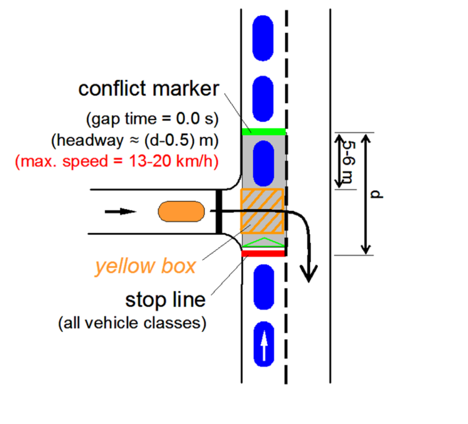
1. Position the left red bar (conflicting marker) on the stop line of the yielding road.
2. Position the top green bar (conflict marker) on the main road in movement direction, approx. 1 m upstream of the end of the conflict area.

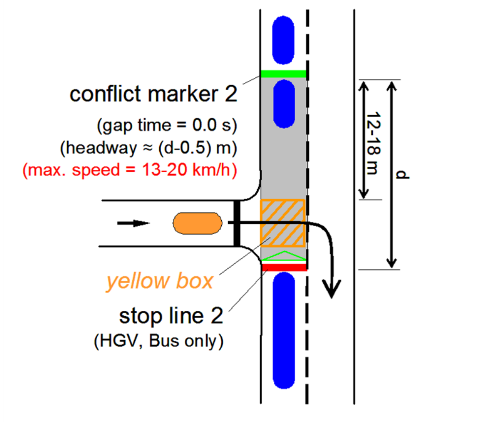
   This makes sure that the min. headway and min. time gap are checked for the main road only. You thereby exclude the possibility of a yielding vehicle waiting for itself. So do NOT position the green bar (conflict marker) on the connector between minor and main road.
3. Confirm the default values: min. headway = 5 m, min. time gap = 3 s.

Priority rule Example 2: Avoiding tailbacks at a junction

1. Place the red bar below (conflict marker) on the stop line of the yielding link, upstream of the junction.
2. On the same or the following link, place the top green bar (conflict marker) at a distance of at least one vehicle length from the junction.

   For a vehicle composition consisting of HGV and cars, a distance could for example be 20 m. The distance you choose should always account for the real driving behavior in such a situation.

<table>
<tr><th>Cars only</th><th>Additional priority rules for HGV/buses</th></tr>
<tr><td>


</td><td>


</td></tr>
</table>

3. The min. headway must be at least the distance between the stop line and the conflict marker (green bar). This way, you avoid that vehicles enter the yellow hatched conflict area as long as another vehicle is in there.

Note: The min. headway must not extend beyond the stop line. Otherwise, vehicles will also brake even if the can cross the stop line.

4. Enter a **maximum speed** of 10 to 20 km/h.

   This avoids possible congestion at the junction. The value defines the willingness of drivers to keep the junction clear. With a max. speed of 20 km/h drivers are more careful and the junction is more likely to be kept clear.
5. Enter a time gap of 0 sec.
6. Set the headway to a value which is slightly lower than the distance between the two markers.
7. For calibration of the model, use the position of the conflict marker and thus either the min. headway or speed.

Priority rule Example 3: Dual-lane roundabout with dual-lane entry

To model the entry to a roundabout, you need to specify several priority rules. They all serve different purposes. Cars, HGV and buses are treated differently on account of their acceleration capability and vehicle length. These vehicles thus have to be looked at separately.

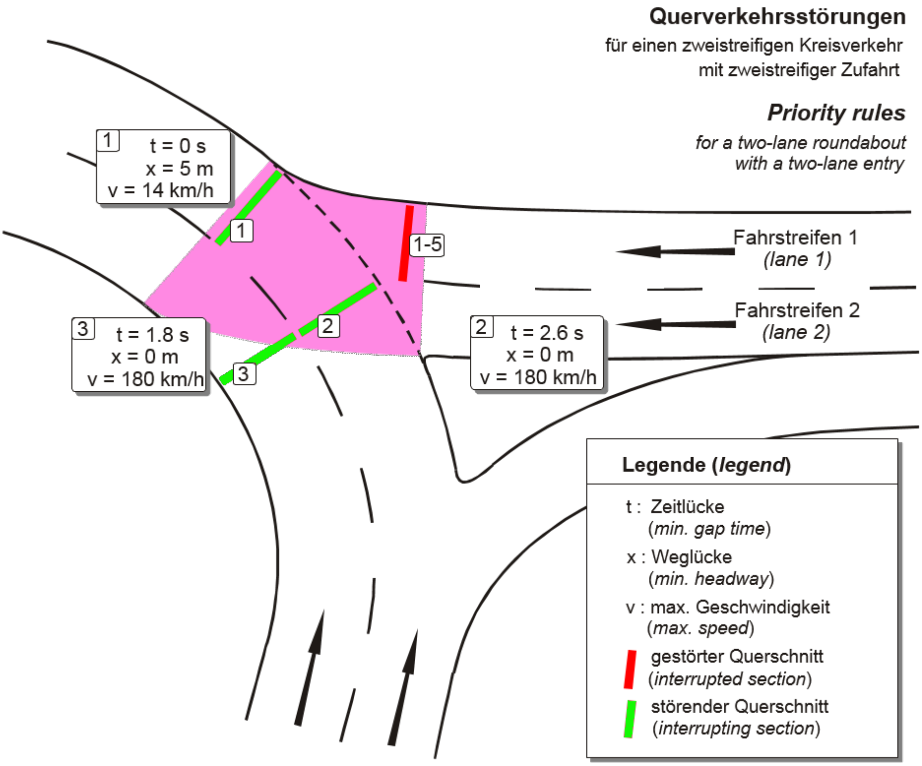
The priority rules and their purpose are depicted in the following illustrations **Step 1** to **Step 4**. For better understanding, you can find the numbers in the illustrations in the text. With these you can assign the associated attributes in the text. The values specified for time gap, headways, and max. speed have been determined through research and thus form a realistic modeling basis for most applications.

Set the priority rules according to the following criteria:

- Position the red bar (stop line) on the stop line of the yielding road. This is the typical waiting position for vehicles. If more than one green bar (conflict marker) refers to this stop line, several green bars must be set instead of separate pairs of priority rules.
- Position the green bars which are used for the headways, just before the connector turns into the roundabout lane. You thereby exclude the possibility of a yielding vehicle waiting for itself. Because of this the capacity of the roundabout would be drastically reduced.
- Position the green bars used for the time gaps at approximately the same distance from the conflict area as the respective red bars.

Step 1: Protect lane 1

Firstly, the priority rules for vehicles entering the roundabout from lane 1 are defined.



All 3 priority rules listed here refer to the same stop line. So this stop line has 3 conflict markers.

Select different positions for the min. time gap and min. headway. This allows you to model a more realistic driving behavior: A vehicle driving at least 14 km/h on a roundabout will allow another vehicle to accelerate into the roundabout, even if it is still in the conflict area. This leads us to the first two priority pairs (1 and 2). They are valid for all vehicle classes.

No. 1 secures the conflict area during slow moving traffic and congestion on the roundabout (min. headway).

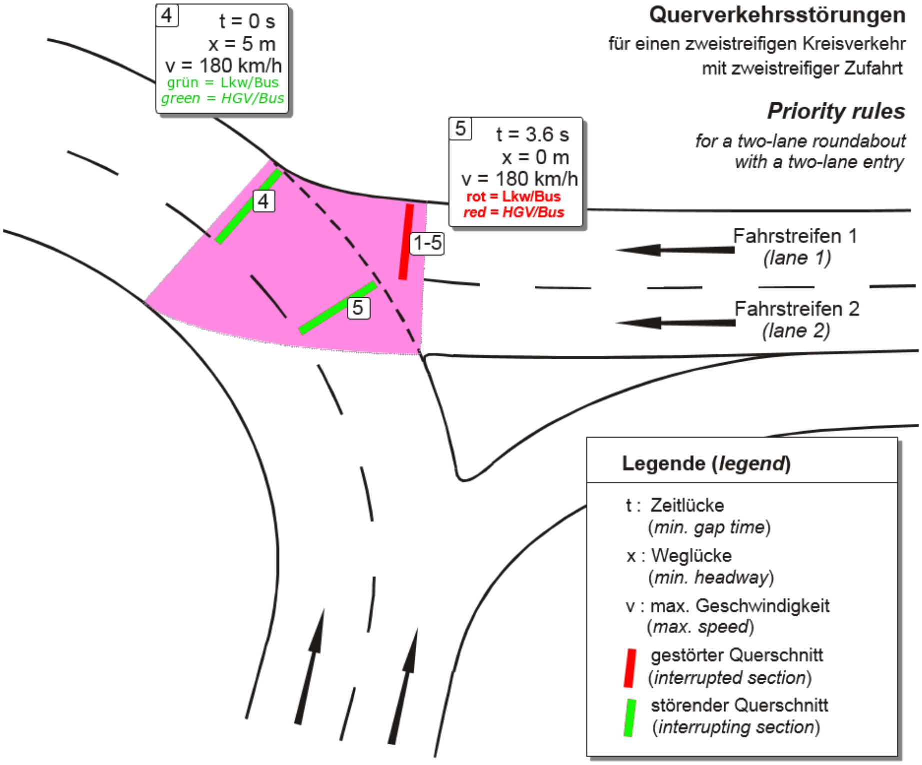
No. 2 provides the conditions for a normal flow of traffic (min. gap time).

As entering vehicles on lane 1 are also affected by traffic on the inner roundabout lane, an additional priority rule with a small gap time condition (No. 3) is needed for the inner roundabout lane. This priority rule is also valid for all vehicle classes.

Step 2: Protect the HGVs against entering vehicles

So far, all priority rules have applied for all vehicle classes. When long vehicles on the roundabout cross the conflict area, it is not enough to take of the vehicles that are slower than 14 km/h. To prevent vehicles entering the roundabout from laterally colliding with an HGV, an additional priority rule (No. 4) has to be positioned at the same point as rules 1-3. This priority rule must account for long, obstructive vehicles only. In our example these are HGV and buses.

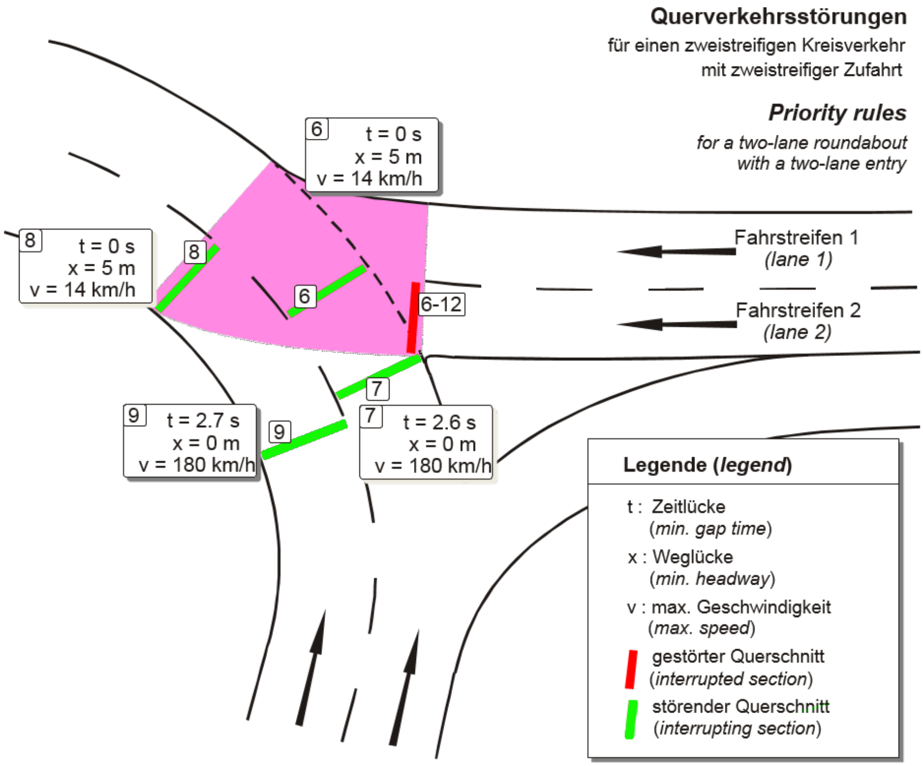
Now we only need to account for the vehicles that have a lower acceleration capability than cars. To do so, we use priority rule No. 5. Contrary to the previous priority rules, this rule needs a new, separate stop line, as it only applies for the vehicle classes HGV and bus. The stop line is positioned at the same point as rule No. 2. However, a longer gap time of 3.6 s must be specified for priority rule No. 5.



Step 3: Priority rules for the left incoming lane

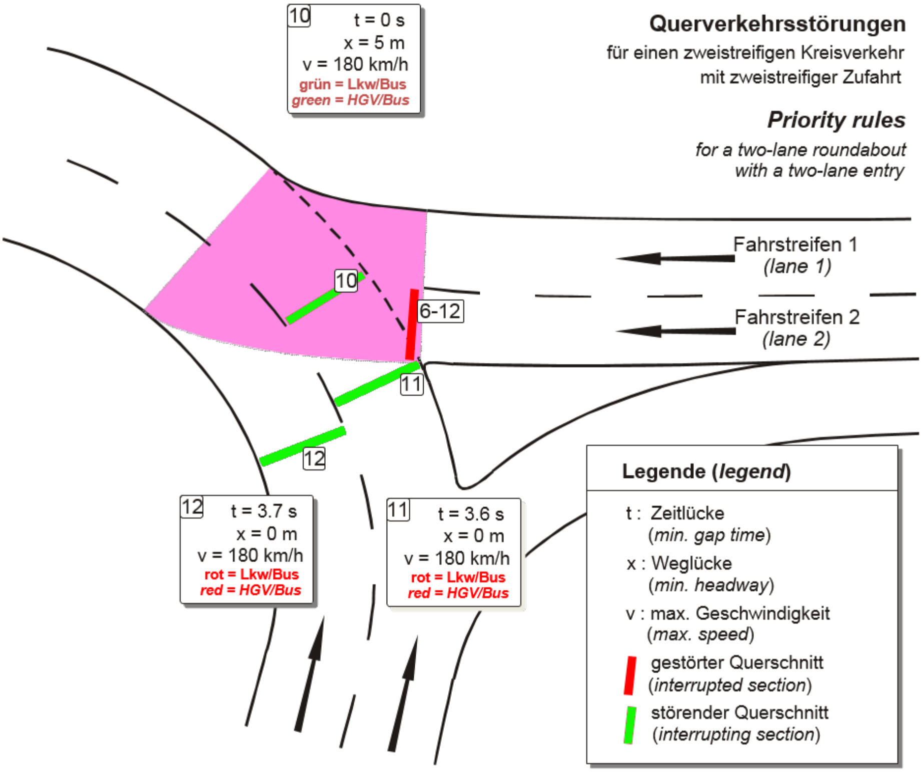
As for the right lane, general priority rules are defined for all vehicle classes with help of the min. time gap and min. headway. As in this case two lanes need to be taken into account, four priority pair rules have to be specified: No. 6 and 7 for the outer roundabout lane and No. 8 and 9 for the inner roundabout lane. All 4 conflict markers refer to the same stop line.

Due to the greater distance to the conflict area, the minimum gap time for the inner roundabout lane (No. 9) must be slightly higher than for the outer lane.



Querverkehrsstörungen
für einen zweistreifigen Kreisverkehr
mit zweistreifiger Zufahrt
Priority rules
for a two-lane roundabout
with a two-lane entry
6
t = 0 s
x = 5 m
v = 14 km/h
8
t = 0 s
x = 5 m
v = 14 km/h
8
6
6-12
Fahrstreifen 1
(lane 1)
Fahrstreifen 2
(lane 2)
7
9
t = 2.7 s
x = 0 m
v = 180 km/h
9
7
t = 2.6 s
x = 0 m
v = 180 km/h
Legende (legend)
t : Zeitlücke
(min. gap time)
x : Weglücke
(min. headway)
v : max. Geschwindigkeit
(max. speed)
gestörter Querschnitt
(interrupted section)
störender Querschnitt
(interrupting section)

Step 4: Priority rules for special vehicle classes



Finally, the priority rules for special vehicle classes are followed: as has already been carried out for lane 1, long vehicles must first be considered, No. 10 is added additionally for the same conflicting marker as for Nos. 6-9. HGV and buses entering the roundabout need longer gap times: No. 11 and 12 are added to a new stop line. Here, too, the time gap for the inner roundabout lane must be slightly longer than for the outer one.

6.13.2 Using conflict areas

Using conflict areas that are automatically shown, you can model conflicts between vehicles on two links or connectors (see "Defining the right of way at conflict areas" on page 564).

Use conflict areas instead of priority rules to model the right of way at intersections. Conflict areas are automatically displayed, are easier to edit and reflect the driving behavior better than priority rules (see "It is better to use conflict areas than priority rules to model driving behavior." on page 561).

6.13.2.1 Using conflict areas to model conflict types

The following conflict types may occur on a conflict area:

- **Crossing**: Two crossing links
- **Merging**: Two connectors lead to the same link or a connector leads to a link with other upstream traffic. If the two connectors lead to the first 5 m of the link and start on two links, a conflict between these two links is accounted for by the conflict area.
- **Branching**: Two connectors come off the same link or one connector comes off a link that continues further downstream for more than 0.5 m. If the two connectors lead to the first 5 m of two links, the conflict between these two links is also accounted for by the conflict area.

By default, the attribute **Conflict type determined automatically** is selected for the conflict area. This allows Vissim to determine the conflict area for the **conflict type** (see "Attributes of conflict areas" on page 565).

6.13.2.2 Displaying conflict areas

On the network objects toolbar, click **Conflict areas** to automatically show conflict areas in the network, where two links or two connectors overlap. For each conflict area, you can select the link that has the right of way. The conflict area may also remain passive and thus without any impact on the vehicles.

Conflict areas are not inserted in the following cases:

- If the height (z coordinate) of both links or both links and the connector differs more than 1.0 m in their overlapping area.
- If the overlapping is less than or equal to 0.5 m
- If at least one of the links ends less than 5 m after the start of the conflict area and no connector begins in it. This does not apply for the following links:
  - Connectors
  - Links which are defined as pedestrian areas (see "Modeling links as pedestrian areas" on page 922).
  - Links with input flows with vehicle compositions which contain vehicle types with the category **Pedestrians** (see "Modeling vehicle compositions" on page 452)

6.13.2.3 It is better to use conflict areas than priority rules to model driving behavior.

Conflict areas allow you to model driving behavior better than with priority rules, as in conflict areas, drivers plan how to traverse the conflict area:

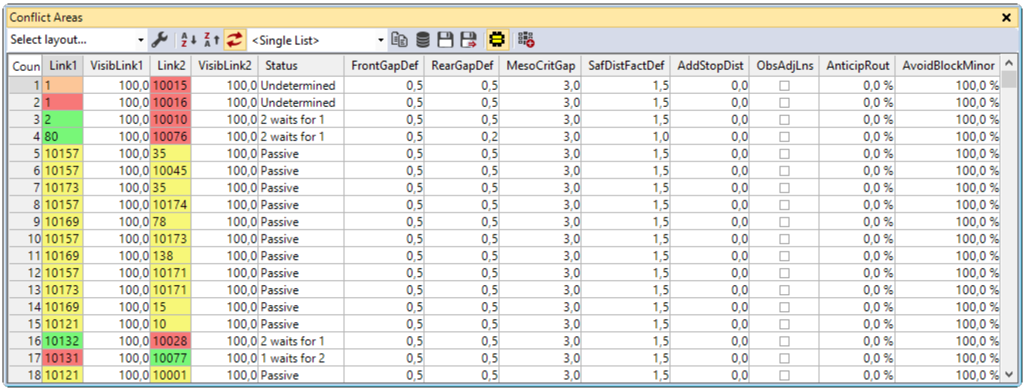
A yielding driver watches the vehicles in the main traffic stream and then decides when to filter in. He then plans to accelerate for the next few seconds. Acceleration allows him to pass the conflict area. He thereby accounts for the traffic downstream from the conflict area. If he knows that he will have to stop or drive slowly because of other vehicles, he will account for more time to cross the conflict area or he will decide to wait for longer.

Vehicles in the main traffic stream also react to conflict areas: If a vehicle does not manage to cross the entire conflict area because the driver has misjudged the situation, the vehicle in the main traffic stream will brake or even stop. If a queue is forming at a signal control downstream of the conflict area, the drivers of the vehicles in the main traffic stream try not to stop within the conflict area in order not to block any crossing traffic. The drivers that have the right of way carry out a comparable decision-making process for crossing the conflict area as the drivers whose vehicles are yielding.

6.13.2.4 Colors indicate the status of conflict areas

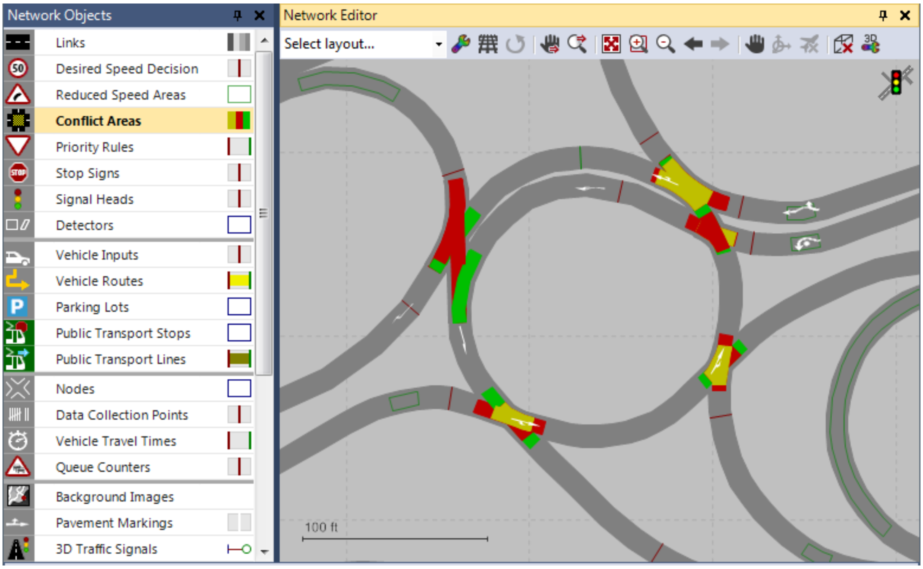
The status of conflict areas is displayed in the **Conflict Areas** list and in the Network editor.

- Green: main flow (right of way)
- Red: minor flow (yield)
- Both red: for branching conflicts, so that vehicles can "see" each other. There is no right of way, as vehicles simply remain in their original sequence.
- Both yellow: passive conflict area without right of way

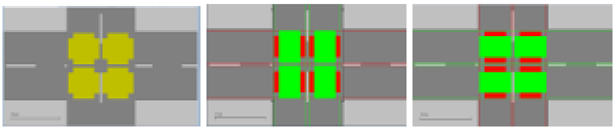


Coun	Link1	VisibLink1	Link2	VisibLink2	Status	FrontGapDef	RearGapDef	MesoCritGap	SafDistFactDef	AddStopDist	ObsAdjLns	AnticipRout	AvoidBlockMinor
1	1	100,0	10015	100,0	Undetermined	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
2	1	100,0	10016	100,0	Undetermined	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
3	2	100,0	10010	100,0	2 waits for 1	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
4	80	100,0	10076	100,0	2 waits for 1	0,5	0,2	3,0	1,0	0,0	☐	0,0 %	100,0 %
5	10157	100,0	35	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
6	10157	100,0	10045	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
7	10173	100,0	35	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
8	10157	100,0	10174	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
9	10169	100,0	78	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
10	10157	100,0	10173	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
11	10169	100,0	138	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
12	10157	100,0	10171	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
13	10173	100,0	10171	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
14	10169	100,0	15	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
15	10121	100,0	10	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
16	10132	100,0	10028	100,0	2 waits for 1	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
17	10131	100,0	10077	100,0	1 waits for 2	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %
18	10121	100,0	10001	100,0	Passive	0,5	0,5	3,0	1,5	0,0	☐	0,0 %	100,0 %

In the Network editor:



Display of different statuses:



6.13.2.5 Driving behavior at conflict areas

The driving behavior of vehicles approaching a conflict area shall produce the maximum capacity for a minor traffic stream, without affecting vehicles of the main traffic stream. Vehicles in the main traffic stream might be hindered by vehicles on the merge lane, the smaller the user-defined safety distance factor is.

- A vehicle in a minor traffic stream will calculate whether it will be able to filter into the main stream with every time step while approaching the conflict area. Thereby safety distances are taken into account (see "Defining the Wiedemann 74 model parameters" on page 294), (see "Defining the Wiedemann 99 model parameters" on page 296). If the driver feels there is a large enough gap in the main traffic stream, he will simply continue to drive. If the gap is too small, the vehicle will decelerate as if it had to stop in front of the conflict area. This calculation is repeated with the next time step. So braking is either cancelled or the driver continues driving and might even accelerate, e.g. when finding a gap in the traffic stream to enter.
- A vehicle on the main road is careful not to collide with any vehicles on the minor road. If it realizes that a vehicle on the minor road will still be within the conflict area when it arrives

there, it will brake in order to reach the conflict area just after the other vehicle has left it. As decelerating causes it to arrive later, it might continue its journey without any further braking in a later time step and pass right after the vehicle on the minor road.

- A vehicle on a minor road will brake before reaching a conflict area, if there is not enough space downstream of the conflict area to leave it. This means that particularly when there are several adjacent conflict areas, drivers have to either pass all of them or none, if there is not enough space for a full vehicle length.
- With a conflict area of the conflict type **crossing**, a vehicle in the main stream will try to keep the conflict area clear, if this vehicle belongs the percentage specified in the **Avoid blocking** attribute (see "Attributes of conflict areas" on page 565).

> Note: Vehicles of the main stream will not keep conflict areas of the **merging** or **branching** conflict types clear. To make vehicles keep conflict areas of the merging or branching type clear, you need to specify a priority rule (see "Priority rule Example 2: Avoiding tailbacks at a junction" on page 555)

- A vehicle in the minor stream will not enter a conflict area if it has to assume that it will not be able to leave it before the next vehicle of the main stream arrives. It thereby takes the safety distance into account (see "Attributes of conflict areas" on page 565)
- A vehicle of a minor stream that has already entered a conflict area will always try to leave it, even if this means that it has to enter another conflict area for which the **Gap** condition is not or no longer met.
- To avoid a collision, a vehicle may temporarily stop at an intersection within a red conflict area. If vehicle A recognizes that vehicle B is about to leave a conflict area, vehicle A can wait within another conflict area, for which it does not have the right of way, until B leaves.
- A vehicle waiting in a conflict area uses the next sufficiently large time gap to exit it. This behavior can also occur in a yielding vehicle, when it has to yield to the vehicle with the right of way. Example: A vehicle in a minor stream wants to turn into the major stream. It is waiting at an intersection in a conflict area because the vehicle in the major stream has the right of way. The vehicle in the major stream wants to turn left into the minor stream and is yielding because of oncoming traffic in the conflict area. However, if the time gap for the vehicle in the minor stream is sufficiently large, allowing it turn into the major stream in spite of the two vehicles, it leaves the conflict area and turns into the major stream. If you want a vehicle to take a headway into account, define a priority rule (see "Modeling priority rules" on page 541).

6.13.2.6 Defining the right of way at conflict areas

You can define the priority for conflicting traffic flows at automatically generated conflict areas.

1. On the Network objects toolbar, click **Conflict Areas**.

   The conflict areas are shown in color in the Network editor. Per default, conflict areas that have not yet been assigned a right of way are highlighted in yellow.

> Note: You can change the setting to where you do not have to press the CTRL key (see "Right-click behavior and action after creating an object" on page 152).

2. In the Network editor, click the conflict area for which you want to define the right of way.

   The conflict area is highlighted.

3. Hold down the CTRL key and right-click into the Network editor until the desired right of way is shown.

   - Green: main flow (right of way)
   - Red: minor flow (yield)
   - Both red: for branching conflicts, so that vehicles can "see" each other. There is no right of way, as vehicles simply remain in their original sequence.
   - Both yellow: passive conflict area without right of way

   *In the Network editor, the conflict area changes its color. In the **Conflict Areas** list, the following attributes are automatically adjusted:*

   - **Link 1** and **Link 2** are displayed in the color of the right of way
   - **Status**: right of way and colors

4. Edit the attributes (see "Attributes of conflict areas" on page 565).

*The attributes are saved in the **Conflict Areas** list.*

Note: Alternatively, right-click into the Network editor and choose the desired right of way from the shortcut menu. To do so, you need not select **Conflict Areas** on the Network objects toolbar. The shortcut menu will automatically show the right of way options available. Dependining on the current right of way of the conflict area, these can be the following:

- **Set Status to 2 waits for 1**:
  - Second link is highlighted in red: Minor flow (yield)
  - First link is highlighted in green: Main flow (right of way)
- **Set Status to 1 waits for 2**:
  - First link is highlighted in red: Minor flow (yield)
  - Second link is green: Main flow (right of way)
- **Set Status to Undetermined**: Both links turn red: For branching conflicts, so that vehicles can "see" each other. There is no right of way, as vehicles simply remain in their original sequence.
- **Set Status to Passive**: Both links turn amber: No right of way specified

---

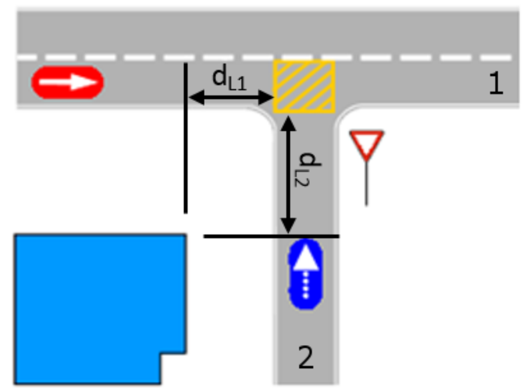
6.13.2.7 Attributes of conflict areas

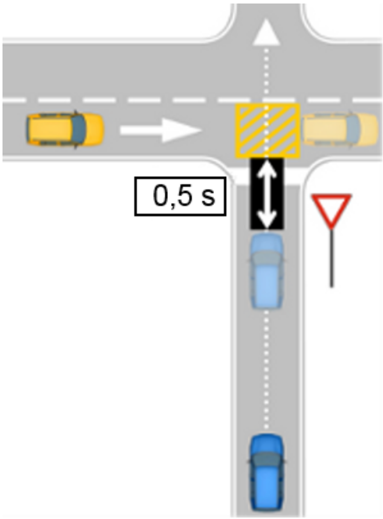
The attributes of a conflict area influence the driving behavior of any vehicle approaching the conflict area. This is why a vehicle may change its intention, and thus its driving behavior, in a particular traffic situation. The situation arising thereof may differ from the attribute values defined. This is particularly true for any resulting gaps in front of and behind two vehicles.

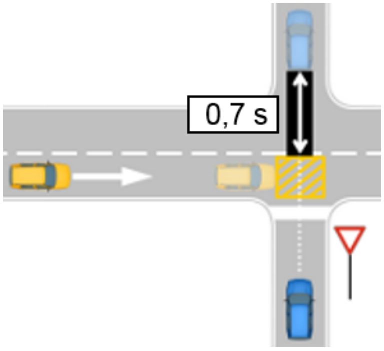
In the network objects list of the network object type **Conflict Areas**, you can edit all attributes and attribute values of conflict areas (see "Opening lists" on page 95), (see "Selecting cells in lists" on page 106).

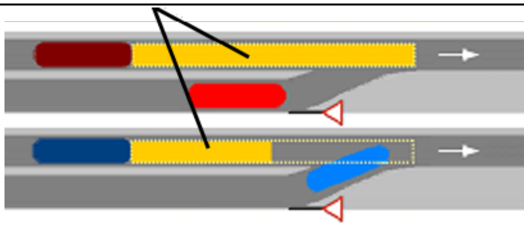
Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

1. Make the desired changes:

Element	Description
No	Unique number
Link1, **Link2**	**Link 1**, **Link 2**: Number and name of link with the conflict area
VisibLink1, **VisibLink2**	**Visibility link 1**, **Visibility link 2**: Maximum distance at which an approaching vehicle can "see" the vehicles on the other link. As long as a vehicle is on a non-priority link still far away from the conflict area, it plans to stop as close as possible to it. Avoid values < 1 m to keep the vehicle moving. In the Figure, the blue vehicle at the bottom on link 2 has reached the point from where it can fully "see" past the blue building on link 1 where the red vehicle is (on the left). This means the visibility for link 2 = *dL2*. 
Status	Identification of right of way by colors: - **2 waits for 1**: Link 1 green - Link 2 red - **1 waits for 2**: Link 1 red - Link 2 green - **Undetermined**: Both red for branching conflicts, so that vehicles can "see" each other. There is no right of way, as vehicles simply remain in their original sequence. - **Passive**: Both yellow: passive conflict area without right of way
FrontGapDef **FrontGap**	**Front gap (default)**: - For the types **merging conflicts** and **crossing conflicts**: Minimum gap time in seconds between the rear end of a vehicle in the main

Element	Description
	traffic stream and the front end of a vehicle in the minor traffic stream. Default 0.5 seconds To adhere to the minimum gap time, the yielding vehicle slows down as it approaches the conflict area and stops in front it, as long as the vehicle that has priority is front of or in the conflict area. Once the vehicle with the right of way has left the conflict area, the yielding vehicle can enter it and no longer takes the **Front gap** into account. ➤ Into the **Gaps** list on the right, you can enter the minimum gap time **FrontGap** per vehicle class. The figure below shows the current situation (the non-transparent vehicles) and the future situation (the semi-transparent vehicles). In the future situation, the vehicle in the major stream has just left the conflict area. Up until this time, the **Front gap** is considered the time required by the vehicle in the minor stream to reach the empty conflict area (in this case: 0.5s) 
RearGapDef **RearGap**	**Rear gap (default)**: Only for type **Junction**: ➤ Minimum gap time in seconds between the rear end of a vehicle in the minor traffic stream and the front end of a vehicle in the main traffic stream. This is the time that must be provided, after a yielding vehicle has left the conflict area and before a vehicle with the right of way enters it. Vehicles are perceived within a maximum distance of up to 100 m. Default 0.5 seconds ➤ Into the **Gaps** list on the right, you can enter the minimum gap time per vehicle class.

Element	Description
	The figure shows the current and future situation (as light, semi-transparent vehicles), once the vehicle in the main traffic stream has reached the conflict area. The rear gap is evaluated as the time that has elapsed since the vehicle in the minor traffic stream has left the conflict area (in this case: 0.7s) 
MinGapBlockDef **MinGapBlock**	**Minimum gap blocking (default)**: Only applies if the attribute **Avoid blocking the major flow** is not selected and thus a yielding vehicle may enter the conflict area, blocking the major flow: - Default minimum gap time for the yielding vehicle for entry before the vehicle with the right of way. **Minimum gap blocking (default)** is used for all vehicles that are not part of a vehicle class for which a class-specific gap time has been defined. Default 3.0 s - Into the **Gaps** list on the right, you can enter the **Minimum gap blocking** per vehicle class.
MesoCriticGap	**Meso critical gap**: Edit this value in the **Meso turn conflicts** list or in the coupled list **Nodes** - **Meso turn conflicts** (see "Attributes of meso turn conflicts" on page 835), (see "Attributes of nodes" on page 709).
SafDistFactDef	**Safety distance factor**: only for the type **merging conflicts**: This factor is multiplied with the normal desired safety distance of a vehicle in the main traffic stream in order to determine the minimum distance a vehicle of the yielding traffic stream must keep when it is completely in the conflict area **merging conflicts**. Into the **Gaps** list on the right, you can enter the safety distance factor per vehicle class. The figure below shows identical situations, but with different factors: top = 1.0, bottom = 0.5. This is why the blue vehicle (bottom) can still enter the conflict area, while the red vehicle (top) has to stop. In the figure, the relevant part of the safety distance is highlighted in yellow.

Element	Description
	
AddStopDist	**Additional stop distance**: only relevant for vehicles that are required to yield: Distance in meters that moves an imaginary stop line upstream of the conflict area. As a result, vehicles required to yield stop further away from the conflict and thus also have to travel a longer distance to pass the conflict area. Default 0 m.
ObsAdjLns	**Observe adjacent lanes**: ☑ If this option is selected, at merging conflicts, incoming vehicles of the main traffic stream that are required to yield will account for the vehicles in the main traffic stream that want to change to the conflicting lane. This option reduces the simulation speed.
AnticipRout	**Anticipate routes**: Percentage of vehicles required to yield that account for the routes of vehicles with the right of way. These are approaching with the main traffic stream and will turn further upstream. They will thus not reach the conflict area. Value between 0 and 1.
AvoidBlockMinor	**Avoid blocking the minor flow**: If a vehicle with the right of way belongs to the percentage rate selected, it will check the space available downstream of the conflict area and does not drive into the conflict area under the following conditions: - when the space available downstream of the conflict area is less than the *total of the individual vehicle length + 0.5 m and the blocking vehicle is slower than 5 m/s* and slower than 75% of its desired speed - when the obstacle is a red signal
AvoidBlockMajor	**Avoid blocking the major flow** - ☑ If this option is selected, a yielding vehicle does not enter or stop within the conflict area, unless it can drive through it in one go. The option is selected by default. For the yielding vehicle, the attribute **Minimum gap blocking** is not taken into account. - ☐ If this option is not selected, a yielding vehicle can enter and stop within the conflict area, even if it cannot drive into the following lane conflict, as it is blocked by a vehicle that has the right of way. In this case, the yielding vehicle might block the major flow. For the yielding vehicle, the attribute **Minimum gap blocking** is taken into account.

Element	Description
	Example: You are modeling a long conflict area between a vehicle route and a very wide pedestrian route with many narrow lanes and with a priority for pedestrians. Deactivate the option to let a vehicle successively drive into and traverse the lane conflicts without having to wait for a time when it can drive through the entire conflict area, and possibly other conflict areas lying close behind it, in one go.

The network object has additional attributes that you can show in the Attributes list. Among them are the following for example:

Element	Description
Conflict type determined automatically	**ConflTypDetmAuto**: ☑ If this option is selected, Vissim automatically determines the conflict type based on whether links intersect or connectors begin or end on the link. ☐ If this option is not selected, in the Attributes list, in the **Conflict type (manual)** column, the text box is no longer hatched and you can select the conflict type.
Conflict type (manual)	**ConflTypMan**: Select conflict type (see "Using conflict areas" on page 560): **Merge** **Branching** **Crossing** A conflict type is valid for all lanes of a conflict area. You cannot select different conflict types within a conflict area.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Defining time gaps [s]

The attributes are described further above.

6.13.3 Modeling stop signs and toll counters

Model stop signs on access roads to intersections with a combination of right-of-way control and stop sign.

Priority rules account for conflicting traffic as well as for the min. headway and min. gap time. Stop signs, however, make vehicles stop for at least one time step, independent of whether there is conflicting traffic or not.

Conflict areas also take desired acceleration, maximum acceleration as well as the vehicle length of the two vehicles in both streams into account and reflect the driving behavior better than priority rules (see "It is better to use conflict areas than priority rules to model driving behavior." on page 561).

Toll counters are a variant of stop signs that are additionally assigned a dwell time distribution value.

Stop signs can be used for the following modeling scenarios:

- **Regular stop sign**: You need to define a stop sign and a priority rule. Position the stop sign at the same position as the stop line (red) of the priority rule (see "Modeling priority rules" on page 541).
- **Signal controller with turn on red**: Right turns are allowed in spite of a red signal. In the **Green arrow** tab, select **Only on red**, to enable the stop sign only when the selected signal group of the selected **SC** indicates **red**.
- **Toll counter**, e.g. customs office or toll gate: Vehicles stop according to the time distribution defined in the attribute **DwellTmDistr** (**Dwell time distribution**). On the **Time Distribution** tab, select **Use time distribution** (**UsesDwellTmDistr**).

6.13.3.1 Defining stop signs and toll counters

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Stop Signs**.
2. Hold down the CTRL key and right-click on the desired position of the stop sign in the link or the connector.

   *A colored bar is added. The window **Stop Sign** opens.*
3. Edit the attributes (see " Attributes of stop signs and toll counters" on page 572).
4. Confirm with **OK**.

*The attributes are saved in the list **Stop Signs**.*

6.13.3.2 Attributes of stop signs and toll counters

The **Stop Sign** window opens when you insert a network object and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152).

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

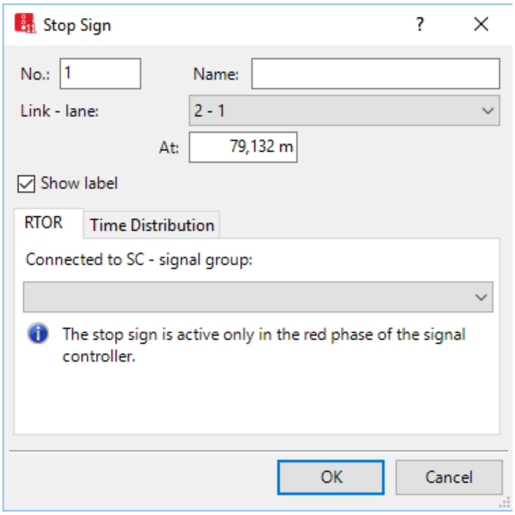
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



Basic attributes of stop signs

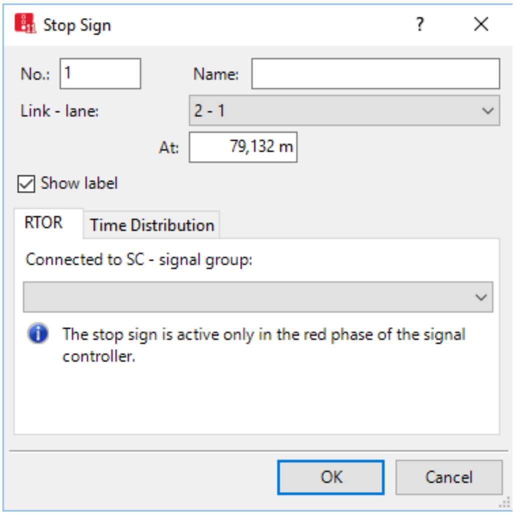
The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number
Name	Designation
Element	**Description**
Link - lane	Number of the link and lane, on which the stop sign is located.
At	**Pos**: Distance from the beginning of link or connector
Showing label	If the option is not selected, the label for the individual stop sign is hidden, even when the label for all stop signs is selected.

RTOR tab

The tab contains, amongst others, the following attributes:

Element	Description
Connected with SC - signal group	Number of signal controller and number of signal number group in which the green shall be displayed. Right turn on red: defines a green arrow symbol 720 (see " Using stop signs for right turning vehicles even if red" on page 575)



Time Distribution tab

In the **Time Distribution** tab, you can use dwell time distributions to define toll counters as a variant of stop signs.

The tab contains, amongst others, the following attributes:

Element	Description
Use time distribution	**Uses dwell time distributions** (**UsesDwellTmDistr**): Select wait time at toll counters: ☑ If the option is selected, under each vehicle class a time distribution (see "Using time distributions" on page 246) can be allocated in the list. All vehicles of a vehicle class, which use a time distribution, adhere to their corresponding, chosen time distributions. Vehicles without allocated distributions stop for one time step.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Dwell time distributions**.

The attributes are described further above.

3. Enter the desired data.

The data is allocated.

6.13.3.3 Using stop signs for right turning vehicles even if red

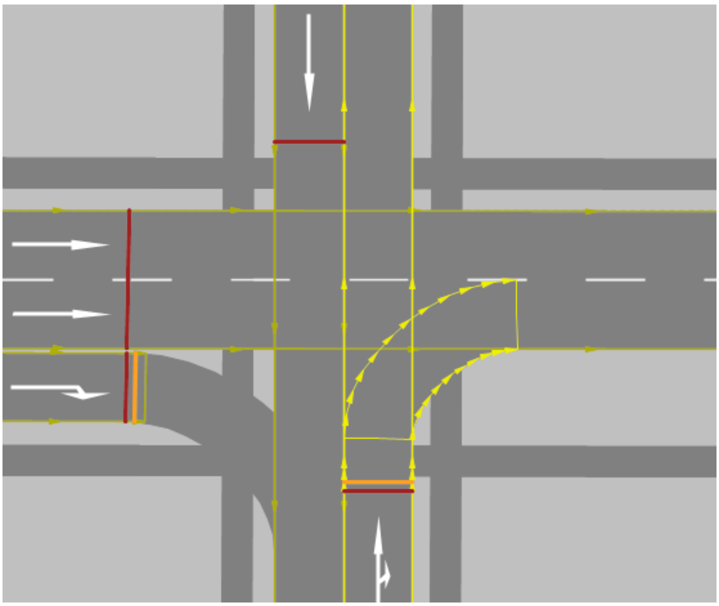
You can also set stop signs to model the behavior for turning, even if red at sign 720 green arrow (according to StVO German Traffic Code).

Examples:

- Special right-turn only lanes: The stop sign must be placed on the right turn lanes. Additionally, a signal head can be placed on the lane and for a vehicle type, for example, a tram or a pedestrian can be selected. Thereby the turning vehicle in the lane is not affected, but the state of the signal is visible.
- Combined right turning and straight lanes: If the option is selected, the stop sign must be placed on the connector for right turns. This makes the stop sign only visible for turning vehicles. The signal heads are placed at the same positions, however, it would be better to place them on the link rather than on the connector. The signal head controls the traffic driving straight.

The image shows:

- Five signal heads: (dark) red bars at the top, on first three lanes on the left, and at the bottom right
- Two stop signs: bright (orange) bars on the left, separate turning lane and on the bottom, right combined turning lane



1. On the Network objects toolbar, click **Stop Signs**.
2. In the Network Editor, right click on the desired position of the link.
3. From the context menu, select the entry **Add New Stop Sign**.

   *The window **Stop Sign** opens.*
4. Select the **RTOR** tab.
5. Select the option ☑ **Only on Red**.
6. Select the SC and the signal group.
7. Confirm with **OK**.

The stop sign is only active during the red phase of the allocated signal control.

6.13.4 Merging lanes and lane reduction

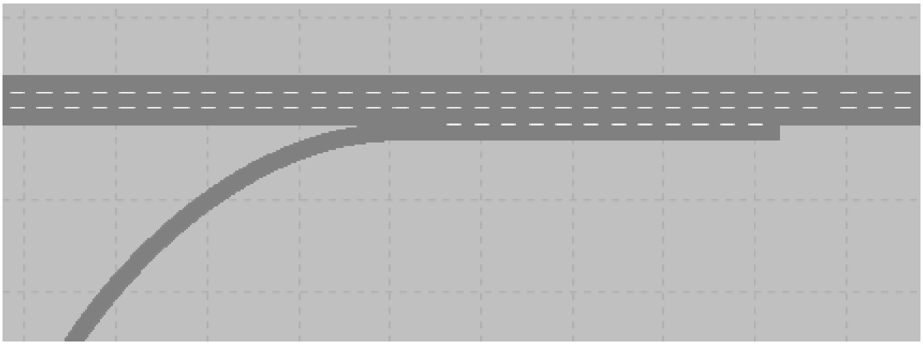
To model driving behavior at acceleration lanes and lane reduction, it is best to define links as merging lanes.

- Make sure the weaving section is a link with the following properties:

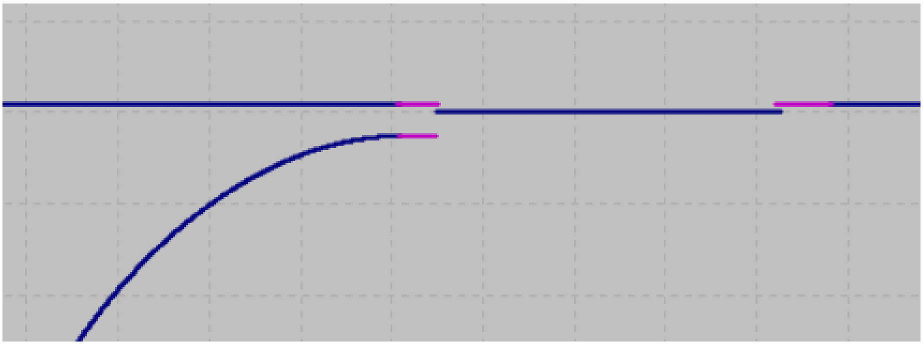
  $Number\ of\ lanes = Number\ of\ lanes\ on\ main\ link + number\ of\ merging\ lanes$
- Make sure that after the weaving section there is only one connector to the main link. For a realistic graphical representation, add a dummy link at the end of the merging lane. This will make reduction of the lanes less abrupt. Do not use a connector for this link.
- Make sure the through traffic is following a route, so that it does not switch to an acceleration lane. This route must end no sooner than on the link after the merging lane. Downstream of the merging lane, the value of the **Lane change** attribute (**Lane change distance**) of the connector must be larger than the weaving section length (see "Attributes of connectors" on page 422). If this is not the case, a vehicle on the main link may change to the acceleration lane and thus to the weaving section. It will then need to change back to the main link. This kind of behavior produces unrealistic lane changes.
- The routes of the merging traffic must not end at the weaving section, but extend beyond it. If not, the interweaving traffic will not know that it needs to change lanes before the end of the merging lane in order to get on the main link.

Example of a one-lane link merging into a three-lane link.

Merging section in normal display mode:



Merging section in wireframe display mode:



6.14 Modeling signal controllers

You can define signal controllers (SCs) for the following signal control procedures:

- Use signal controller **Balance-Central** (see "Using Balance-Central signal controllers" on page 612)
- **Econolite ASC/3** (for North America only) (see "Add-on module Econolite ASC/3" on page 614)
- Use signal controller **Epics/Balance-Local** (see "Using Epics/Balance-Local signal controllers" on page 613)
- The SC type **External** allows you to access dialog DLL files and user-defined program DLL files for control (see "Add-on module external signal control SC" on page 615)
- **Fixed time**, with add-on module Vissig (see "Using SC type Fixed time" on page 607)
- **Fourth Dimension D4** (for North America only) (see "Add-on module Fourth Dimension" on page 617)
- **LISA+ OMTC** (see "Add-on module LISA+ OMTC" on page 618)
- **McCain 2033** (for North America only) (see "Add-on module McCain 2033" on page 618)

- **Ring Barrier Controller** (for North America only) (see "Using the Ring Barrier Controller RBC add-on module" on page 619)
- **SCATS** (Australia) (see "Add-on module SCATS" on page 621)
- **SCOOT** (see "Add-on module SCOOT" on page 622)
- **Siemens VA** (Siemens-VS-PLUS, TL, PDM-C by Siemens AG, Munich) (see "Add-on module Siemens VA (TL / Siemens VS-PLUS)" on page 624)
- **TRENDS** (see "Using add-on module TRENDS" on page 624)
- **VAP** (vehicle-actuated signal control, freely programmable logic according to RiLSA 1992, guidelines for signal controllers in Germany) (see "Add-on module Traffic-dependent VAP Programming" on page 626)
- **VS-Plus** (Rudolf Keller AG, Basel) (see "Add-on module VS-Plus" on page 628)

For the signal control procedures **Fixed time control**, **Epics Balance/Local** and **External**, you can use the add-on module Vissig to define and configure signal groups. The Network editor allows you to add signal heads to the Vissim network. You assign each signal head a signal group of your choice.

Notes: DLL files for external control procedures are provided to you together with your Vissim version as 64-bit version, depending on the license purchased.

If your license includes the SC type **External**, you need to compile the DLL files according to your 64-bit Vissim version.

Vissim only allows you to access data of an external signal controller, if you have saved the network file. This way, it is made sure that the controller files and the network file **.inpx* are saved to the same directory.

6.14.1 Modeling signal groups and signal heads

Signal groups are the smallest control unit belonging to a controller device assigned a unique SC number in Vissim. You can assign an LSA up to 125 signal groups, depending on its control logic. As signal groups are not visible in reality, Vissim also distinguishes between signal heads and signal groups. A signal group can be assigned to several signal heads that always show the same picture.

Vehicles approaching a red signal will stop at a standstill distance in front of the signal controller (see "Editing the driving behavior parameter Following behavior" on page 286).. The default standstill distance is 0.5 m.

Vehicles approaching an amber signal will cross the intersection if they cannot come to a safe standstill at the stop line.

Optionally, you can select a method that calculates the probability of the vehicle stopping or continuing. For this calculation, three driving behavior parameters are required (see "Defining driving behavior parameter sets" on page 282).

By default, signal groups and thus all signal heads are updated at the end of every simulation second. If you have specified more than 1 time step per simulation second, signal groups are not switched at each time step. This means that in the network display, each signal header will show a picture that vehicles on its link will respond to in the next time step. Depending on the signal header type, switching is also possible every 1/10 s.

Signal headers allow you to model all common situations with precision. This includes different types of signal groups for several vehicle types on one lane. For instance, for a bus with its separate signal phases, driving on a mixed lane, select the vehicle class of your choice for each signal head.

Note: For any SC with conflicting traffic that is allowed to drive at the same time, you need to define priority rules (see "Creating priority rules" on page 541).

6.14.1.1 Defining signal heads

In the network, you may position signal heads at the stop line accurate to lane level. In Vissim, signal heads are displayed as red lines per default. To model 3D signal heads realistically for a simulation, including the display options for mast, arm, signal head, sign and light, define 3D signal heads (see "Modeling 3D signal heads" on page 584).

1. Ensure that a SC with signal groups is defined (see "Defining SC and signal control procedures" on page 602).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

2. On the Network objects toolbar, click **Signal Heads**.
3. Hold down the CTRL key and right-click on the desired position of the signal head in the link.

   *A colored bar is added. The **Signal Head** window opens.*

4. Edit the attributes (see "Attributes of signal heads" on page 579).
5. Confirm with **OK**.

*The attributes are saved in the list **Signal Heads**.*

6.14.1.2 Attributes of signal heads

The **Signal Heads** window opens when you insert a network object and have selected to have the Edit dialog automatically opened after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Signal Heads list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

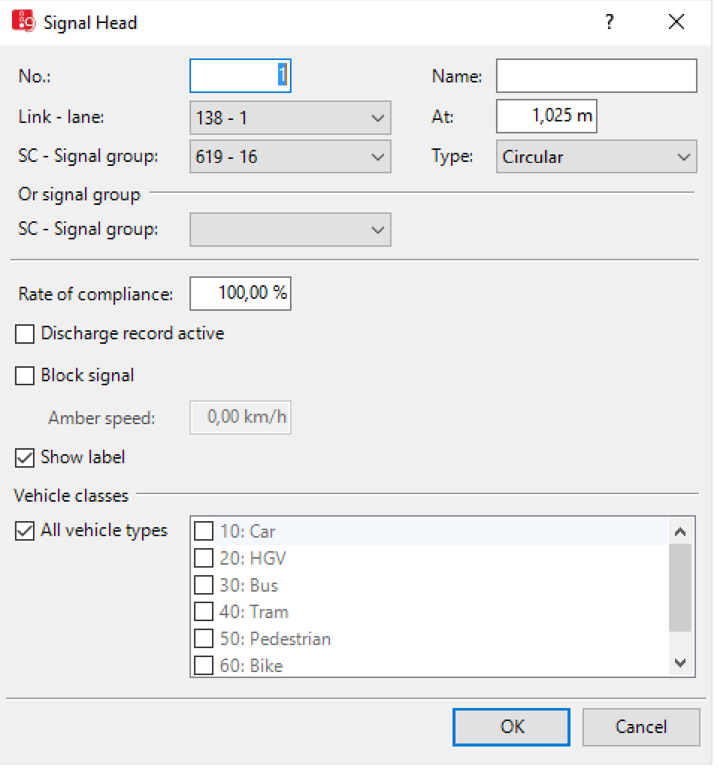
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



1. Make the desired changes:

Element	Description
No.	Unique **number**
Name	Designation
Link	Link onto which the signal head has been inserted
Lane	**Ln**: Number of lane to which you have added the signal head.
At	**Position** (**Pos**): Distance from start of the link or connector
SC - signal group	Number and name of respective signal controller **SC** and respective signal group **SG**
Type	Display of the signal head in 3D animation during a simulation run or test run: - Circular - Left arrow - Right arrow - Invisible: Hides the signal heads (in 2D mode as well).

Element	Description
	If the normal signal group of an arrow signal head has the signal state **Red** or **Off** and an Or signal group is defined for this signal head, the signal state or the Or signal group is shown as a bar without an arrow. This also applies if the normal signal group is yellow and the Or signal group is red-amber or reversed.
Or signal group	**OrSG**: Is the **Or Sig.gr.** option: Second signal group which shall influence this signal head. Such a signal head shows exactly in green if at least one or both signal groups are green. Shows the first signal group in red, shows the signal group the picture of the second signal group, also in amber or red-amber. For the display of signal states of both individual signal groups, a short link with a signal head on both signal groups can be created next to the link. - **SC**: Number of SC or the Or signal group - **Signal group**: Number of Or signal goup You can use the **Or signal group** to model vehicles turning right with green right arrows, which are indicated by their own signal group before and afterwards and by a circular symbol during the actual phase as contractually stipulated. Define two signal heads on different links: - a signal head for vehicles traveling straight ahead; - a signal head for vehicles turning right. This must be located on a connector not used by vehicles traveling straight ahead. In the signal group of the green arrow for vehicles turning right, enter the signal group of the vehicles traveling straight ahead as an **Or signal group**.
Compliance rate	**ComplRate**: Every vehicle and every pedestrian has an individual random number. This number is between 0.0 (0%) and 1.0 (100%) and is evenly distributed. If this random number is greater than the compliance rate of a signal head, the vehicle or the pedestrian will ignore the respective signal head. Minimum value: 0.0 = 0% Maximum value: 1.0 (default value) = 100% If the compliance rate is below 100%, use conflict areas to model intersection control (see "Using conflict areas" on page 560).
Discharge record active	**DischRecAct**: If this option is selected, the vehicles of this signal head area are taken into account in the discharge record (see "Saving discharge record to a file" on page 1024).
Block signal	**Is block signal** (**IsBlocksig**): Selecting this option defines the signal head as a block signal (see "Modeling railroad block signals" on page 673). The fields **SC**, **Signal group** and **Or signal group** are deactivated.
Amber speed	**Block signal amber speed** (**vAmberBlock**): Speed assigned to a train when it travels past the block signal and the state of the block signal is Amber (see "Modeling railroad block signals" on page 673).

Element	Description
	The **Block signal** option must be selected. Default value 0 km/h.
Label	☐ If this option is not selected, the label for the signal head is hidden if the label for all signal heads is selected.
Vehicle classes	**VehClasses**: Vehicle classes for which the signal head is valid. For example, you can define a separate signal for buses on a link, which should be ignored by private transportation. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
Pedestrian classes	**PedClasses**: If the signal head is defined on a link for which the attribute **Is pedestrian area** (option **Use as pedestrian area**) is selected: For pedestrian simulation, the **Pedestrian classes** for which the signal head is valid.
Slow down distance	**SlowDownDist**: In list only: Distance from stop line at which pedestrians start to reduce their speed in order to stop at the stop line. Default 3 m.

2. Confirm with **OK**.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Pedestrian Classes (see "Attributes of pedestrian classes" on page 879)

The attributes are described further above.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.14.1.3 Zooming into signal heads and detectors of an SC

In the Network Editor you can enlarge an area that contains signal heads and detectors of an SC.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** table is opened.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Zoom**.

6.14.2 Modeling 3D signal heads

In a 3D simulation, you can model 3D signal heads realistically due to the extensive display options for mast, signal arm, signal head, sign and light.

6.14.2.1 Defining 3D signal heads

Note: You can define network objects of the network object type **3D Signal Heads** in 2D mode and in 3D mode. Other **signal heads** may only be defined in 2D mode (see "Defining signal heads" on page 579).

1. On the Network objects toolbar, click **3D Signal Heads**.

Notes:

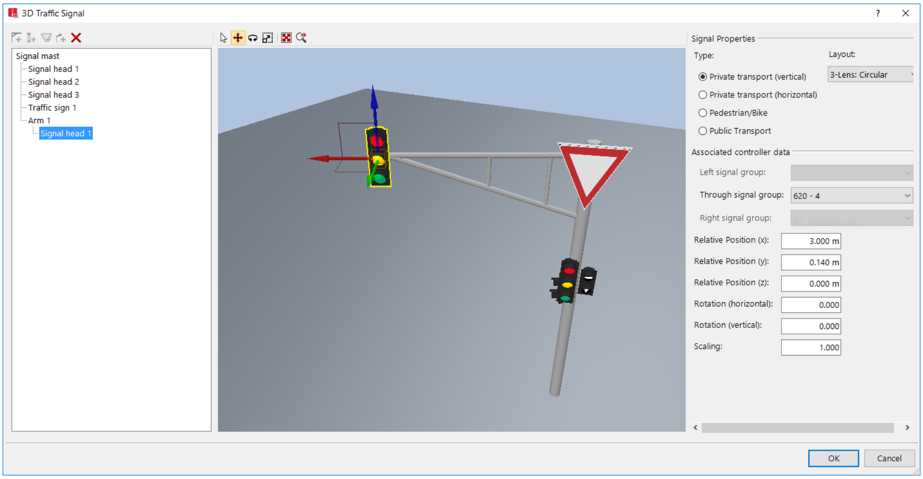
- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

2. In the Network Editor, hold down the CTRL key and right-click the desired position in the link where you want to add the 3D signal head.

   *The **3D Traffic Signal** window opens. In the window, add an arm, a signal head, a sign and/or light to the signal mast. Then arrange the objects and edit their attributes (see "Attributes of 3D signal heads" on page 587). The **3D signal head** window contains the following sections:*

- Section on the left: default entry **Signal mast**. If you have added an arm, signal head, sign, or light to the signal mast, this is indicated below it. The number added to the name facilitates the identification of the element.
- **3D view** in the middle: displays a preview of the signal mast with the elements added. Above the 3D view, there is a toolbar with functions that allow you to edit the elements.
- Section on the right: If an element has been selected in the section on the left or in the preview, the attributes of this element are displayed here (see "Attributes of 3D signal heads" on page 587).

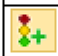
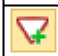
*The figure shows the **3D Signal Head** window, displaying a signal mast with an arm, three signal heads and a traffic sign. Several attributes of these elements were edited. The signal head at the signal arm is selected. On the toolbar, you have enabled the **Stage based editing** button.*



Adding arms, signal heads, signs, or street lights

You can align one or several of these elements with the signal mast and edit their attributes. If you add an arm, you can align one or multiple elements with it.

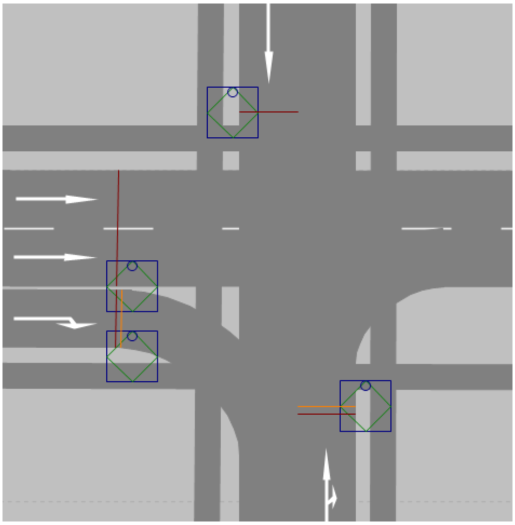
1. In the section on the left, click **Signal mast**.
2. If you have already added an arm and would like to align an element with it, in the section on the left, below the **Signal mast <No>**, click the **SignalArm <No>**.
3. On the toolbar, below **3D signal head parts**, click the desired toolbar button:

	**Arm**
	**Signal head**
	**Sign**
	**Street light**

4. Confirm with **OK**.

*The attributes are saved in the **3D signal heads** list.*

In the 2D mode, in the Network Editor, the contour or outline of the 3D model is displayed.



Legend	
Element	**Description**
Square	3D signal head not selected: green with a red circle in the inner square 3D signal head selected: dark blue in 2D mode 3D signal head selected: light blue in 3D mode
Circle	Mast
Red line	Direction and length of signal arm
Blue arrows on the signal arm	Signal Heads
Navy blue circles	In 2D mode: Signal head directly on the mast

In the 3D mode, in the Network Editor, the 3D model is displayed and selected.

Zooming into the 3D view

- Enlarging the view: Rotate the mouse wheel forward.
- Minimizing the view: Rotate the mouse wheel backward.

Editing elements in the 3D view

Functions available on the toolbar:

	Select	You can click an element to select it. The attributes of the selected element are displayed.
	Move	Colored arrows are displayed next to the element. They indicate movement towards the x-axis, y-axis and z-axis. Click an arrow, hold the mouse button down and move the element.
	Rotate	The element selected is circled with colored rings in vertical and horizontal direction. Click a ring, hold the mouse button down and rotate the element.
	Scale	For the element selected , lines and cuboids are displayed along the x-axis, y-axis and z-axis. Click a cuboid, hold the mouse button down and enlarge or reduce the size of the element.
	Reset view	After you have zoomed in or out and wish to center the entire signal header and show its maximum view.
	Zoom to selected element	Enlarge the view of the selected element.

Deleting arms, signal heads, signs, or street lights

1. In the section on the left, under **Signal mast**, click the **SigalArm <No>**, **Signal head <No>**, **Traffic sign <No>**, or **Street light <No>** of the element you want to delete.
2. On the toolbar, click the button.
3. Confirm with **OK**.

6.14.2.2 Attributes of 3D signal heads

The **3D signal head** window opens automatically when you insert a network object and have selected to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the 3D Signal Heads list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

- In the preview, click the desired element or in the section on the left, click the desired entry **Signal mast**., **SigArms <No>**, **Signal head <No>**, **Sign <No>**, or **Light <No>**.

  *In the preview, the element is selected. In the section on the right, the **Properties** are displayed.*

1. Make the desired changes:

Mast properties	**Description**
Type	**Mast style** (**MastStyle**): Surface of the mast
Height	Vertical length of the mast, default 3,000 m
Diameter	**MastDiam**: Mast diameter, default 0.14 m
Color	Click in the box to define or select a color.
Relative position (x)	Moves mast towards the x axis of the insert position. Negative values move the mast to the left. The value used is expressed in the Network Editor, in 2D, by the extent to which the green diamond is moved from the blue square.
Relative position (y)	Moves mast towards the y axis of the insert position. Negative values move the mast downwards. The value used is expressed in the Network Editor, in 2D, by the extent to which the green diamond is moved from the blue square.
Relative position (z)	Moves the mast vertically. Negative values move the mast downwards.
PosZOffset	**Position (z-offset)**: Base height of the mast foot, for example, > 0 for a mast on a bridge
Scale	Zooms in or out of the 3D model in the Network Editor, default value 1000

Arm properties	Description
Type	- Basic: vertical tube without diagonal frame - Double framed (down): vertical tube with diagonal frame that runs from the end of the arm, diagonally downward to the signal mast - Double framed (up): vertical tube with diagonal frame that runs from the end of the arm, diagonally upward to the signal mast
Flipped	☑ Select this option if you want the arm to face right. ☐ Deselect this option if you want the arm to face left.
Length	Default 5,000 m

Signal properties	Description
Type	- Traffic (vertical) - Traffic (horizontal) - Pedestrian/Bike - Public Transport Depending on the **Type**, in the **Layout** list, you can select different layouts for signal heads.
Layout	From a list of defined signal groups, select a signal group for the desired direction of travel

Defining layouts with counters

By default, counters show the current remaining red and green times of a signal group within a cycle for fixed time controllers. Counters assigned to a 3D traffic signal with a flashing hand or **Don't Walk** sign indicate the remaining time of the flashing signal. This corresponds to amber time.

Counters can also be used in combination with signals for the desired vehicle type. Counters are activated as soon as they are linked to a signal group.

Counters have the following properties:

- The counter is red as long as the signal group is red or red/amber.
- The counter is green as long as the signal group is green, but not if it is flashing green, for example.
- The counter is disabled (dark) during all other states of a signal group, for example, amber, flashing red, flashing amber, flashing green.
- The red and green times displayed (start times of the counter) result from the times of the last red or green phase of the signal group. Thus the red/amber state also belongs to the red phase. The counter therefore always starts with the value of the last duration of the respective state. The counter is not enabled if the duration was zero.

- Counters do not work correctly if more than one red or green phase is defined within a cycle, for example, with a double throw, because a counter then also starts with the duration of the last green or red phase.

Notes:
- In the first and second cycles after the start of the simulation, counters may display no values or incorrect values because a counter uses the respective value of the duration of a state from the previous cycle. This also applies to the first and second cycle after each signal program change.
- Only use counters with a fixed time controller. If you use counters with a traffic-dependent signal control, the counters display incorrect times due to the unforeseeable red and green times.

Traffic sign properties	Description
Shape	Geometric shape
Width	Length of the longest horizontal expansion of the shape, default 1000 m
Height	Length of the longest vertical expansion of the shape, default 5000 m
Inner area width	For traffic signs with a frame: horizontal expansion of the inner area width, default 0.950 m
Inner area height	For traffic signs with a frame: vertical expansion of the inner area width, default 0.950 m
Inner area position (x)	Moves inner areas towards the x-axis, default 0.025 m
Inner area position (y)	Moves inner areas towards the y-axis, default 0.025 m
Inner area position (z)	Moves inner areas vertically, default 0.025 m
Texture	Select a graphics file with texture for display
Border Color	Click in the box to define or select a color.
Background color	Click in the box to define or select a color for the inner area.

Street light properties	Description
Length	Default 3,000 m
Flipped	☑ Select this option if you want the streetl ights to face right. ☐ Deselect this option if you want the street lights to face left.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Signal arms: Signal arms assigned using the **Add Arm** button
- Signal heads: Signal heads assigned using the **Add Signal** button
- Streetlights: Streetlights assigned using the **Add Light** button
- Traffic signs: Traffic signs assigned using the **Add Sign** button

The attributes are described further above.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.14.2.3 Editing 3D signal heads in the Network editor

You can move and rotate 3D signal heads in 2D mode in Network Editors.

You can edit 3D signal heads in 2D mode and 3D mode in the **3D Traffic Signal** window.

Editing 3D signal heads in 2D Mode

In 2D mode, the mast, signal arm, signal head, sign and light are represented by various symbols (see "Modeling 3D signal heads" on page 584). You can select and move these symbols in the Network Editor.

1. In the Network Editor, click with your mouse on the symbol of the component, hold down the mouse button and drag the cursor to the desired position.
2. Release the mouse button.

Editing 3D signal heads in the 3D Traffic Signal window

1. In the Network editor, double-click the desired 3D signal head.

   *The **3D Traffic Signal** window opens. The selected component of the 3D signal head is displayed in the **3D Traffic Signal** window.*

2. Make the desired changes:

Element	Additional key	Function	Key
Rotate view	–	Click in the background, hold down mouse button and drag cursor	Arrow keys or E,D,S,F
Rotate component	CTRL	Click on component, hold down mouse button and drag cursor	–
Zoom 3D Signal Head	–	Turn the scroll wheel of the mouse	PAGE UP PAGE DOWN

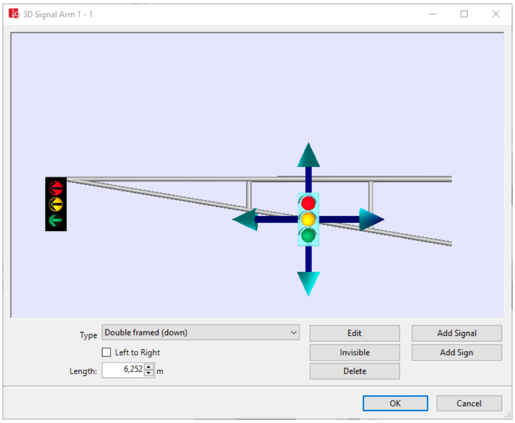
*The changes are displayed in the **3D Traffic Signal** window.*

3. To edit one of the components, double-click the desired component in the **3D Traffic Signal** window.
4. The **<Name component>** window opens (see "Attributes of 3D signal heads" on page 587).
5. Make the desired changes:

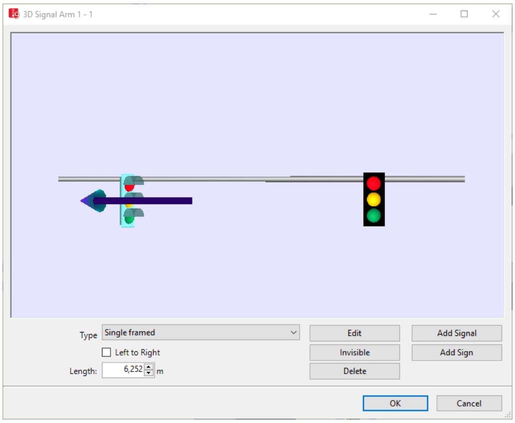
Function	Additional key	Action with the mouse
Move component	SHIFT	Click and drag object
Rotate component around center	CTRL	Click and drag object
Scale component	–	Select object, then turn scroll wheel of mouse

If you want to move or rotate other components, arrows indicate the direction in which you can execute a function:

Move:



Rotate:



6.14.3 Using detectors

You can use detectors to record vehicles or pedestrians for vehicle-actuated signal controls.

A detector must be assigned to the desired SC and can be limited to the PT lines of your choice (see "Attributes of detectors" on page 594).

To record traffic volumes in your Vissim network, use data collection points and evaluate them with data collection measurements (see "Defining data collection points" on page 446), (see "Evaluating data collection measurements" on page 1093).

6.14.3.1 Using detectors to record vehicles

In order to record vehicles for traffic-actuated signal control, various types of detectors are deployed, such as induction loops, infra-red sensors, pedestrian push buttons, overhead wire contacts, point-blocking circuits or reporting points for radiotelegrams.

In Vissim, you model detectors as network objects on links for which you enter a length. A message impulse is then transmitted to the signal controller as soon as the front of a vehicle reaches the detector. Another impulse is transmitted as soon as the tail of the vehicle leaves the detector. These message impulses are interpreted by signal control logic and converted into appropriate switching signals for control.

6.14.3.2 Using detectors to record pedestrians

Vissim recognizes pedestrians on pedestrian links by means of detectors. If a pedestrian leaves a pedestrian link or enters a free pedestrian link, Vissim detects this at the end of a SC time step. The change from **occupied** to **vacant** or **vacant** to **occupied** are transmitted to the control logic as the **front ends** and **rear ends**. If you have selected a sound file, it is started when the first pedestrian arrives at an empty detector.

6.14.3.3 Defining detectors

You must have defined at least one SC (see "Defining SC and signal control procedures" on page 602).

1. On the Network objects toolbar, click **Detectors**.
2. With the mouse pointer, point to the position in the link at which the detector is to begin.
3. Hold down the CTRL key and the right mouse button, and drag the pointer to the desired end position.
4. Release the keys.

   *The detector is added. The **Detector** window opens.*
5. Edit the attributes (see "Attributes of detectors" on page 594).
6. Confirm with **OK**.

*The attributes are saved in the **Detectors** list.*

6.14.3.4 Attributes of detectors

The **Detectors** window opens automatically when you insert a network object and have selected to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Detectors list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

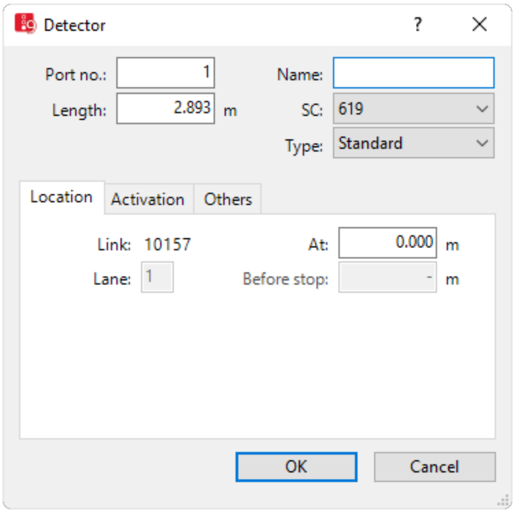
The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
Port no.	Physical **Port number** (**PortNo**) of the detector, which identifies the detector in the control procedures. If multiple detectors of SC (signal control) are available via the same number, they behave as if they are connected in parallel to one entry port of the control device (logical OR-operation).
Name	Designation
Length	Length of the detection range of a detector. The value 0.000 is e.g. permissible and useful for modeling trolley wire contacts and pedestrian sensors. These are represented in the network as thin horizontal lines.
SC	SC to which detector is assigned. If in the **Type** box, > **PT Calling Pt.** is selected, the **SC** box is deactivated. PT calling points do not belong to a specific SC.
Type	**Type**: Select detector type (see "Modeling PT lines" on page 518): - **Standard**: Standard detectors detect vehicles, including PT vehicles. - **Pulse**: Impulse detectors do not send information regarding occupancy to the control procedures. - **Presence**: does not send information regarding the impulse via the front end or back end of the vehicle to the control procedures. - **PT calling pt** (**PT calling point**): only records PT vehicles that send PT telegrams.

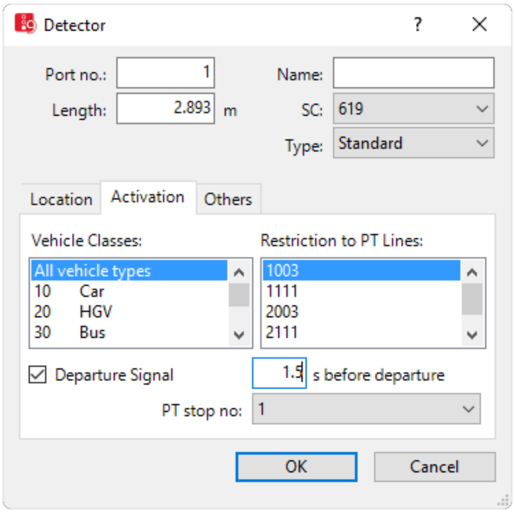
Location tab



The list in the tab contains, amongst others, the following attributes:

Element	Description
Link	Link, in which the detector is located
Lane	**Ln**: Number of lane on which the detector is installed.
At	**Position** (**Pos**) Distance from start of the link or connector
Before stop	if a signal head exists: Distance of the front end of the detector to the next signal head of the SC (signal control) on its lane

Activation tab for vehicles

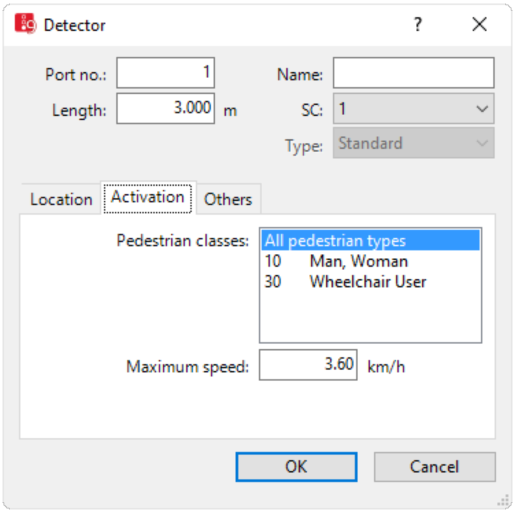


The list in the tab contains, amongst others, the following attributes:

Element	Description
Vehicle Classes	**VehClasses**: Vehicle classes detected by detector
Restriction to PT Lines:	**PTLines**: One or multiple PT lines, for which the detector shall be relevant. Thus vehicles of these PT lines are only detected if their vehicle class is selected.
Departure Signal	If this option is selected, the detector triggers an impulse for closing the doors of the PT vehicle under the following conditions: - If a PT vehicle is located on a detector and stops at the stop specified in the **PT stop** box, plus the boarding and alighting is completed in the first time step after the time defined in the **s before departure** box or before then. - When a PT vehicle is located on a detector that has already decided to omit the PT stop selected in the **PT stop** box. The impulse is then triggered upon reaching the detector. The time the door is closed can thus be determined via the detector.

Activation tab for pedestrians

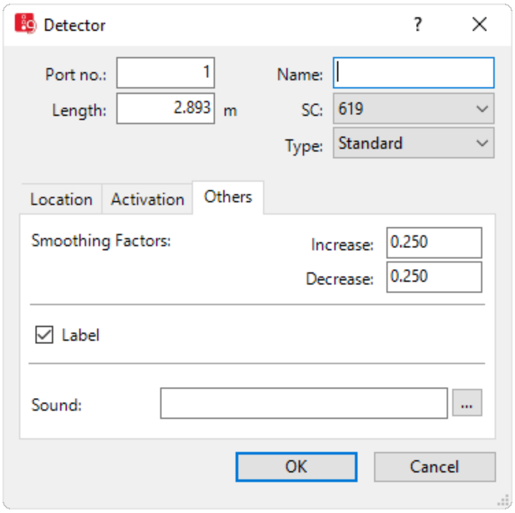
If the detector is defined for a link whose attribute is **Is pedestrian area** (option **Use as pedestrian area**), the detector serves for modeling calls via a pedestrian crossing button and pedestrian classes are displayed.



1. Make the desired changes:

Element	Description
Pedestrian Classes	Pedestrian classes detected by detector. A pedestrian is recognized by the detector only if the pedestrian type belongs to a selected pedestrian class.
Maximum speed	The detector recognizes pedestrians walking slower than this speed as pedestrians requesting a green light.

Others tab



1. Make the desired changes:

Element	Description
Smoothing Factors	**SmthFactDecr, SmthFactIncr**: Factors for the exponential smoothing of occupancy values, which is conducted for individual control procedures. (see "Exponential smoothing of detector occupancy rates" on page 601). - **Smoothing factor decrease**: Factor used whien the detector is not occupied (attribute **Presence** = **Passive**). - **Smoothing factor increase**: Factor used when the detector is occupied (attribute **Presence** = **Active**).
Label	☐ If the option is not selected, the label for the detector is not displayed, even if the label for all detectors is selected.
Sound file (Sound)	**SoundFile**: Wavetable sound file **.wav*, which is played each time a vehicle is detected by the detector. This file is saved in the same folder as the network file **.inpx*. A sound card or suitable Microsoft Windows driver must be installed.

2. Confirm with **OK**.

The network object has additional attributes that you can show in the Attributes list. Among them are the following for example:

Element	Description
Presence	**Presence state**: - **Active**: A vehicle is located on the detector and is detected. This also applies for pedestrians. - Always **Passive** for detectors of the type **Impulse** and **PT Calling Pt.**.
	A vehicle is located on a detector, if the following conditions apply: - The vehicle is located on the same link as the detector. - The front edge of the vehicle has passed the start section of the detector: The **Position** (**Pos**) attribute of the vehicle is greater than the **Position** (**Pos**) attribute of the detector. - The rear edge of the vehicle has not yet passed the end section of the detector: The **Position** (**Pos**) attribute of the vehicle, minus the length of the vehicle (rear edge), is smaller than the **Position** (**POS**) attribute of the detector, plus the length of the detector (end position).

Element	Description
	A pedestrian is considered on a detector, if the center of the pedestrian is located within the rectangle that defines the length of the detector and the width of the lane on which the detector is located. Ensure that the detector is at least as long as the distance a pedestrian can walk within a simulation time step back. In addition, account for the maximum possible speed of the pedestrian at this position. If the detector dimensions are too short, meaning that in one time step, the center of the pedestrian will be in front of the detector and in the the next time step, the pedestrian will be behind it (having already passed it), the detector will not be able to detect the pedestrian. *In front of* and *behind* refer to the visually assessed walking direction on the pedestrian link. As pedestrians have no link coordinate, Vissim cannot determine whether the walking direction is towards or away from the detector.
Detection	**Detection state**: - **Active** for detectors of the types **Standard** and **Occupancy**, as long as the **Presence** attribute is active. - **Active** for detectors of the types **Standard** and **Impulse** after a vehicle or pedestrian occupies or leaves a detector, until the signal controller resets the detection state to **Passive**. - A fixed time control resets the value of the **Detection** attribute after each SC-time step. - Always **Passive** for detectors of the type **PT Calling Pt**.
Impulse	**Impulse state**: - **Active**, as soon as a vehicle or pedestrian reaches a detector that is not occupied. The impulse state remains **Active** until the signal controller resets the impulse state back to **Passive**. - A fixed time control resets the value of the **Impulse** attribute after each SC-time step. - Always **Passive** for detectors of the type **Occupancy** and **PT Calling Pt.**.
Gap time	**Time**: Period [s] after the attribute **Presence** (**Presence state**) had the value **Active**. 0 = **Presence** attribute is **Active**.
Occupancy	**Occ**: Period [s] that has passed since occupancy of the detector. 0 = The detector is not occupied (**Presence** attribute = **Passive**). Always 0 for detectors of the type **PT Calling Pt**.
Occupancy rate	**OccupRate**: Proportion of time the detector was occupied during the last simulation second. The occupancy rate is exponentially smoothed based on each simulation second. Value range 0 to 100 %

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Vehicle classes (see "Using vehicle classes" on page 280)
- Pedestrian classes (see "Using pedestrian classes" on page 879)
- Public transport lines (see "Modeling PT lines" on page 518)

The attributes are described further above.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.14.3.5 Exponential smoothing of detector occupancy rates

You can use exponential smoothing to smooth the occupancy rate of a detector. This is necessary as detectors are either fully occupied or not occupied and thus do not provide sufficient information for signal control decisions. Exponential smoothing allows you to calculate the occupancy rate with help of the following equation, using the last t seconds:

$$s(t) = \alpha \cdot x + (1 - \alpha) \cdot s(t - 1)$$

Thereby the following applies:

$s(t)$	new, exponentially smoothed value
$s(t-1)$	old, exponentially smoothed value (1 second ago)
x	new detector value
α	Smoothing factor [0 to 1]

This means the new, exponentially smoothed value is the weighted average of the new, detected value and the exponentially smoothed value after the last simulation second.

The new detector occupancy rate is weighted with alpha.

The old, smoothed value is weighted with $(1 - \alpha)$.

In Vissim, you can use the following attributes to enter different values for alpha (see "Attributes of detectors" on page 594):

- **Increase**: For increasing x values ($x > s(t-1)$)
- **Decrease**: For decreasing x values ($x < s(t-1)$)

The exponentially smoothed occupancy rate represents a kind of a floating average of the detected values of all previous time steps. Thereby the most current values are weighted the strongest. With $\alpha = 1$, there is no smoothing. The equation then equals the new detector value x.

In the Vissim network, in the attribute list **Detectors**, you can show values of the attributes **Occupancy rate** (**OccupRate**) in % and **Occupancy** (**Occup**) i (see "Editing attributes in a list" on page 350).

6.14.4 Using signal control procedures

You can define new SCs with your signal groups. To do so, for the control procedure, select a type. This e.g. specifies whether you want the SC to be fixed-time controlled or controlled via a control procedure. Depending on your Vissim license, you can access add-on modules with external signal control procedures (see "Showing licensed signal control procedures" on page 607).

Determining the cycle second for a fixed cycle time

Vissig determines the current cycle second for the SC type **Fixed time**.

For the SC types VAP, TRENDS, VS-PLUS and Siemens VA, Vissim determines the current cycle second.

- In Vissim, the first cycle starts at 00:00:00. At the simulation start, the cycle second is calculated based on this start time.
- If the start time specified in the simulation parameters is 00:00:00, the cycle and simulation start at the same time.
- If the start time specified in the simulation parameters is not 00:00:00, at the simulation start, the current cycle second may have a different value than 0, as the first cycle was started at 00:00:00.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

6.14.4.1 Defining SC and signal control procedures

1. From the **Signal Control** menu, choose > **Signal Controllers**.

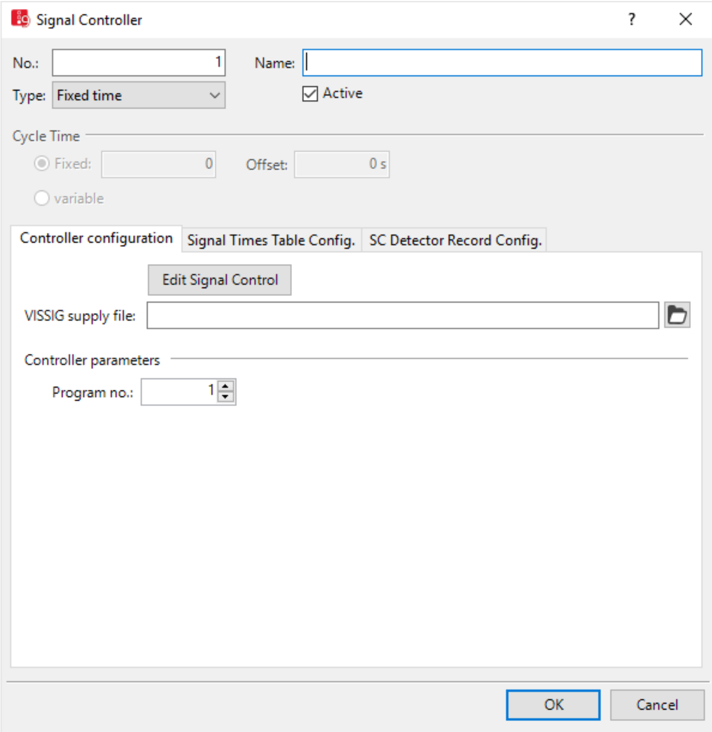
   *The **Signal Controllers** list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. In the list, on the toolbar, click the **Add** button ✚.

A new row with default data is inserted.

*The **Signal Controller** window opens.*



3. In the **Signal Controller** window, in the upper section, enter the desired basic attributes (see "Attributes of SC" on page 604).
4. Depending on the signal controller selected in the **Type** list, in the **Controller configuration** tab, make the desired settings. These vary depending on the type selected. For further information, read the following sections:

- Using the SC type Fixed time (see "Using SC type Fixed time" on page 607).
- In the description of the signal control procedure selected (see "Modeling signal controllers" on page 577), (see "Overview of add-on modules" on page 38).

5. If for the types **Fixed time**, **Epics/Balance-Central** or **External** you want to create or edit signal programs using Vissig, click the **Edit SC** button (see "Opening and using the SC Editor" on page 631).
6. To evaluate simulation data, in the tabs **Signal Times Table Config.** or **SC Detector Record Config.**, make the settings of your choice depending on the control procedure (see "Showing signal times table in a window" on page 1098), (see "Evaluating SC detector records" on page 1070).
7. Enter the desired data.

8. Confirm with **OK**.
9. Save the network file **.inpx*.

You can edit the attributes in the attributes list (see "Attributes of SC" on page 604).

6.14.4.2 Attributes of SC

The **Signal Controller** window opens when you define an SC and have selected that you want the program to automatically open the Edit dialog after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Signal controllers list is opened.

Into the **Signal Controller** window, you enter attribute values for the signal controller. For a signal controller which has already been defined, you can call the window using the following function:

- In the **Signal Controllers** list, double-click the row with the desired SC.

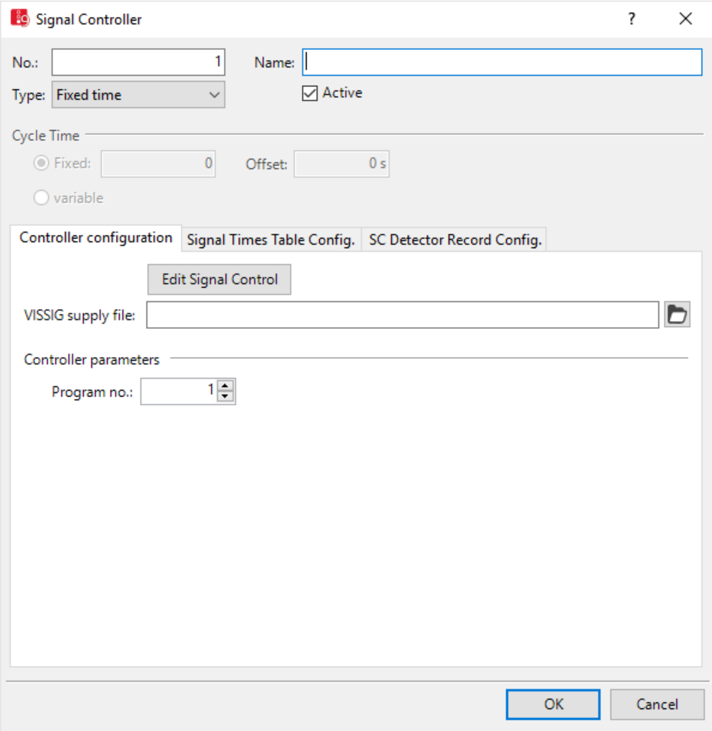
The network object may have additional attributes. You can show all attributes and attribute values of a network object in the **Signal Controllers** list. You can open the list via the following function:

- From the **Lists** menu, choose > **Intersection Control** > **Signal Controllers**.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

Every signal controller can have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



Basic attributes of SC

The basic attributes of the network element are shown in the upper area of the window and in the list of network objects for the particular network object type.

Element	Description
No.	Unique number of the SC
Name	Description
Active	Enables or disables the SC during simulation
Cycle time	Optional cycle time (**CycTm**). Duration in seconds. Only for control procedures **Siemens VA**, **TRENDS**, **VAP**, **VS Plus**, as for these the cycle time is not set externally.
variable	The cycle time is variable (**CycTmIsVar**)
Type	Control procedures for SC
Offset	The times of the first cycle and all others are moved back by the offset time in [s].

Under them, the following tabs are displayed:

- **Controller configuration**: Select a procedure-dependent control file
- **Signal Times Table Config.**: Configure signal times table that runs during the simulation (see "Showing signal times table in a window" on page 1098).
- **SC Detector Record Config.**: Configure SC detector record for SC with external control procedure (see "Evaluating SC detector records" on page 1070).

For the control procedures **Siemens VA**, **TRENDS**, **VAP** AND **VS-Plus**, the **Signal groups** tab is displayed, as for them the signal groups are not set externally (see "Defining signal groups for Siemens VA, TRENDS, VAP, VS-Plus" on page 630).

Showing and editing dependent objects as relation

The **Signal Controllers** list is displayed as the left of two coupled lists.

In the list on the left, you can select an SC. In the list on the right, you can show the objects assigned to it. Depending on the procedure type (**Type** attribute) used, there are the following restrictions:

- Relations are not available for all procedure types.
- Relations cannot be edited as attribute values with all procedure types.

1. On the list toolbar, in the **Relations** list, click the desired entry.

- Detector attributes (see "Attributes of detectors" on page 594)
- Attributes of SC detector record configuration (see "Evaluating SC detector records" on page 1070)
- Attributes of signal groups: Select **Normal**, **Flashing**, or **Green arrow**
- Attributes of the signal times table configuration (see "Configuring signal times table on SC" on page 1100)
- WTT file assigned (see "Other files" on page 1199)

The right-hand list is shown. If there is no assignment, only the column titles are shown.

6.14.4.3 Changing control procedures for SC type

You can choose between a traffic-dependent procedure and **Fixed time** for the SC type.

Depending on the type, parameters such as amber time, red amber time, start of green may be missing. Add these after changing the control procedure.

Parameters of the old type which are no longer required are deleted. If you switch back to the old type, you must re-enter these parameters.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit...**.

   The ***Signal Controller*** *window opens.*

4. In the **Type** list box, click the control procedure of your choice.

5. Add the missing data in the tabs.

6.14.4.4 Showing licensed signal control procedures

- From the **Help** menu, choose > **License**.

The licensed signal control procedures are highlighted in the ***Signal Controllers*** *section.*

6.14.4.5 Using SC type Fixed time

The SC type **Fixed time** allows you to define fixed time signal controllers. To edit the signal plan, you use the graphical SC editor. Depending on the Vissim license, you have add-on functions in the SC editor with Vissig, which deviate from the standard version of Vissim (see "Opening and using the SC Editor" on page 631).

Vissig determines the current cycle second for a fixed cycle time

- In Vissim, the first cycle starts at 00:00:00. At the simulation start, the cycle second is calculated based on this start time.
- If the start time specified in the simulation parameters is 00:00:00, the cycle and simulation start at the same time.
- If the start time specified in the simulation parameters is not 00:00:00, at the simulation start, the current cycle second may have a different value than 0, as the first cycle was started at 00:00:00.

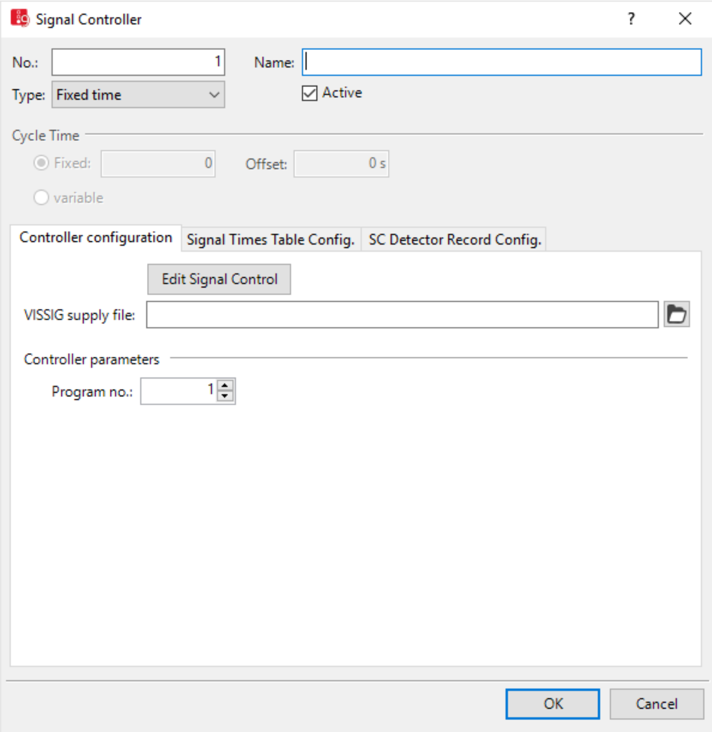
1. From the **Signal Control** menu, choose > **Signal Controllers**.

   The ***Signal Controllers*** *list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   The ***Signal Controller*** *window opens.*



4. In the **Type** field, select > **Fixed time**.

Note:

- The **Cycle time** section is not enabled. You cannot edit the base attributes **Cycle time** and **Offset** in the **Signal controller** window. To change these attributes, click the **Edit Signal Control** button. Then in the navigator, select **Signal programs**. In the table, you can edit **Cycle time**, **Offset**, and **Switch point** (see "Opening and using the SC Editor" on page 631).
- The current cycle second (determined by Vissim for a fixed cycle time) is calculated based on the start of the first cycle at midnight. This makes no difference when 00:00:00 is set as the start time.

5. Open the **Controller configuration** tab.
6. Make the desired changes:

Element	Description
VISSIG supply file	Signal control file **.sig* in XML format. Notes: - When importing older Vissim network files, the files **.sig* are automatically created and stored in the directory in which the network file **.inpx* is stored. - When you duplicate a SC, the original SC and the duplicate both use the same signal control file **.sig*. This means the changes you make to a SC affect all SCs that use the same signal control file **.sig*.
Program no.	**ProgNo.**: The signal program or daily signal program list that you want to simulate (see "Defining and editing daily signal program lists" on page 664). A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.
Edit Signal Control button	The signal program editor opens (see "Opening and using the SC Editor" on page 631). The signal groups must be created or deleted in the SC Editor. Changes in the channel number of existing signal groups are detected in Vissim and adjusted accordingly in the corresponding signal heads. Note: The SC frequency is loaded from the external data of the controller. Internally, the SC frequency is the smallest integer multiple of all SC frequencies. The value may not exceed 10 because Vissim allows a maximum of ten simulation steps per second.

The control procedure additionally uses the following files:

Element	Description
Program file	**ProgFile**: File with control logic, *VISSIG_controller.dll* by default
Dialog DLL file	**GUIFile**: File name of supply interface, *VISSIG_GUI.DLL* by default
Supply file 1	**SupplyFile1**: Configuration file, *vissig.config* by default. Note: If the supply files are stored in the *Exe* directory of your Vissim installation, they are automatically loaded.
WTT files	**WTTFile**: Value type tables: Contain the data types of the control logic and the type of display in the SC detector record or in the **Signal times table** window. *VISSIG.wtt* by default.

Performing green time optimization of stage-based fixed time controllers

Note: You will need the add-on module Vissig.

You can improve the quality of the signal times table of a selected Vissig SC or all Vissig SCs with the green time optimization of stage-based fixed time controllers.

To this end, Vissim repeatedly calculates simulations of the entire network. All controllers are disabled in the process, except the SCs selected. Thus upstream SCs have no effect. The simulations are continued as long as changes in green times of the stages lead to an increase in the flow (volume) or to a reduction in the average vehicle delay. You can also cancel the iteration. The stage lengths with the best result have the highest flow and the lowest average vehicle delay and are stored in the **.sig* file after the optimization.

Sequence of optimization in Vissim

- Vissim determines the average delay of all vehicles that have passed through the nodes on the lanes with signal heads of the signal group, using an automatically created node evaluation for each signal group over the entire simulation run.
- For optimizing, the signal group in which the vehicles have the highest delay is determined for each stage.
- The stage with the lowest maximum average delay is selected as the best stage.
- The stage with the highest maximum average delay is selected as the worst stage.
- A second of green time is deducted from the best stage.
- A second of green time is added to the worst stage.
- If a second can no longer be deducted from the best stage, the second best stage is used. If this can no longer be shortened, the next worst stage is always taken iteratively. If no other stage can be shortened, the optimization is terminated.
- A signal program is considered to be better than another if one of the following criteria is met:
  - If the flow formed by the total number of vehicles driven through the node during the simulation run has increased significantly by at least 25 vehicles or by 10% if this is less.
  - If the flow has not significantly decreased by 25 vehicles or by 10% and the average delay across all vehicles has decreased.
- If a signal program is better than the best rated, it replaces this as the best. The optimization is continued with the next step.
- The optimization is terminated if one of the following criteria is met:
  - Once the signal program does not improve within 10 simulation runs.
  - Once the flow decreases by more than 25% compared to the best signal program.
  - Once the average delay increases by more than 25%.

Requirements for the green time optimization of stage-based fixed time controllers

Ensure that the following requirements are met:

- The following must be defined in Vissig:
  - Signal groups (see "Defining signal groups in the SC editor" on page 639)
  - Intergreen matrix (see "Defining an intergreen matrix" on page 642)
  - Stages (see "Defining and editing stages" on page 645)
  - Stage based signal program (see "Defining signal programs" on page 653)
- The following must be defined in Vissim:
  - Signal heads (see "Defining signal heads" on page 579)
  - a surrounding node that contains the junction with the SC for node evaluation (see "Defining nodes" on page 708), (see "Evaluating nodes" on page 1057)
  - Adjacent nodes for legs of junction
- The interstages must include the minimum green times and relevant intergreens because the length of a stage can be reduced to zero by optimizing. The signal program must be consistent even if all stages have zero duration.
- The stage-based signal program has a user-defined cycle time and stages of any length. You can use the stage length that was proposed in Vissig when generating the signal program from interstages, for example, an equal distribution across all. You can use the proposed stage length because the original stage lengths are changed by optimizing.
- The demand and the path selection must be defined in the Vissim network. The following must thus be defined:
  - Vehicle inputs and routing decisions (see "Defining vehicle inputs" on page 456), (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459)
  - or for dynamic assignment, parking lots or parking lots and a trip chain file (see "Defining parking lots for dynamic assignment" on page 700), (see "Modeling traffic demand with origin-destination matrices" on page 721), (see "Modeling traffic demand with trip chain files" on page 730)
  - a path file (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771), (see "Path search and path selection" on page 738)
- The route choice does not necessarily have to be defined by static node flow routes. You can also use the dynamic assignment or static routes across multiple nodes because it is only required that the vehicles drive through the nodes of the SC.
- Other signal controls are not considered.

Starting green time optimization for an SC

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

2. Right-click the entry of your choice.
3. From the shortcut menu, choose **Optimize Signal Control**.

Starting green time optimization for all SCs

- From the **Signal Control** menu, select **Optimize All Fixed Time Signal Controllers**.

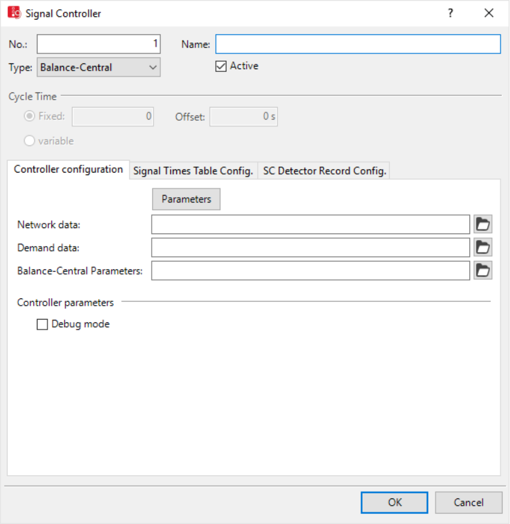
6.14.4.6 Using Balance-Central signal controllers

Note: You need a license for the signal controller.

The **Balance-Central** signal controller allows you to simulate adaptive light signal control using real time traffic data and thus also enables you to simulate adaptive network control.

An example and information on **Balance-Central** in English can be found in the directory *..\Examples\Examples Training\Signal Control\UTC - Workflow PTV Balance PTV Epics*.

1. When you define an SC, in the **Type** list, click **Balance-Central**.



2. Open the **Controller configuration** tab.
3. Make the desired changes.

Element	Description
Parameters button	Showing Balance Central-parameters. You can change the parameters and save them to the *Balance.ini* file. After making changes, you can save the file to the default directory *..\Users\Public\Public Documents\PTV Vision\PTV Vissim* or another directory of your choice.
Network data	**.anm* file with abstracted network model data

Element	Description
Demand data	**.anmroutes* file with volumes and routes
Balance Central parameters	Specify path to an **.ini* file that contains Balance-Central parameters. Click the **Parameters** button to display the parameters in the file.

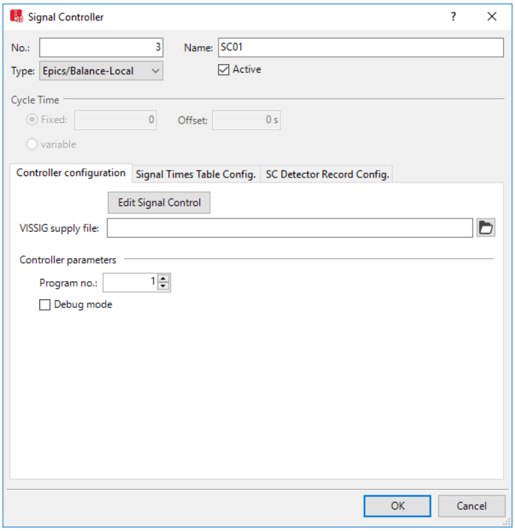
6.14.4.7 Using Epics/Balance-Local signal controllers

Note: You need a license for the signal controller.

The Epics signal control allows you to simulate adaptive light signal control for an individual node using real time traffic data.

Examples and information on **Balance-Central** in English can be found in the directory *..\Examples\Examples Training\Signal Control\UTC - Workflow PTV Balance PTV Epic* and *..\Tram Priority.PTV Epics*.

1. When you define an SC, in the **Type** list, click **Epics/Balance-Local**.



2. Open the **Controller configuration** tab.
3. Make the desired changes:

Element	Description
Program no.	**ProgNo**: The signal program which is to be simulated. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.
Debug mode	**Debug mode enabled** (**Debug**): ☑ If this option is selected, the signal flow can be followed during simulation in debug mode.

6.14.4.8 Add-on module Econolite ASC/3

Note: You must have a license for the add-on module.

Econolite ASC/3 is a control procedure used in North America. For further information, please visit:

http://www.econolite.com/index.php/products/controllers/software/#software

Some control procedures expect the supply file to be saved in the same directory that contains the *.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

You are provided with the *asc3gui.dll* files and the *asc3.dll* program file for the 64-bit Vissim version purchased.

Note: Current versions of ASC/3 use the *asc3gui.dll* file. In old projects, replace the old file name *asc3_gui.dll* by the new file name *asc3gui.dll*.

The control procedure additionally uses the following files:

When you add an SC of the type **Econolite ASC/3**, the respective files are automatically saved to the *Exe* directory of your Vissiminstallation:

- Data file **.db*
- Program file *asc3.dll*
- Dialog DLL file *asc3gui.dll*
- File with value type tables *ASC3.wtt*

1. If desired, click the **Parameters** button.

   The Econolite database editor opens.

2. Make the desired changes.
3. If desired, click the **Edit Mapping** button.

   The ASC3 I/O mapper opens.

4. Make the desired changes.

6.14.4.9 Add-on module external signal control SC

Note: You must have a license for the add-on module.

You can simulate with the add-on module Signal Controllers. This is available as a separate executable program (**.exe*) or program library (**.dll*). These can either be standard procedures supplied by PTV GROUP or other providers, or procedures that users have developed themselves (using the API add-on).

The files must be in the programming language C or C++.

Note: The DLL files must be compiled appropriately for the Vissim 64-bit version in use.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

If you have a license which includes the SC type **External**, the API source code modules and the documentation can be found in the *API* folder of your Vissim installation.

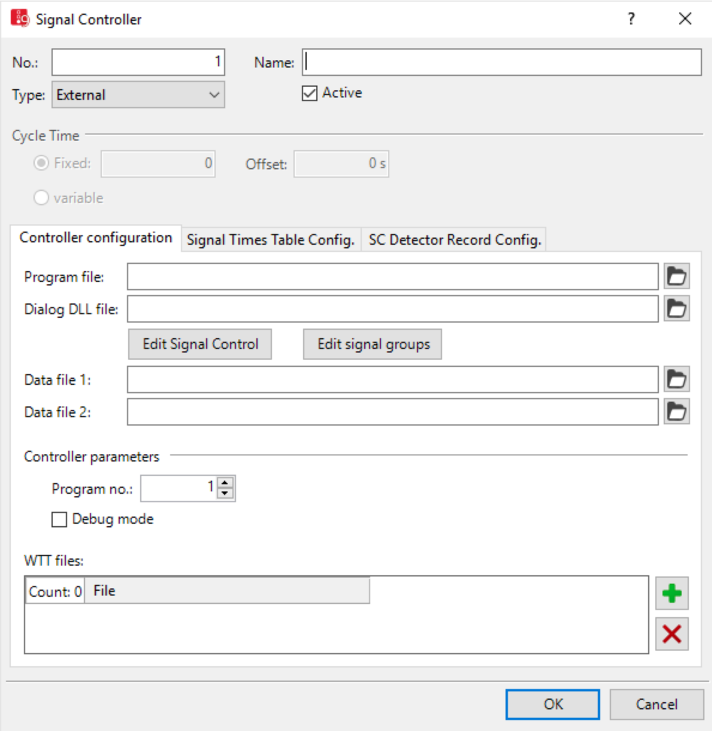
1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*



4. In the **Type** field, select > **External**.
5. Make the desired changes:

Element	Description
Program file	**ProgFile**: User-defined DLL file with control logic
Dialog DLL file	**GUIFile**: User-defined DLL file of supply interface
Data file 1	**SupplyFile1**: Data for the control logic Note: The supply files are only necessary if you do not want to include the data of the control logic in the program text of your user-defined DLL file.
Data file 2	**SupplyFile2**: Data for the control logic
Controller parameters	**ProgNo** (**Program number**):The signal program you want to simulate. The signal program number can be specified for an SC of type **External**. This is also possible during the simulation run.

Element	Description
Debug mode	**Debug mode enabled** (**Debug**): ☑ If this option is selected, the signal flow can be followed during simulation.
WTT files	**WTTFile**: Value type tables: The tables contain the data types of the control logic and the type of display in the SC detector record or in the **Signal times table** window. By default, a *vissig.wtt* file is saved to the *Exe* folder of your Vissim installation. A WTT file must not necessarily be specified.

File names with a path for the program file, the dialog DLL and the **.wtt* files can be transferred to external signal controllers. The paths are saved as relative paths that contain the current data directory, in which the network file **.inpx* is stored, or the program directory, in which the file *VISSIM<version number>.exe* is stored. Thus these can still work even after the data directory is moved or copied to another computer.

Note: The signal program number for an SC of type **External** can also be specified during the simulation run. The signal program is then changed during the next switch point.

Documentation in English

- *..\<VISSIM Version>\API\SignalControl_DLLs\SC_DLL\SC_DLL_Interface.doc*

Examples

- *..\<VISSIM Version>\API\SignalControl_DLLs\Examples*

6.14.4.10 Add-on module Fourth Dimension

Note: You must have a license for the add-on module.

This control procedure is deployed in North America.

Depending on the Vissim edition purchased, you are provided with the dialog file *D4gui.dll* and the program file *D4.dll* files for a 64-bit version.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

1. When you define an SC, in the **Type** list, click **Fourth Dimension D4**.
2. Select the desired data file.
3. Make the desired changes:

Element	Description
Data file	**ProgDat**: Supply file of control logic
Program no.	**ProgNo**: The signal program which is to be simulated. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.

The control procedure also uses a WTT-file, e.g. *D4.wtt*.

6.14.4.11 Add-on module LISA+ OMTC

Note: You must have a license for the add-on module.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

LISA+ OMTC is a control procedure developed by SCHLOTHAUER & WAUER, a traffic engineering company in Berlin, Germany.

1. Ensure that you have a dialog.dll file and a **.wtt* file, e.g. *VISSIM2LisaDlg.dll* and *Lisa_Vissim.wtt*.
2. When you define an SC, in the **Type** list, click **LISA+ OMTC**.
3. In the **Data file** box, select the file of your choice.
4. To open LISA+ configuration, click the **LISA+ Configuration** button.
5. To edit the settings for the signal groups, click the **Edit signal groups** button.
6. Make the desired changes.

6.14.4.12 Add-on module McCain 2033

Note: You must have a license for the add-on module.

McCain 2033 is a control procedure deployed in North America.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

Note: The GUI file *McCaingui.dll* and the program file *McCain2033.dll* are only provided for 32-bit versions of Vissim. From version 11, Vissim is available as a 64-bit edition only.

- When you define an SC, in the **Type** list, click **McCain 2033**.

Element	Description
Supply file 1	**SupplyFile1**: Supply file of control logic. The files must be saved in the same folder as the Vissim data.
Controller parameters	**ProgNo** (**Program number**):The signal program you want to simulate. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.

The control procedure also uses the WTT-file *mccain2033.wtt*.

6.14.4.13 Using the Ring Barrier Controller RBC add-on module

Note: You must have a license for the add-on module.

The Ring Barrier Controller control procedure is used in North America and has superseded NEMA.

Note: If a network file with NEMA type controls is read in, these controls are automatically converted to controls of the RBC type.

In Vissim, the external SC type **Ring Barrier Controller** has a graphic user interface and additional functions.

In Vissim the settings are saved as an external file with the file extension **.rbc*.

The English version of the RBC manual, *Manual_RBC.pdf*, can be found in the folder *Doc\ENG* of the Vissim installation.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

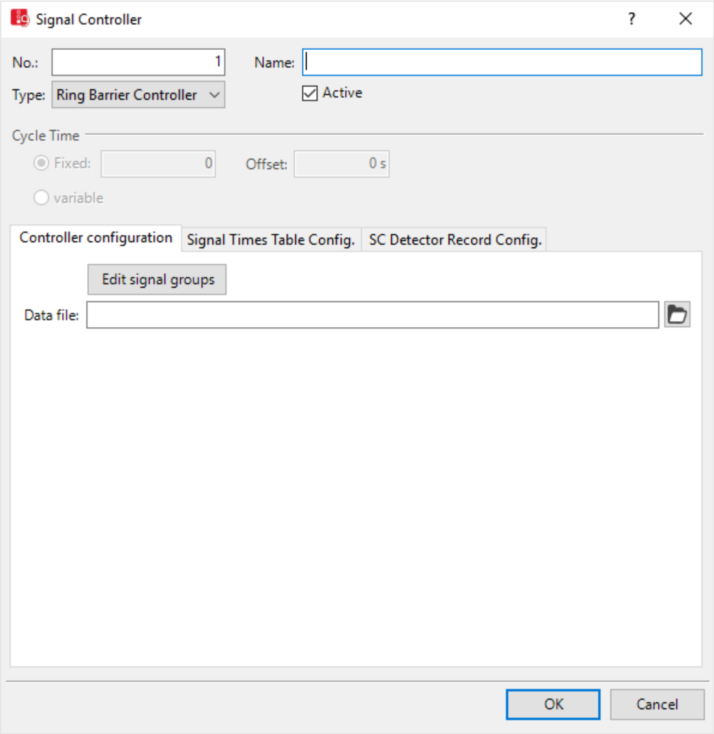
   *The **Signal Controllers** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*



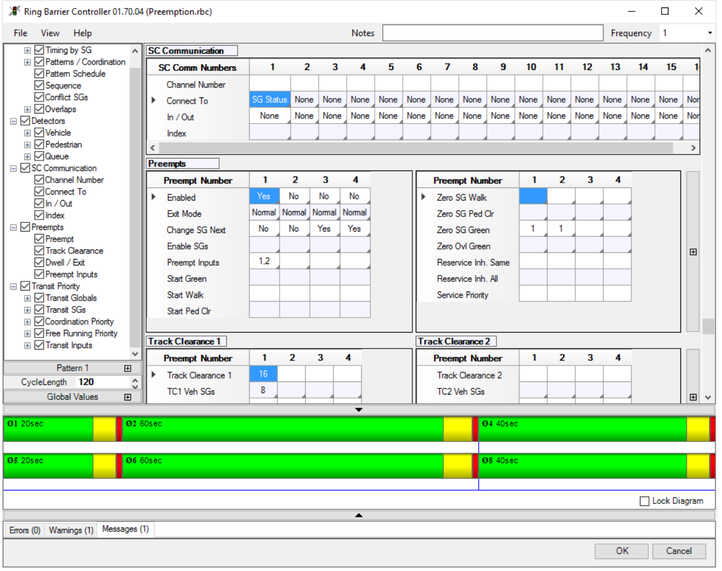
4. Select field **Type** > **Ring Barrier Controller**.
5. In the **Data file** box, select the file of your choice.
6. Make the desired changes:

The control procedure additionally uses the following files:

Element	Description
Program file	**ProgFile**: *rbc_controller.dll* file: program file of the control logic with which the node is to be controlled. Your license is provided as a 64-bit version. By default, the file is saved in the *exe* folder of your Vissim installation.
Dialog DLL file	**GUIFile**: *rbc.dll* file: is provided as a version according to the license purchased. By default, the file is saved in the *exe* folder of your Vissim installation.
WTT files	**WTTFiles**: Value type tables: These contain the data types of the control logic which are to be shown in the Signal Control Detector Record or in the **Signal Times Table** window, as well as the display type. If the control consists of several TL modules, you must specify the associated **.wtt* file for each module. By default, the file is saved in the *exe* folder of your Vis-sim installation.

7. Click **Edit signal groups**.

   *The **Ring Barrier Controller** window opens.*



8. Proceed as described in the manual *Manual RBC.pdf*, in the *DOC\ENG* folder of your Vissim installation.

6.14.4.14 Add-on module SCATS

Note: You must have a license for the add-on module.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

- When you define an SC, in the **Type** list, click **SCATSI**.

The control procedure additionally uses the following files:

- *scats.dll*
- *scats_gui.dll*

- the programs **WinTraff** and **ScatSim** developed by the Roads and Traffic Authority of New South Wales, Australia,
- the program **SimHub** by Roads and Traffic Authority of New South Wales, Australia, if required.

You are provided with the DLL files for the 64-bit Vissim version licensed.

6.14.4.15 Add-on module SCOOT

Note: You must have a license for the add-on module.

The control procedure additionally uses the following files:

- *SCOOT_LOGIC.dll*
- *SCOOT_GUI.dll*
- The Siemens program **PCScoot** www.scoot-utc.com

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

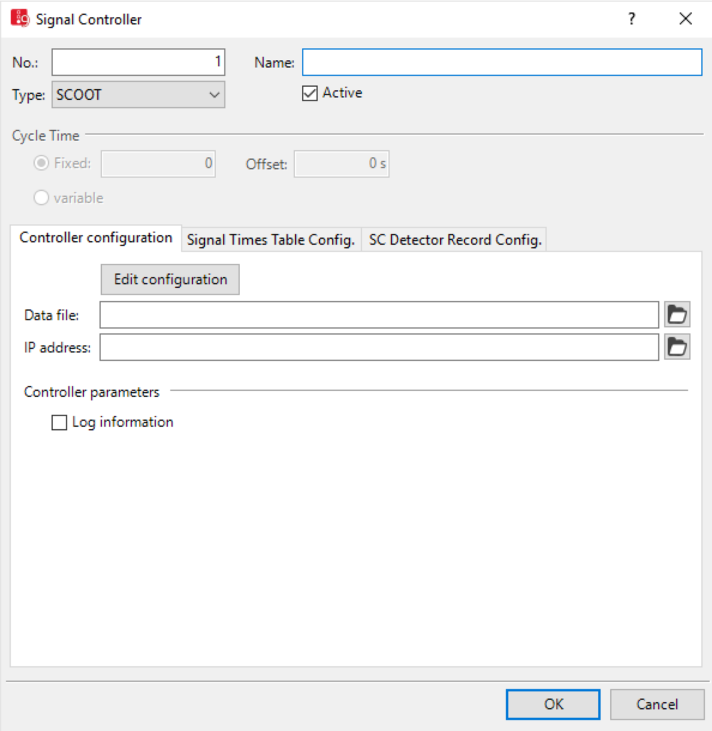
   *The **Signal Controllers** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*



4. In the **Type** field, select > **SCOOT**.
5. Make the desired changes:

Element	Description
Data file	**SupplyFile**: supply file
IP address	Data file of fixed time control **.sig*
Controller parameters	**Log information**: Select this option to write data required for UTC communication to the LOG file: - Debugging information - Log information

The control procedure additionally uses the following files:

Element	Description
Program file	**ProgDat**: File *SCOOT_Logic.dll*. Program file of control logic that shall be used to control the intersection. Your license is provided as a 64-bit version.
Dialog DLL file	**GuiFile**: File *SCOOT_gui.dll*. Your license is provided as a 64-bit version.

Element	Description
WTT files	**WTTDat**: Value type-tables. They contain the data types of the control logic that shall be displayed in the **SC Detector Record** or in the **Signal Times Table** window, as well as the display type.

6.14.4.16 Using add-on module TRENDS

Note: You must have a license for the add-on module.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

- When you define an SC, in the **Type** list, click **TRENDS**.

You can simulate signal controls that internally use the TRENDS core of the company GEVAS, Munich.

The control procedure additionally uses the following files:

- **Program file**: Control program **.EXE*, default *TREND429.exe*
- **STG file**: ASCII supply file
- **VXB file**: Binary supply file
- **WTT file**: Value type table, by default *trend429.WTT*

6.14.4.17 Add-on module Siemens VA (TL / Siemens VS-PLUS)

Note: You must have a license for the add-on module.

At intervals of one second, Vissim calls up a separate control program for each signal controller. To do this, select the supply files *PW1* for the control of each signal controller.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

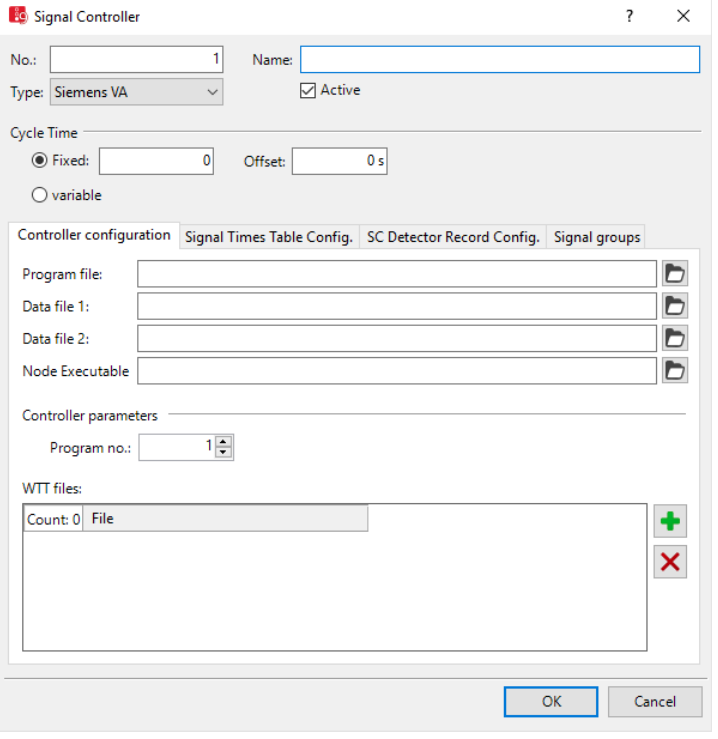
   The ***Signal Controllers*** *list opens.*

By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*



4. In the **Type** box, select > **Siemens VA**.
5. Open the **Controller configuration** tab.
6. Make the desired changes:

Element	Description
Program file	**ProgFile** **.dll* file: Program file of the control logic for more modern Siemens control units that you want to use to control the intersection. In addition, in the **Node Executable** box. choose an **.exe* file. This is the file **.exe*: the program file of the control logic for older Siemens control units, with which the nodes are to be controlled.
Supply file 1, **Data file 2**	**SupplyFile1**, **SupplyFile2**: Supply files of the control logic The files must be saved in the same folder as the Vissim data. ▶ **Import file 1**: Siemens VS-PLUS requires the logic in PW1 format. ▶ **Import file 2**: This is specific to the particular device. For example, it contains the smoothing parameters for the detectors. In general the file which is supplied with the Siemens VS-PLUS can be used for this, e.g. *VSP0400G.pw1* for version 4.00.
Node Executable	This is only necessary if a DLL file and no EXE file are selected in the **Program file** field. This is an EXE file with a control logic which is specific to the node. The field is ignored if an EXE file is chosen as the program file.
Controller parameters	**ProgNo** (**Program number**):The signal program you want to simulate. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.
WTT files	**WTTFiles**: Value type tables: These contain the data types of the control logic which are to be shown in the Signal Control Detector Record or in the **Signal Times Table** window, as well as the display type. If the control consists of several TL modules, you must specify the associated **.wtt* file for each module. : Select WTT file : Delete WTT file from list

6.14.4.18 Add-on module Traffic-dependent VAP Programming

Note: You must have a license for the add-on module.

With the add-on module VAP you can model programmable control logics in the description language VAP. These must be created in the display type A or B according to RiLSA 1992 or 2010.

In contrast with VS-PLUS or TrafficLanguage (TL, Siemens) VAP is not a control procedure which can run in an identical form on devices from different signal manufacturers. However, you can emulate almost all control procedures with VAP.

The **.dll* files for VAP are provided with your Vissim edition. Depending on the license purchased, the files are provided in a a 64-bit version.

In the *Doc* folder of your Vissim installation, you can find a description of the VAP description language in the file *VAP_<Version>_<Language>.pdf*.

1. From the **Signal Control** menu, choose > **Signal Controllers**.

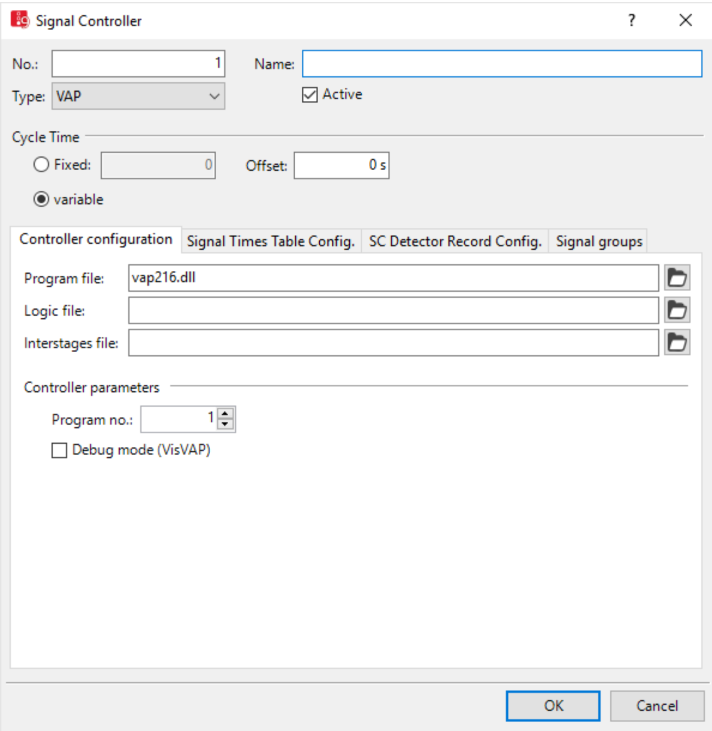
   *The **Signal Controllers** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*



4. In the **Type** field, select > **VAP**.

5. Make the desired changes:

Element	Description
Program file	**ProgFile**:This is the **.dll* or **.exe* file of the control logic with which the nodes are to be controlled. By default *vap216.dll*.
Interstages file	Select the file **.pua* with interstages
Logic file	The **.vap* file with the SC program logic for a VAP control. Make sure that the end symbol is in the last line of the VAP code. Otherwise, the VAP code may be generated incorrectly or not at all.
Program no.	**ProgNo**: The signal program which is to be simulated. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.
Debug mode (VisVAP)	**Debug mode enabled** (**Debug**): ☑ If this option is selected, the signal flow can be followed during simulation in debug mode. VisVAP must be open. The **.vv* file must be loaded.

Graphic display and editing of flow diagrams with VisVAP

VisVAP is a convenient platform for creating signal programs without knowledge of programming. VisVAP is available in addition to VAP. VisVAP provides the following functions:

- Graphic display and editing of flow diagrams
- Conversion of VAP flow diagrams into VAP source text
- Tracking of the sequence of the signal flow during the simulation.
- Specifying relative flows for **pedestrian routes (static)** and **pedestrian routes (partial route)**

You can find information about VisVAP in the file *VisVAP_<Version>_<Language>.pdf* in the *Doc* folder of your Vissim installation.

6.14.4.19 Add-on module VS-Plus

Note: You must have a license for the add-on module.

At intervals of one second, Vissim calls up a separate VS-Plus program for each signal controller. In the program name, the number behind *VSP* states the version number.

For each SC in Vissim, you must select the supply file **.VCE* or **.PW1* for the VS-PLUS control. You can create these files with the planning and supply interface visual VS-Plus.

Some control procedures expect the supply file to be saved in the same directory that contains the **.inpx* file. If a control file is saved to a directory that does not meet the requirements of the control procedure, this can cause the program to crash. The supply files of the control procedures Fixed time/Vissig, VAP and RBC can be saved to any directory Vissim can access. Vissim does not need the supply files to be saved to a specific directory. Should you have any questions regarding supply files, please contact the company that developed the control procedure.

Tips: You can obtain information about the program versions of VS-Plus from the manufacturers, VS-PLUS AG and Siemens AG.

Information for the use of VS-PLUS (see "Add-on module Siemens VA (TL / Siemens VS-PLUS)" on page 624).

---

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

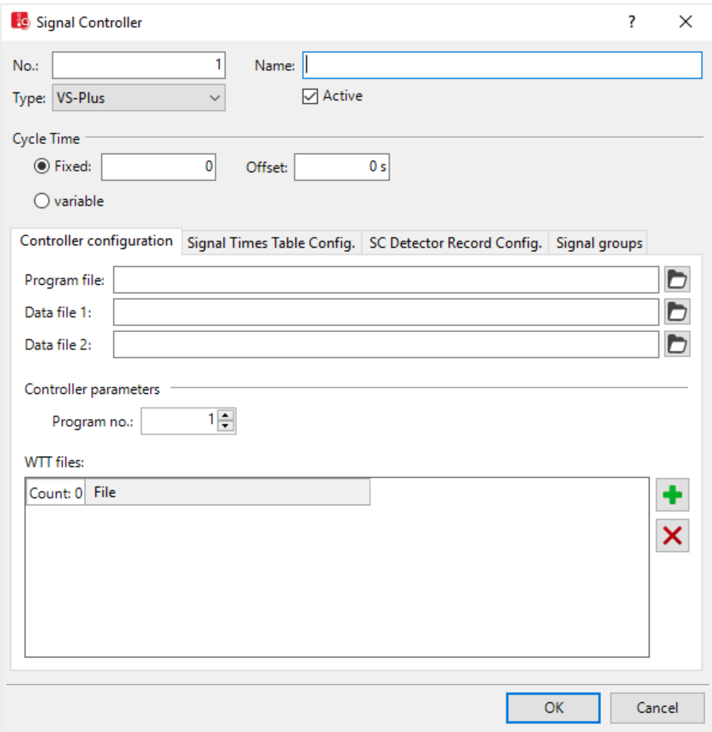
   By default, you can edit the list (see "Using lists" on page 93).

2. Right-click the entry of your choice.

3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*

4. In the **Type** field, select > **VS-Plus**.
5. Open the **Controller configuration** tab.



6. Make the desired changes:

Element	Description
Program file	**ProgFile**: This is the VS-PLUS version with which the node is to be controlled. Always use the same VS-PLUS version to create the supply file with Visual VS-PLUS in the simulation and for the application on the actual control device.
Data file 1 **Data file 2**	**SupplyFile1**, **SupplyFile2**: Interface file and parameter file. The files must be saved in the same folder as the Vissim data. If the VS-PLUS supply is carried out with Visual VS-PLUS, only one VS-PLUS input file **.vce* is generated. Enter their file names in the field **Import file 1**. ➤ **Import file 1**: This is an interface file, e.g. GEO_I.VCE if this was generated by *IVA* or by the VS-PLUS input file **.vce*. ➤ **Import file 2**: This is a parameter file, if this was generated by *IVA*.
Controller parameters	**ProgNo** (**Program number**):The signal program you want to simulate. A change of program is possible if the simulation run is in single-step mode. If the new signal program number is confirmed with **OK**, the changeover is triggered in the next simulation second.
WTT files	**WTTFiles**: Value type tables: These contain the data types of the control logic which are to be shown in the Signal Control Detector Record or in the **Signal Times Table** window, as well as the display type. If the control consists of several modules, you must specify the associated **.wtt* file for each module. : Select WTT file : Delete WTT file from list

6.14.4.20 Defining signal groups for Siemens VA, TRENDS, VAP, VS-Plus

In the **Signal Controller** window or the **Signal groups** list, you can define signal groups for the signal control procedures Siemens VA, TRENDS, VAP and VS Plus.

Defining signal groups in the Signal Controller window

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens. The SCs defined are displayed (see "Defining SC and signal control procedures" on page 602).*

2. Double-click the SC of your choice (Siemens VA, TRENDS, VAP or VS-Plus).

   *The **Signal Controller** window opens.*

3. Ensure that the **Signal Groups** tab is shown in the foreground.

   A list of the attributes of signal groups is displayed.

4. Right-click in the list.

5. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

6. Edit the desired entries.

7. Confirm with **OK**.

Defining signal groups in the Signal Groups list

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens. The SCs defined are displayed (see "Defining SC and signal control procedures" on page 602).*

2. Select the SC of your choice (Siemens VA, TRENDS, VAP or VS-Plus).
3. On the list toolbar, in the **Relations** list, click **Signal groups**.
4. On the toolbar of the **Signal Groups** list on the right, click the **New ...** button .

   A new row with default data is inserted.

5. Edit the desired entries.

6.14.5 Opening and using the SC Editor

In the SC editor, you configure new SCs of the type **Fixed time** and **Epics/Balance Local** or edit existing SCs of this type. To do so, you can make the following settings:

- Defining SC with frequency (see "Defining SC with frequency" on page 638)
- Defining signal groups of SC (see "Defining signal groups in the SC editor" on page 639)
- Editing default signal sequences and signal states of signal groups (see "Editing signal groups" on page 640)
- Defining intergreens (see "Defining an intergreen matrix" on page 642)
- Editing intergreens (see "Editing intergreen matrices" on page 643)
- Defining and editing stages (see "Defining and editing stages" on page 645)
- Assigning stages to signal groups and selecting a default intergreen matrix (see "Editing stage assignment" on page 647)

- Creating stage sequences, interstages, and signal group based signal programs (see "Editing stage sequence" on page 649). Using the Vissig add-on module, you may also define stage-based signal programs.
- Defining signal programs (see "Defining signal programs" on page 653)
- Editing signal programs (see "Editing signal programs" on page 654)
- Editing the interstages created (see "Editing interstages" on page 661)
- Defining daily signal program lists (see "Defining and editing daily signal program lists" on page 664)

If you have added an SC, you must save the network file **.inpx* to open the SC editor.

Note: When in Vissim you edit data that is saved to external files, for example **.sig* files, Vissim does not provide the **Undo** function for any previously selected commands.

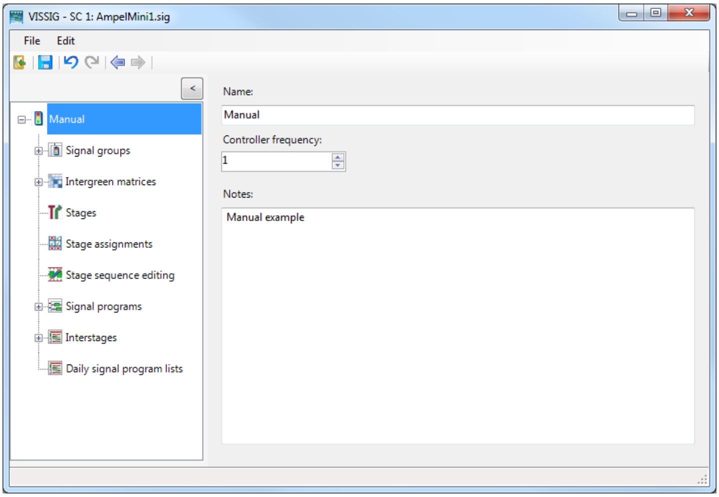
6.14.5.1 Opening the SC Editor

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens. The SCs defined are displayed (see "Defining SC and signal control procedures" on page 602).*

2. Right-click the entry of your choice.
3. From the context menu, choose **Edit Signal Control**.

   *The **SC Editor** opens.*



6.14.5.2 User interface of the SC Editor

The user interface of the SC Editor is divided into the following areas:

Element	Description
Title bar	Program name, number of the selected signal control and currently loaded signal control file **.sig*.
Menu bar	It is operated using the mouse or key combinations. Menu entries contain graphic notes on submenus or windows: ▶ The symbol indicates a subordinate submenu. " ..." Three points stand for a subordinate window.
Toolbar	Control elements for control and editing
Scroll bars	Scroll window content horizontally or vertically
Navigator in section on the left	List of available parameter pages for the currently selected SC. With the Vissig add-on module, you can also edit stages and daily signal program lists. - Button **<** above the Navigator: Closes the pane with the view of the tree structure. A vertical label opens which specifies the parameter page on which the cursor was when it was closed. - Button **>**: Opens the Navigator again on the specified parameter page.
Parameters in section on the right	Parameters for defining and editing the SC (see "Defining SC with frequency" on page 638)

6.14.5.3 Menus in the SC Editor

File menu

Element	Description	Hotkeys
Open	**.sig* configuration file	
Export	Export Excel Workbook or **.pua* file with interstages in text format for VAP	
Save	Save Vissig configuration file **.sig* with same path and name	CTRL+S
Save as…	Save Vissig configuration file **.sig* under a new path or name, may contain Unicode characters.	
Check	Check for inconsistent planning (see "Detecting inconsistent planning" on page 667)	
Exit	Close the SC Editor	ALT+X

Edit menu

Element	Description	Hotkeys
Undo	Cancels the last action. Each executed step can be undone.	CTRL+Z
Redo	Redoes the last undone action. Each step can be restored.	CTRL+Y
Options…	General Settings (Common, Optimizations, View and Export) (see "Making global settings in the SC Editor" on page 635)	

6.14.5.4 Toolbar in the SC Editor

Depending on the entry in the Navigator and the selected editing view, you can execute functions with the following icons:

Icon	Description	Hotkeys
	Back to the **Signal Controller** window	
	Save	CTRL+S
	Undo	CTRL+Z
	Redo	CTRL+Y
	Back in View	
	Forward in View	
	New	
	Duplicate	
	Edit	
	Delete	

6.14.5.5 Signal states

State	Signal state
	Red
	Red-amber
	Green
	Amber
	Flashing Green
	Flashing Amber
	Dark

6.14.5.6 Signal state sequences

The signal state sequence defines the following for each signal state:

- Permitted state or blocked state
- Fixed or variable duration

- Time horizon
- Minimum duration of the signal state in the default sequence. The minimum duration is predefined and can be changed for each signal group.

State	Signal state sequence	Green	Fixed Duration	Minimum Duration
	Permanent Red			
	Permanent Green	x		
	Red-Red/Amber-Green-Amber			1
			1	
		x		5
			3	
	Red-Green			1
		x		5
	Red-Red/Amber-Green-Flashing Green-Amber			1
			1	
		x		5
		x	4	
			3	
	Red-Green-Flashing Green			1
		x		5
		x	4	
	Red-Green-Amber			1
		x		5
			3	
	Off (Flashing Amber)			
	Off (Off)			

6.14.5.7 Making global settings in the SC Editor

Note: The following tabs are only available with the add-on module Vissig.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).

2. In the menu, choose **Edit** > **Options**.

   *The **Options** window opens.*

3. Make the desired changes:

Common tab

- Select the desired language.

The selected language is used as the default language.

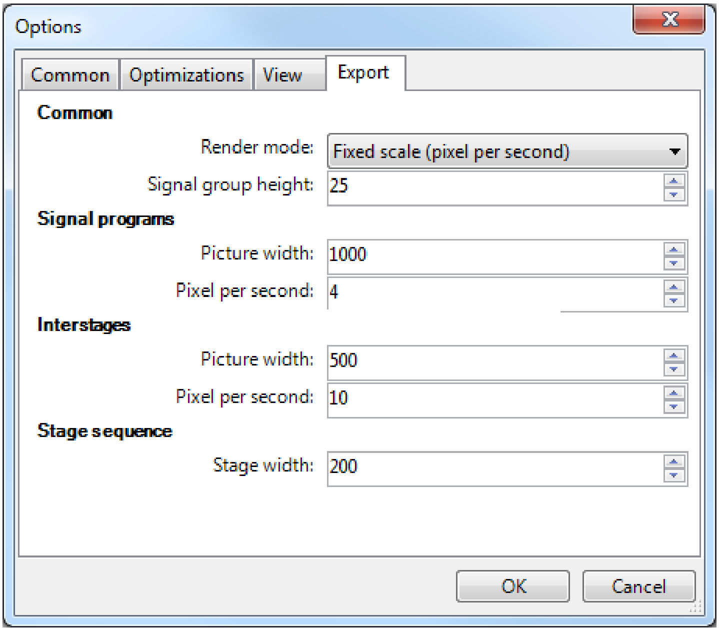
Optimizations tab

Element	Description
Interstages	Interstages: - **Add minimum times in interstage**: ☑ If this option is selected, the target signal states of the individual signal groups have at least the minimum period specified in the signal groups. - **(Setting can be changed in the SC settings)**: Alternatively, in the SC settings with the option **Check minimum times in signal programs**, select or deactivate the option **Add minimum times in interstage**(see "Defining SC with frequency" on page 638). Optimizes green time duration: - **Use optimal length in front**: ☑ If the option is selected, the green is ended at the start of the interstage when changing from green to red. Green starts as early as possible. ☐ If the option is not selected, the green is ended at the start of the interstage when changing from green to red. - **Use optimal length in back**: ☑ If the option is selected, the green does not only start at the end of the interstage or at the end of the interstage minus the minimum period when changing from green to red. Green ends as late as possible. ☐ If the option is not selected, the green only starts at the end of the interstage or at the end of the interstage minus the minimum period when changing from green to red.

View tab

Element	Description
Stages	Stages: ☑ **Show signal group ids**: Signal group numbers are shown in the stage diagram. ☑ **Use compact view**: A schematic rather than a topographic display is shown in the stage diagram.
Signal programs and interstages	**Appearance**: You can select from the following display options: Classic 3d tubes 3d boxes ☑ **Resize automatically**: By changing the window size, the row height automatically adjusts to the window height. ☑ **Show full interstage name in the stage based signal programs**: The names are displayed instead of the numbers of the interstages.

Export tab

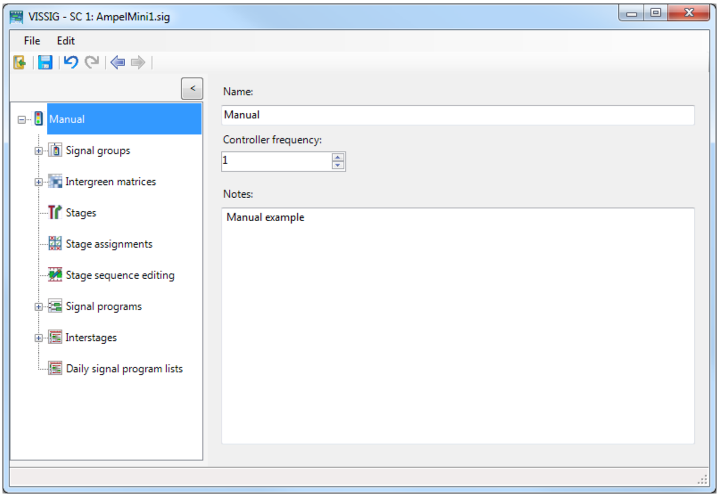


Element	Description
Common	**Render mode** • **Fixed width**: Width of export graphic based on **Picture width** • **Fixed scale (pixels per second)** in pixel per second: Width of export graphic based on **Pixel per second** entered below. **Signal group height**: Signal group height in pixels globally for all graphic exports
Signal programs	**Picture width**: Width of the export graphic in pixels. The given value is taken into account if the **Fixed width** option is selected as the **Render mode**. **Pixel per second**: Number of pixels that represent a second in the export graphic. The given value is taken into account if the option **Fixed scale (pixel per second)** is selected as the **Render mode**.
Interstages	**Picture width**: Width of the export graphic in pixels. The given value is taken into account if the **Fixed width** option is selected as the **Render mode**. **Pixel per second**: Specify the number of pixels that represent a second in the export graphic. The given value is taken into account if the option **Fixed scale (pixel per second)** is selected as the **Render mode**.
Stage sequence	**Stage width**: Width of the stage in pixels for the export. The given value is taken into account if the **Fixed width** option is selected as the **Render mode**.

6.14.5.8 Defining SC with frequency

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Click on the top entry in the Navigator.

   The base data of the SC opens.



You can define a new signal control or change the settings for the selected SC.

Element	Description
Name	Label for the top entry in the navigator: no name: **My signal control** is displayed by default in the Navigator and the SC number is added to it. The SC number is generated automatically.
Controller frequency	Number of calls of the SC per simulation second, value range 1-10. The SC frequency controls the frequency of light signal controls during the simulation. A warning is displayed if the SC frequency is incompatible with the simulation parameter **Simulation resolution**. The least common multiple of the frequencies of all SCs must be a factor of the simulation resolution.
Check minimum times in interstage programs	If this option is selected, the target signal states of the individual signal groups have at least the minimum period specified in the signal groups.
Notes	optional entry of text

3. Click the **Save** button.

6.14.5.9 Defining signal groups in the SC editor

You can open the SC editor to define signal groups for the following SC types:

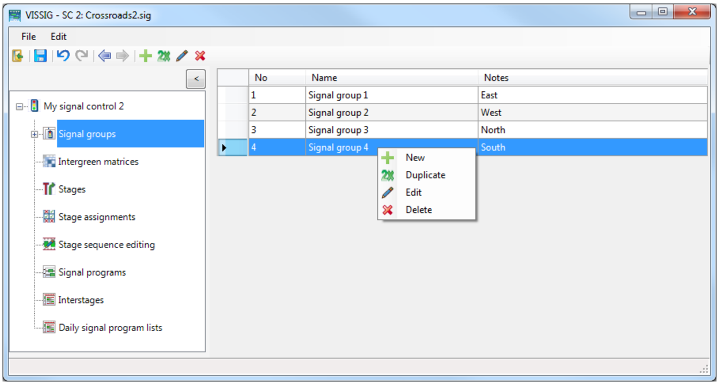
- Fixed time
- Epics/Balance Local

For signal controllers of the type Siemens VA, TRENDS, VAP and VS Plus, you do not define signal groups in the SC editor (see "Defining signal groups for Siemens VA, TRENDS, VAP, VS-Plus" on page 630).

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. In the Navigator, click **Signal groups**.

   The signal groups list opens.

   If no signal group has been defined yet, only the column titles are displayed.



3. Right-click in the list.

   The context menu opens.

4. Click on the desired entry.

Element	Description
Add	Add new signal group with the first port number available. By default, the signal state sequence **Red-Red/Amber-Green-Amber** is assigned. To keep data input to a minimum, create a signal group for each signal group type you need. Then supply their standard signal sequence and duplicate the template created as often as required.
Duplicate	Copy signal group and insert with a new number
Edit	Change data of the selected signal group
Delete	Delete selected signal group

   By default, you can edit the list (see "Using lists" on page 93).

5. Click the **Save** button.

6. Edit the signal group (see "Editing signal groups" on page 640).

6.14.5.10 Editing signal groups

You can change the signal state sequence and its minimum durations. For signal states that in the signal sequence possess a variable duration, the value entered is interpreted as the minimum duration. For any other signal states, the value entered is interpreted as the duration. This, for instance, allows you to supply 2 seconds red-amber, 5 seconds amber for 70 km/h (speed limit in Germany) or 15 seconds minimum green for a heavily loaded straight flow.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).

2. In the Navigator, click **Signal groups**.

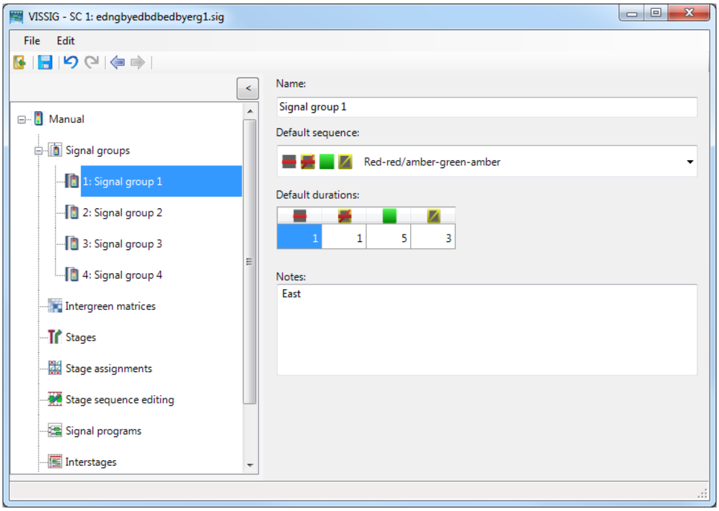
   The signal groups list opens.

3. Right-click the entry of your choice.

4. From the shortcut menu, choose **Edit**.

   In the window, the name, default signal sequence, signal duration, and a description are displayed.

   Tip: Alternatively, you can show the window via the following functions:
   - In the Navigator, click the signal group.
   - In the signal groups list, in the signal group row of your choice, double-click the **Notes** field.



5. Make the desired changes:

Element	Description
Name	Name of signal group, may contain Unicode characters.
Default sequence	With the exception of permanent signal sequences, you can choose any of the signal state sequences defined (see "Performing green time optimization of stage-based fixed time controllers" on page 609): ▸ **red** > **red/amber** > **green** > **amber** ▸ **red** > **green** ▸ **red** > **red/amber** > **green** > **flashing green** > **amber** ▸ **red** > **green** > **flashing green** ▸ **red** > **green** > **amber**

Element	Description
	Note: If in the SC editor, in the **Edit** menu, you choose **Options** > **Optimizations** and then click **Add minimum times in interstage**, the increase in the minimum green time or transition time might mean that individual interstages have to be recalculated. This is why before changes are accepted, Vissim checks whether the interstages need to be recalculated due to these changes. If required, a message is then displayed, asking whether you still want the changes to be carried out. Within this context, changes made in the meantime to optimization settings or stages might also require recalculations, which are not due to the actual change.
(Minimum) durations	Symbol for signal state sequence and minimum durations in seconds of the signal state
Notes	optional entry of text

6. Click the **Save** button.

6.14.5.11 Defining an intergreen matrix

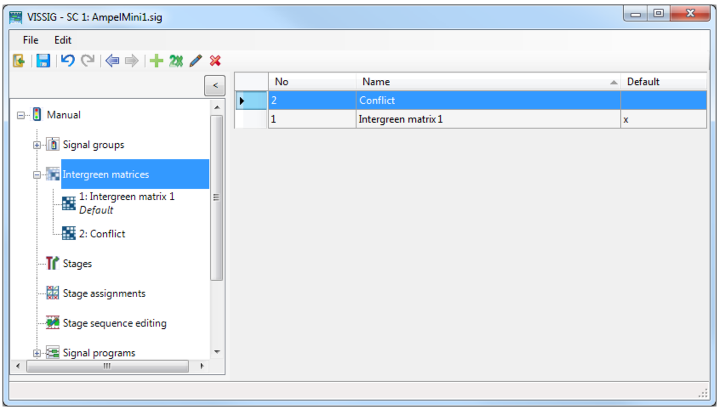
Intergreen is the time required to elapse between the green end of a clearing flow and the green start of an entering flow. By keeping the intergreen, you ensure that the clearing flow does not conflict with the entering flow.

Note: Conflicting flows and intergreens cannot be calculated. When intergreen data is entered, there are no checks, e.g. for the symmetry of intergreen matrices.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. In the Navigator, click **Intergreen matrices**.

   The intergreen matrices list opens.

   If no intergreen matrix has been defined yet, only the column titles are displayed.



4. Right-click in the list.

The context menu opens.

5. Click on the desired entry.

Element	Description
Add	Add new intergreen matrix
Duplicate	Copy intergreen matrix and insert with a new number
Edit	Change data of the selected intergreen matrix
Delete	Delete selected intergreen matrix

To sort a column of the list, click the column header.

6. Make the desired changes:

Element	Description
No	Number of the intergreen matrix
Name	Name of the intergreen matrix
Default	The default intergreen matrix selected under **Stage assignment** is marked with an x. Double-click into the **Default** column to open the respective editing view.

You can assign an intergreen matrix individually to each signal group-based signal program. In the Navigator, under Stage assignments, you can choose an intergreen matrix as default (see "Editing stage assignment" on page 647). The program then takes the intergreen matrix into account to create stages, stage transitions, and stage-based signal programs.

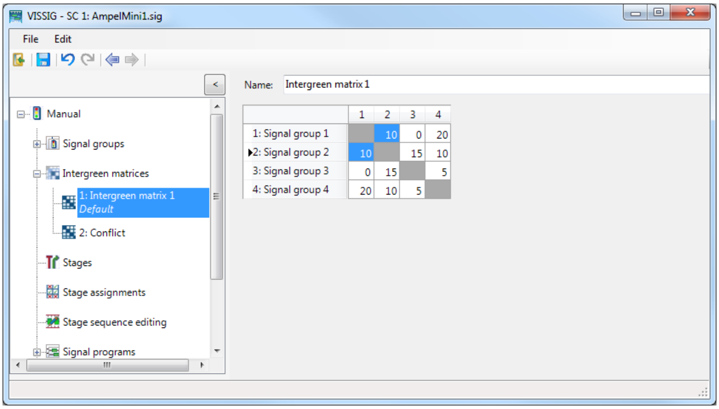
7. Click the **Save** button.
8. Edit the intergreen matrix (see "Editing intergreen matrices" on page 643).

6.14.5.12 Editing intergreen matrices

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Click **Intergreen matrices** in the Navigator.

   The intergreen matrices list opens.
3. Right-click the entry of your choice.
4. From the shortcut menu, choose **Edit**.

Tip: Alternatively, you can display the window by clicking the intergreen matrix in the Navigator.



5. Make the desired changes:

Element	Description
Name	Name of the intergreen matrix
White fields	Enter intergreen values
Gray fields	Editing not possible

When a cell is activated, the corresponding cell in the opposite triangle is also highlighted.

6.14.5.13 Importing intergreen values from Excel

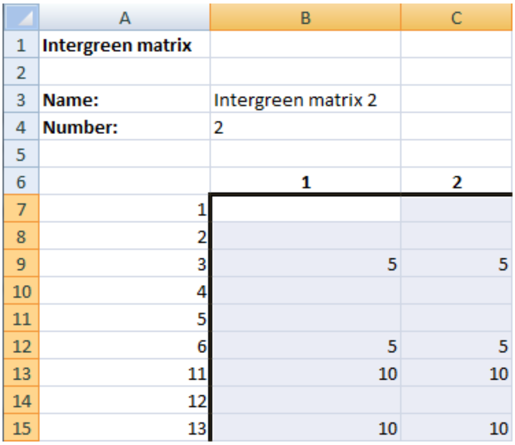
You can import intergreen values from Microsoft™ Excel™. If, for example, you have exported an Excel workbook, you can insert the data of the intergreen matrices into a new or existing matrix using Copy & Paste.

Inserting Excel data in a new empty matrix

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Click **Intergreen matrices** in the Navigator.

   The intergreen matrices list opens.
3. Right-click in the list.
4. From the shortcut menu, choose **Add**.
5. Right-click the new entry.
6. From the shortcut menu, choose **Edit**.

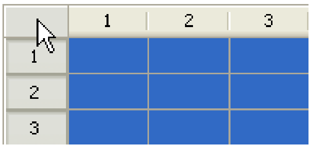
   *An empty matrix named **Intergreen matrix** is created.*
7. Change the name and number of the intergreen matrix if necessary.
8. Switch to your Excel file.
9. Open the worksheet of the intergreen matrix from which you want to extract the data.



	A	B	C
1	**Intergreen matrix**		
2			
3	**Name:**	Intergreen matrix 2	
4	**Number:**	2	
5			
6		**1**	**2**
7	1		
8	2		
9	3	5	5
10	4		
11	5		
12	6	5	5
13	11	10	10
14	12		
15	13	10	10

10. Hold down the left mouse button and highlight only the rows and columns that contain values.
11. Press the key combination CTRL+C.
12. Switch to the editing view of the new intergreen matrix.
13. Click in the upper left field of the matrix.

    The entire matrix is highlighted in blue.



14. Press the key combination CTRL+V.

    The values are inserted in the matrix.

Notes:

- You can insert parts of the Excel spreadsheet in the intergreen matrix. Correctly position columns and rows of the source area and the destination area.
- No values may be in the gray fields. The process is canceled if you insert data in these fields.

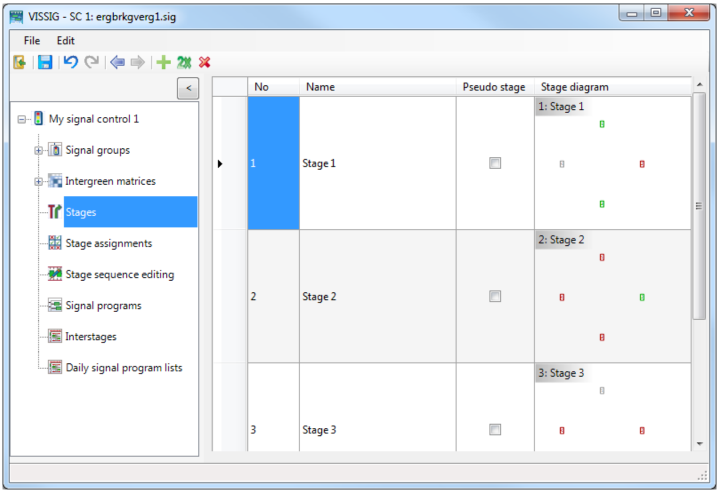
6.14.5.14 Defining and editing stages

Note: You will need the add-on module Vissig.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Click **Stages** in the Navigator.

The stages list opens.

If no stage is defined, only the column titles are shown.



4. Right-click in the list.

   The context menu opens.

5. Click on the desired entry.

Element	Description
Add	Add new stage with the first free number
Duplicate	Copy stage and insert with a new number
Delete	Delete selected stages

6. Click the **Save** button.

7. Make the desired changes:

Element	Description
No.	Number of stage
Name	Name of stage

Element	Description
Pseudo stage	☑ If the option is selected, the stage duration is not changed during green time optimization. The duration of a pseudo stage is zero seconds in all stage-based signal programs. This option acts as a "stage transition divider" to handle the special case when more than one switching command is received during a stage transition of a signal group. In this case, the stage transition is divided into two stage transitions, each linked to a pseudo stage. The **Pseudo Stage** option can only be selected if no stage transition has been defined for this stage.
Stage diagram	Graphical representation of stage

8. Click the **Save** button.

Note: To display pavement markers in Vissim, the following nodes are required:

- A node for the relevant node
- A node for each neighboring node

For these nodes, the attribute **Use for evaluation** must be selected.

6.14.5.15 Editing stage assignment

Note: You will need the add-on module Vissig.

You can permit or block individual signal groups or classify their state as not relevant.

- In the **Default intergreen matrix** box, select the entry of your choice.

  By default, the selected intergreen matrix is marked with an ***x*** *in the overview table of the* ***Intergreen matrices****.*

Switching state of signal group

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Click **Stage assignments** in the Navigator.

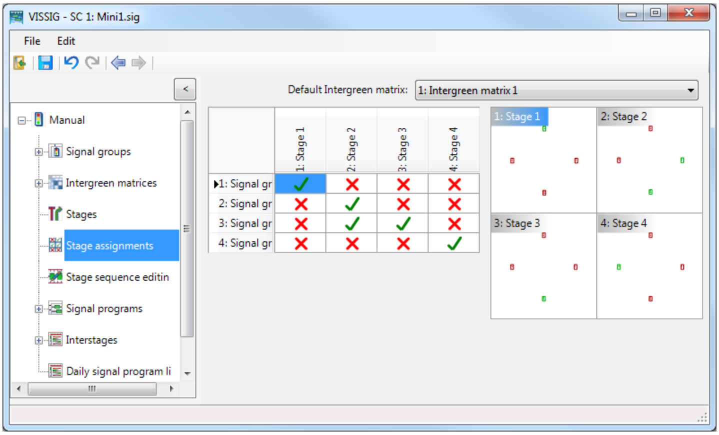
   The stage assignment list opens.
4. Double click on the desired cell.

Tip: Alternatively, you can click the number in the graphical representation of the stage.

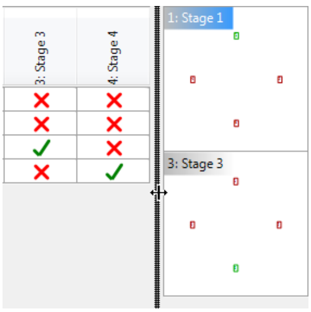
The state of the signal group in the stage is switched. The system switches between the two states in cycles:

✓	Signal group is in the Permitted stage
✗	Signal group is in the Blocked stage
—	State of signal group is in the Not Relevant stage, for example, for partial node control

If an intergreen matrix is selected as default, the conflicts (conflict traffic streams) are tested during the creation of the stages. When conflicting signal groups are permitted in the same stage, these are highlighted in red in the table.



Tip: You may change the width of the two window sections.



Displaying selected or all stages

You can switch between the display of all stages or a magnified display of the selected stage.

1. Right-click in the graphic.
2. From the context menu, choose the function of your choice.

Defining display of pavement markers

You can define the display of the pavement markers with or without displaying the signal group number.

1. Right-click in the graphic.
2. Select the desired function.
3. Confirm with **OK**.

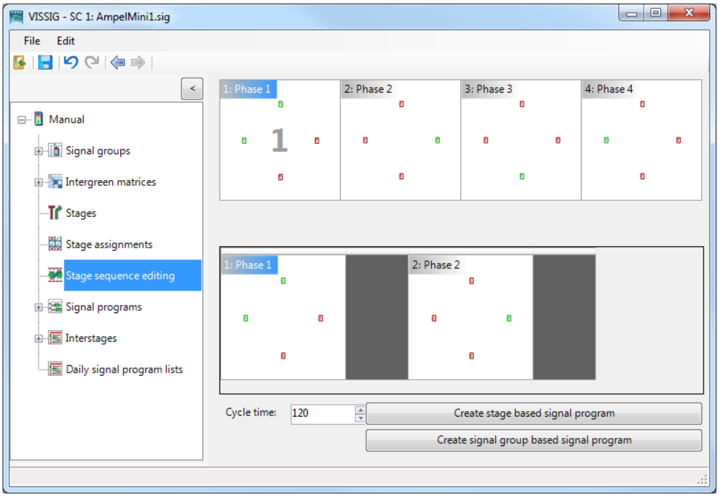
6.14.5.16 Editing stage sequence

Note: You will need the add-on module Vissig.

You can form stage sequences from the stages defined and create the interstages.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. In the navigator, click **Stage sequence editing**.

   The existing stages are displayed in the upper pane. The currently selected stage sequence is displayed in the lower pane.



In the upper pane, you define individual interstages and stage sequences.

Tip: You can change the ratio between the upper and lower pane using the horizontal divider.

Defining interstages

1. Click on the From stage.

   The From stage is marked.

2. Hold down the CTRL key and click on the To stage.

   The stages are called 1 and 2.

3. Right-click in the window.
4. From the context menu, choose **Create an interstage**.

   The interstage is created and graphically displayed.

Tip: You can define several interstages, for example, with and without minimum duration, between two stages.

Notes:

- In Vissig, a signal group in an interstage can be switched once at most: The signal group can change from **Free** to **Blocked** or from **Blocked** to **Free**.
- If you model an interstage in which a signal group is switched twice, in Vissig you must define a pseudo stage and two interstages, one directly before the pseudostage and the other one following the pseudostage.

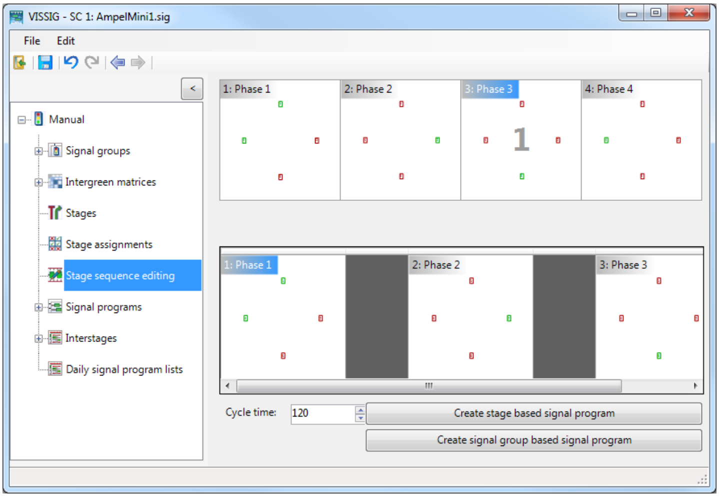
Defining stage sequence

You can define a stage sequence for calculating the signal program in the upper pane.

1. Hold down the CTRL key and click on the desired stage sequence.
2. Right-click in the window.
3. In the context menu, select the entry **Create sequence**.

   The new stage sequence is displayed in the lower view.

4. To add an additional stage to the already defined stage sequence, mark an individual stage and choose **Add to sequence** in the context menu or double-click the stage.



Deleting stage and stage sequences

- To delete a selected stage from the stage sequence, choose **Remove stage** in the context menu.
- To delete the entire stage sequence, choose **Clear sequence** in the context menu.

Exporting stage sequence as graphic file

1. In the menu, choose **Edit** > **Options Export** tab.
2. Make the desired settings for formatting the graphic.
3. From the context menu, choose **Export** entry.

Creating signal program with fixed cycle time

You can create a signal program with a fixed cycle time via the following functions from the stage sequences:

- Enter the cycle duration in seconds in the **Cycle time** field.
- Click on the **Create stage based signal program** button.

  A stage-based signal program with a prescribed cycle duration is calculated. The editing view of the signal programs is opened for further processing of your signal program.

- Click on the **Create signal group based signal program** button.

  A signal group based signal program with a prescribed cycle duration is calculated. The editing view of the signal programs is opened for further processing of your signal program.

Manually defining interstages

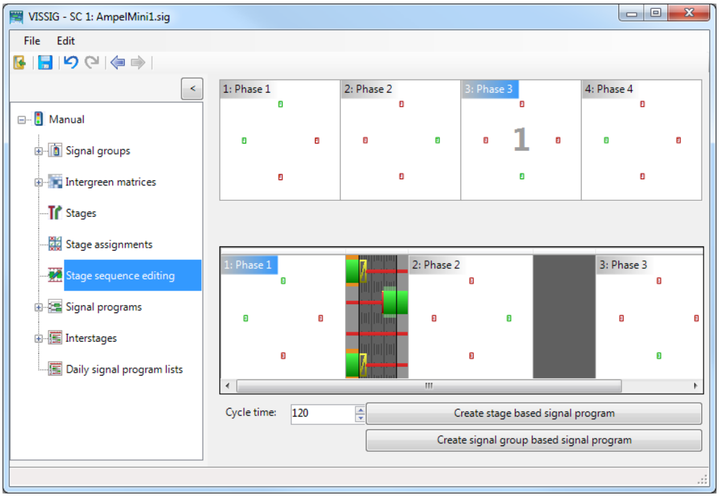
You can manually define the interstages that you want to use to calculate the signal program.

1. Right-click the gray area between the two desired stages.
2. In the context menu, select the entry **Interstages** > **Create** or an existing interstage.

If the interstages are not manually selected, the system checks whether a suitable interstage already exists for each stage change of the stage sequence.

- If a suitable interstage exists, the first suitable interstage is used for the calculation.
- If a suitable interstage does not exist, a new interstage is created. The new interstage is saved for stage-based signal programs.

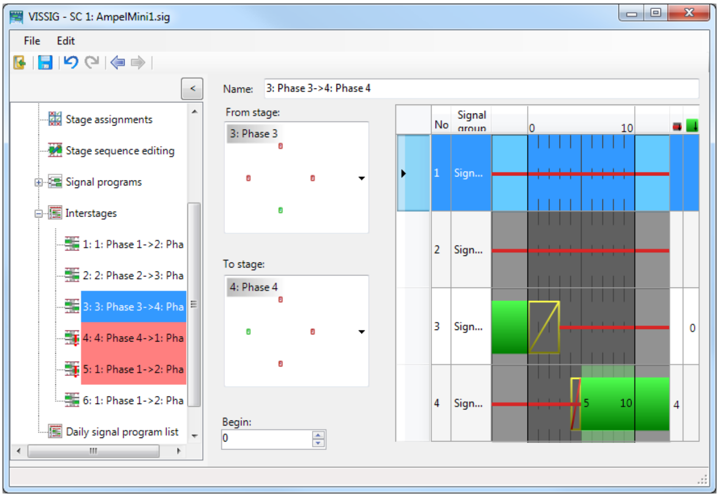
The interstage is deleted again for signal group-based signal programs after the calculation of the signal program.



Editing interstage

1. Double-click on the interstage.

   The editing view of the interstage opens.



2. Make the desired changes.
3. Click on the icon .

   *The **Stage sequence editing** window opens.*

4. Click the **Save** button.

6.14.5.17 Defining signal programs

You may also automatically create signal programs with a fixed cycle time after editing the stage sequence (see "Editing stage sequence" on page 649).

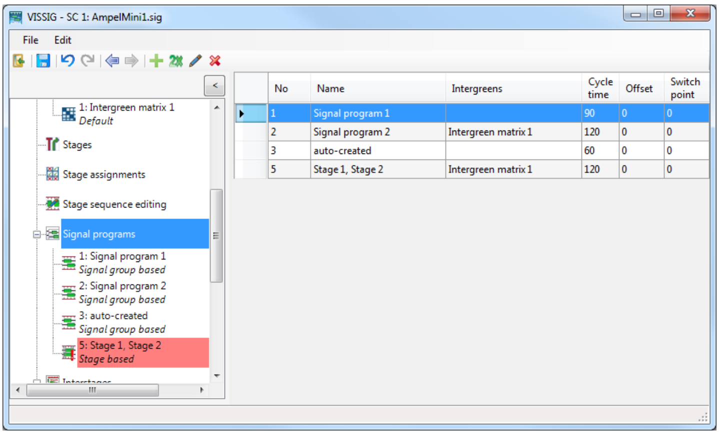
Signal programs have the following data:

Element	Description
Number	Unique number
Name	Description of signal program
Intergreens	Intergreen matrix assigned under Stage assignments
Cycle time	Cycle time defined under Stage sequence
Offset	Offset for the synchronization, for example, within a green wave
Switch point	Switch point at which the system can switch to another signal program, for example, to a daily signal program list

If an intergreen matrix is assigned to a signal program, the adherence to the intergreens can be checked while editing the signal program. The intergreen time violations are displayed automatically.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Double-click **Signal programs** in the Navigator.

   The signal programs list opens.



The **Signal programs** entry in the Navigator can contain the following types of signal programs:

- Signal group-based signal programs. These can be created in the overview table.
- Stage-based signal programs created from stage sequences

The type of signal program is displayed in italics in the Navigator tree below the name of the signal program.

4. Right-click the desired entry in the list of signal programs.

   The context menu opens.

5. Click on the desired entry.

Element	Description
Add	Add a new row to a new signal program
Duplicate	Copy signal program and insert with a new number
Edit	Change data of the selected signal program
Delete	Delete selected signal program
Export	Export graphical display of the signal program

   To sort a column of the list, click the column header.

6. Make the desired changes:

Element	Description
No.	Number of the signal program
Name	Name of the signal program
Intergreens	Used intergreen matrix
Cycle time	Cycle duration in seconds
Offset	Stage scheduling is postponed by this time. For the synchronization, for example, within a green wave.
Switch point	The system can change between two signal programs at this point. Both signal programs must display the same signal states for all signal groups at this point. In the signal program display, a vertical blue line indicates the switch point.

7. Click the **Save** button.
8. Edit the signal program (see "Editing signal programs" on page 654).

6.14.5.18 Editing signal programs

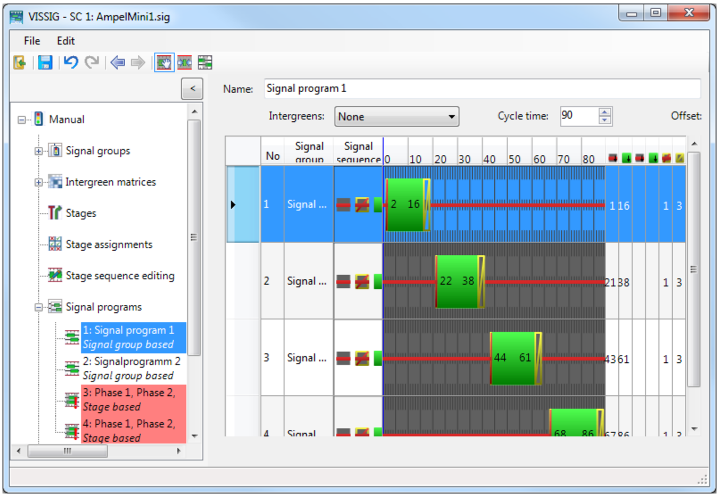
Signal group-based signal programs allow individual editing of individual signal groups and/or switching times. You can edit these graphically and in the time table. You can change the display type of the table columns and the signal states via the context menu.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Double-click **Signal programs** in the Navigator.

   The signal programs list opens.

4. Click on the desired entry.

   The signal program is displayed.

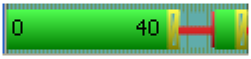


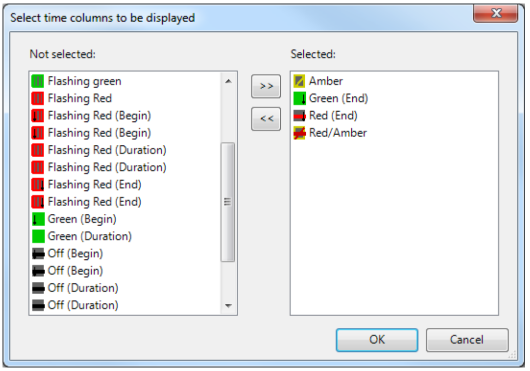
Functions in the context menu

1. Right-click the entry of your choice.

2. Make the desired changes:

Element	Description
Export	From Vissig, you can export stage sequences, signal programs, and interstages as graphic files (see "Exporting data from the SC Editor" on page 669).
Display	View of Signal Programs: - Classic - 3D tubes - 3D boxes
Resize automatically	The row height is automatically adjusted if you change the window size
Show entire signal program	Redraws the signal program and adjusts the row height to the window height
Signal group based editing, stage based editing	Select a row and move a starting point or end point of a signal state of variable duration with the mouse button held down. You can also select the editing mode via the toolbar.
Stretch/Compress	Select a row and expand or compress the signal state in the time axis labeling with the mouse button held down. You can also select the editing mode via the toolbar.

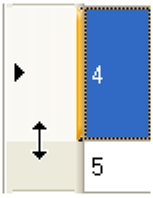
Element	Description
Extract interstage	In Vissig, in the graphical view, you may cut out an interstage of your choice for signal group based signal programs. After selecting this function, the cursor changes into a cross in the Cycle Times column above the graph. With the mouse held down, drag the area that you want to extract. The editing view of the selected interstage opens (see "Editing interstages" on page 661). You can also select the editing mode via the toolbar.
Second green time	Only in signal group-based signal programs: Create additional green time. A second green time (double throw) can be created only via the context menu.  
Time axis labeling absolute (offset considered)	Labeling of the time axis starts with the offset time set, not with 0.

Element	Description
Edit columns…	Only in signal group-based signal programs: the **Select time columns to be displayed** window opens. The settings are saved to the registry of your computer and may change depending on the user.  You can select entries and display them as columns via the arrow symbols or remove the columns. A column is added for table entries of signal states with a fixed duration, as the amber time e.g. is identical for each green time. A double column is added for table entries of signal states with a variable duration, as there can be two green times within one cycle which can have different durations, start and end times. The columns for stage-based signal programs with signal states with a fixed duration are not displayed, even when selected. For stage-based signal programs, the fixed durations of the definition of signal groups are used that cannot be changed.

Changing row height

1. Click on the desired row.
2. Point with the cursor to the line between two rows in the first gray column.

   The cursor changes into a bidirectional arrow.



3. With the left mouse button held down, drag the line up or down.

Functions in the toolbar

You can select the editing mode

Symbol	Description
	Edit signal states
	Stretch/Compress
	Extract interstage

Defining formatting of graphical representation

- In the menu, choose **Edit** > **Options** tab **View** (see "Performing green time optimization of stage-based fixed time controllers" on page 609).

Editing switching times graphically

1. Click on the **Edit signal states** icon.
2. Click on the desired entry.
3. If you move the cursor over the switching time view, you can select one of the following options:

View	Description
	Move signal state with variable duration with the mouse button held down.
	Only for signal group based signal programs: Drag starting time or end time of a signal state with variable duration with the mouse button held down.
	You cannot directly move signal states with a fixed duration (transition states).
	Only for stage-based signal programs: When a signal state is being shifted, the reserve times are indicated by a green background. Intergreen time violations are indicated by an orange background. Violations of minimum durations are indicated by a red background. In front of rows with intergreen time violations or violations of minimum durations, the symbol is displayed.
	Within the time axis labeling, you can move the entire signal program or the zero point of the signal program.

Note: Keep the cursor in the selected area. Otherwise, the move is canceled.

You can change the durations of the transition states individually in the table for the currently edited signal program, if they differ from the default durations defined in the signal group, for example higher amber times in bad weather programs for signal group based signal programs.

Stretching or compressing signal programs

1. Click on the  **Stretch/Compress** icon.
2. If you move the cursor over the time axis labeling, you can select one of the following options:

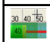
View	Description
	Compressing a signal program: Hold the mouse button down and drag the cursor to the left. The area highlighted in red is extracted.
	Stretching the signal program: Hold the mouse button down and drag the cursor to the right within the time axis labeling. The duration of the area highlighted in green is inserted at the start of this area.
	The highlighted area is displayed with a gray background: Compress/Stretch is not possible (violation of minimum durations).

3. If you want to cancel **Stretch/Compress**, release the mouse button outside the time axis labeling.

Extracting and saving interstage

Only for signal group based signal programs:

1. Click on the  **Extract interstage** icon.
2. Hold the mouse button down and drag the cursor to the right within the time axis labeling.

View	Description
	Valid interstage: The background is displayed in green. After you have released the mouse button within the time axis labeling, the system automatically changes to the editing view of the interstage and saves this interstage. If the From stage and the To stage of the interstage do not exist, these are also created for the extracted interstage. You can edit From stage and To stage (see "Editing interstages" on page 661).
	Invalid interstage: The background is displayed in gray if a amber stage was not completely highlighted. The interstage cannot be saved.
	Only one status change per signal group: If you highlight a second green time at the same time, the background is displayed in gray. No interstage can be saved.

3. If you want to cancel **Extract interstage**, release the mouse button outside the time axis labeling.

Editing stage-based signal programs

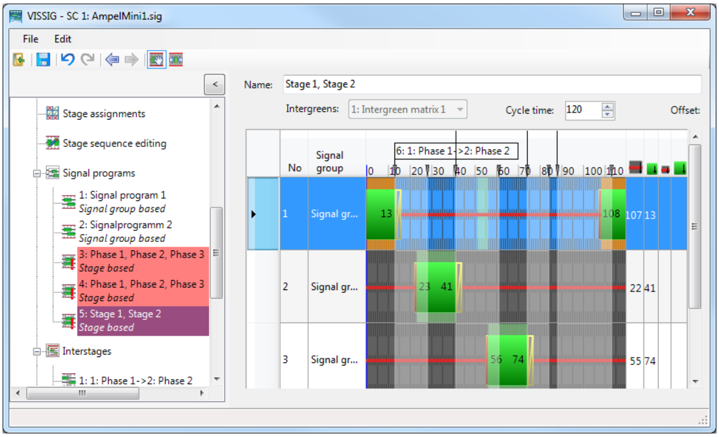
You can change the position of the stages within the signal program.

In stage-based signal programs, it is not possible to

- edit every switching time individually
- extract interstage

1. Make sure:
    - you have defined an intergreen matrix as default (see "Editing stage assignment" on page 647)
    - you have defined stages (see "Defining and editing stages" on page 645)
    - you have defined a stage assignment (see "Editing stage assignment" on page 647)
    - you have edited the stage sequence and created the signal program (see "Editing stage sequence" on page 649)

    *On the toolbar, you have enabled the **Stage based editing** button.*



2. Make the desired changes:

View	Description
	Move the start markings or end markings of interstages
	Move the name of interstages
	Move the interstages within the signal program. This is possible only for interstages with a duration greater than 0 seconds.

The duration of the transition states for interstages are set by default to the default values from the definitions of the signal groups.

3. If desired, compress or stretch the individual stages.

Zooming in on the graphical representation of the signal program

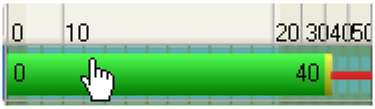
You can enlarge a specific area of the graphical signal programs. Thus you can edit individual switching times more precisely, for example, with a switch point every 1/10 seconds if SC cycle = 10.

1. Click on the desired entry.

The row is highlighted.

2. Click in the graphics area of the row.

   The cursor changes to the following symbol:

3. If you want to stretch the time range, click the scroll wheel in the graphics area of the row.
4. If you want to reset the time range again, click the scroll wheel again in the graphics area of the row.

6.14.5.19 Editing interstages

Note: You will need the add-on module Vissig.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Make sure that stage sequences are defined with interstages (see "Editing stage sequence" on page 649).
4. Click **Interstages** in the Navigator.

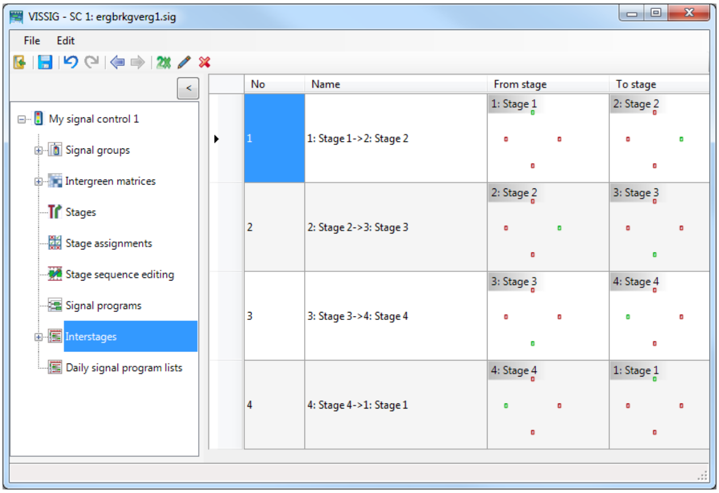
   The interstages are displayed.

5. Right-click the entry of your choice.
6. Choose the desired entry from the context menu.

Element	Description
Duplicate	Copy interstage and insert with a new number
Edit	Change data of the selected interstage
Delete	Delete selected interstage
Export	Export graphic of the selected interstage

7. Edit the interstage:

Element	Description
No.	Number of interstage
Name	Name of interstage. The name is taken from the stage sequences in **Stage sequence editing** in the Navigator (see "Editing stage sequence" on page 649).
From stage	Source stage of Interstage
To stage	Target stage of Interstage
Recalculate	After changing a stage, recalculate the stage sequence and update the display.



8. To edit an interstage, double-click the row with the **No.** and **Name** of the interstage.

The editing of switching times is done like the editing of signal group-based signal programs (see "Editing signal programs" on page 654).

Functions in the context menu of interstages

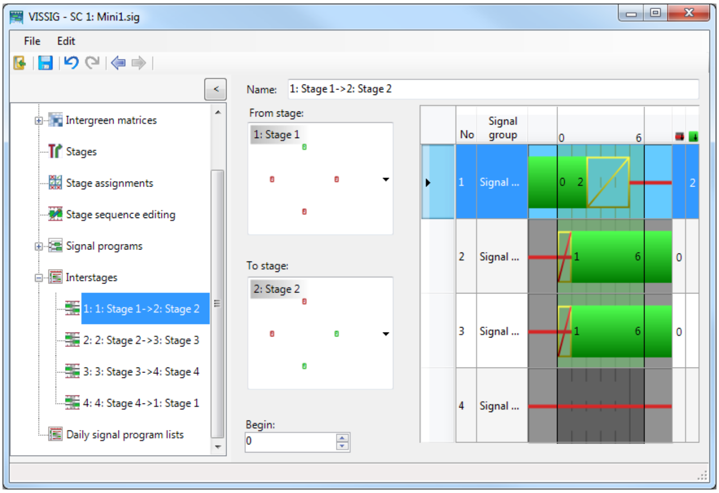
1. Right-click in the right panel of the graphic.

   The context menu opens.

2. Make the desired changes:

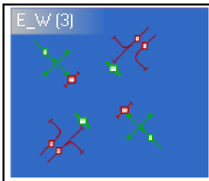
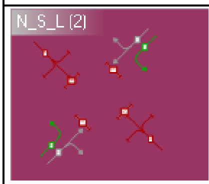
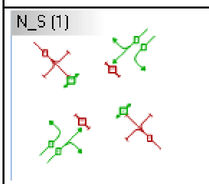
Element	Description
Export	Save interstages as a graphic file (see "Exporting data from the SC Editor" on page 669).
Appearance	Select display: ➤ Classic ➤ 3d tubes ➤ 3d boxes
Resize automatically	The row height is automatically adjusted if you change the window size
Show entire signal program	Redraws the signal program and adjusts the row height to the window height.

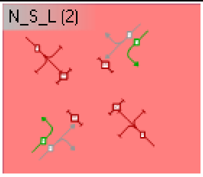
Note: You can export interstages in the menu **File** > **Export** > **PUA** in PUA format. This format is required, for example, for VAP control procedures.



Editing attributes of the selected interstage

1. Select the desired interstage.
2. Make the desired settings in the two areas **From stage** and **To stage**.
3. If you move the cursor over a stage, the possible source stages and target stages are indicated in color in the list box. The selection of a different source stage or target stage automatically starts a recalculation of the interstage.

	Blue background: Indicates the selected stage.
	Purple background: Interstage with red background selected. The selection leads to an automatic recalculation of the interstage.
	White background: This selection does not lead to an automatic recalculation of the interstage.

	Red background: Indicates a stage which is unsuitable for the interstage. The selection leads to an automatic recalculation of the interstage. An interstage is calculated for an initial state (From stage) and a destination state (To stage). If another initial state or destination state is selected, the interstage must be be redefined.
	White background: This selection does not lead to an automatic recalculation of the interstage. Black line: The stage is not relevant.

4. Double-click in the desired stage area of a signal group.

   The state of this signal group in the stage is switched. The interstage is automatically recalculated.

   With the next steps, you can move the initial state to the left (**begin** < 0) and the destination state to the right (**end** > duration of interstage). You can move the initial state to the right until the first state change within the interstage. You can move the destination state to the left until the final state change within the interstage. Switching points are not moved.

5. Select the desired value for the start of the interstage in the **Begin** field.
6. Select the desired value for the end of the interstage in the **End** field.

   A reduction is possible only to the duration of the longest transition state (Amber/Red-Amber).

Note: If the state of a signal group in the source stage or target stage is not relevant when calculating an interstage, no signal switch occurs for this signal group. The same signaling state is displayed within the interstage as in the relevant stage. If the state of a signal group is neither relevant in the source stage nor in the target stage, the signaling state is also displayed as not relevant within the interstage.

6.14.5.20 Defining and editing daily signal program lists

Note: You will need the add-on module Vissig.

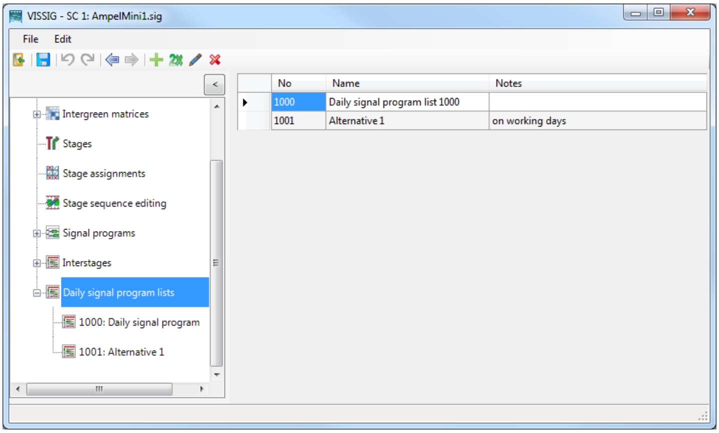
You can create variants of a chronological sequence of signal programs for your time intervals. You can save these variants as daily signal program lists.

You can use the daily signal program lists to switch between various signal programs in the simulation. In this case, enter the number of a daily signal program list rather than the number of a signal program as the program number.

1. Open the SC Editor (see "Opening and using the SC Editor" on page 631).
2. Double-click the desired SC in the Navigator.
3. Click **Daily signal program lists** in the Navigator.

The Daily signal program lists list opens.

If no daily signal program list is defined, only the column titles are displayed.



4. Right-click in the list.

   The context menu opens.

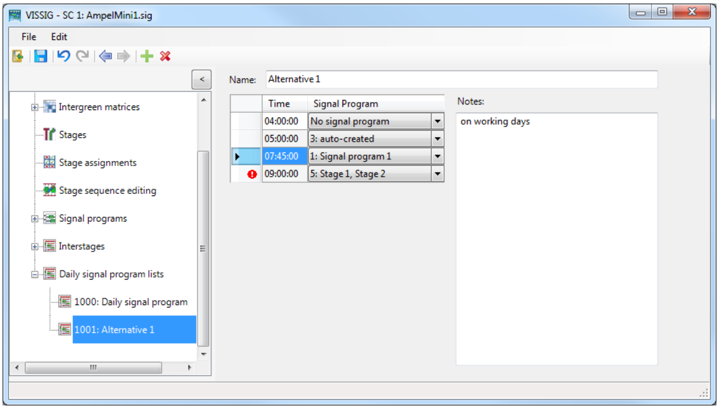
5. Click on the desired entry.

Element	Description
New	Add new daily signal program list with the first free number
Duplicate	Copy daily signal program list and insert with a new number
Edit	Change data of the selected daily signal program list
Delete	Delete selected daily signal program list

Note: Signal times tables and daily signal program lists follow a common numbering system. When you create a daily signal program list, the numbering starts with the first free number. You can overwrite this with a number that is not already used for a signal times table.

6. Click the **Save** button.
7. Right-click the new entry.

   The daily signal program list opens.



8. Right-click in the list.

9. From the shortcut menu, choose **Add**.

10. Enter the desired data.

Element	Description
Name	Optional name for the daily signal program list
Time	Time at which the time interval begins in which the assigned signal program applies, format **hh:mm:ss** Highlight **hh** or **mm** or **ss**. Set the time with the arrow keys: upwards: forward on the time axis downwards: back on the time axis If the daily signal program list does not cover the 24 hours of a day, insert the start of the interval for **No signal program** as the last time. Vissim automatically accepts **No signal program** for the interval from 0:00:00 to the first user-defined time. In this case the signal controls remain black.
Signal Program	Select the signal program that applies in the respective time interval. You can use signal group-based or stage-based signal programs for daily signal program lists.
Notes	optional entry of text

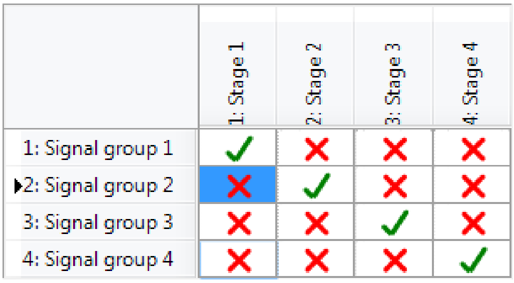
11. Click the **Save** button.

Note: If in daily signal list Vissim marks a signal program with the symbol, the states of this signal program and the next signal program have not been coordinated. Edit one or both signal programs so that the states of the two signal programs are coordinated for the change (see "Editing interstages" on page 661).

6.14.5.21 Detecting inconsistent planning

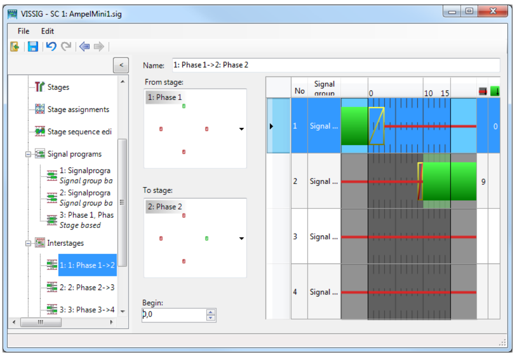
Due to the dependencies between the individual data objects, changing an object can lead to inconsistencies in objects which are dependent on it. These inconsistencies are permitted expressly so that you can make settings as flexible as possible and test them. To support consistent planning, the following tests and mechanisms are implemented:

- A change in the intergreens can lead to intergreen violations occurring in the associated signal programs and/or interstages. These are displayed graphically when you open the object.
- If additional conflicts are added, it can lead to invalid stages. In the **Stage assignments** area, conflicting green is highlighted by red cells:

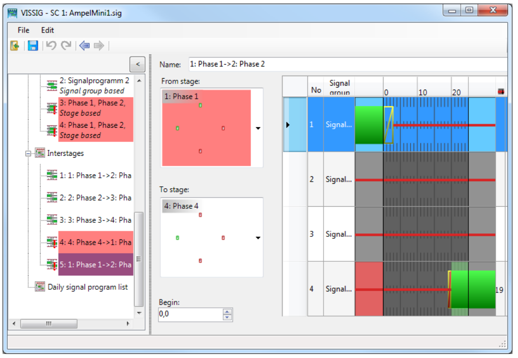


	1: Stage 1	2: Stage 2	3: Stage 3	4: Stage 4
1: Signal group 1	✓	✗	✗	✗
2: Signal group 2	✗	✓	✗	✗
3: Signal group 3	✗	✗	✓	✗
4: Signal group 4	✗	✗	✗	✓

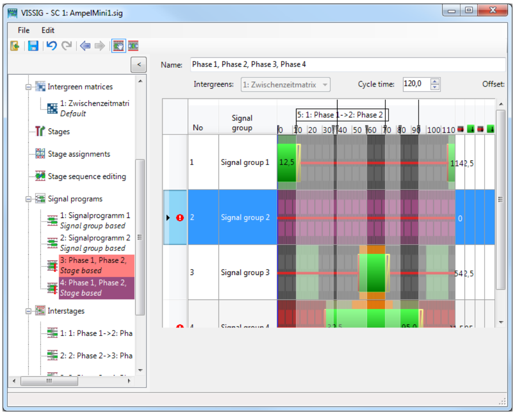
- A change in the stages may cause interstages to become invalid because the initial state or target state of a signal group has changed. The following cases can be distinguished:
  - If the state of a signal group is changed from **Permitted** or **Blocked** to **Not relevant**, the related interstages remain consistent. Since the interstage is consistent, it is not recalculated when you select the modified stage again. To recalculate the interstage, you must first select an arbitrary stage and then the modified stage. In the figure, in stage 2, the state was changed from **Blocked** to **not relevant** for **N RS** and **S RS**:



  - If the state of a signal group is changed to **Permitted** or **Blocked**, the associated interstages are usually inconsistent. This is indicated by the names of the inconsistent interstages highlighted in red in the navigator tree. In the figure, in stage 2, the state was changed from **Blocked** to **Permitted** for **S RS**:



- In the editing view of an inconsistent interstage, the schematic display of the responsible stage is highlighted in red and the different stage states in the rows are also highlighted in red. The explicit selection of the stage highlighted in red starts a recalculation of the interstage so that this is now consistent with the modified stage.
- A change of interstages can cause the associated stage-oriented signal programs to be inconsistent. This is indicated by the names of the inconsistent signal programs highlighted in red in the navigator tree.
- Rows that have an inconsistent signal sequence due to the change of interstages are marked in the editing view as incorrect and the inconsistent area is highlighted in red. Interstage 2 was recalculated after the above change of stage 2 by the explicit selection of stage 2:



You can check the signal program for inconsistencies:

1. From the **File** menu, choose > **Check**.

If in the signal program, errors are found in elements, a treeview opens with these elements highlighted in red.

2. Correct these elements.

6.14.5.22 Exporting data from the SC Editor

You can export the following data:

- Graphic files
- Stage sequences
- Signal programs
- Interstages
- **.pua* file
- Export to Microsoft™ Excel™ only with Vissig
- Include graphic files in Microsoft™ Word™

Exporting graphic files

You can export stage sequences, signal programs and interstages as graphic files. You can format graphics in the SC Editor. From the **Edit** menu, choose **Options**. Then click the **Export** tab.

You can export the following file formats:

- *.bmp
- *.gif
- *.jpg
- *.png
- *.svg
- *.tif

Exporting a stage sequence

You can export the graphical representation of a stage sequence.

1. Generate a stage sequence (see "Editing stage sequence" on page 649).
2. Right-click the lower part of the **Stage sequence editing** area.
3. From the context menu, choose **Export** entry.

   *The **Save as** window opens.*
4. Choose the desired folder.
5. Specify a **File name** for the graphic file.
6. Select a **File type**.
7. Click on **Save**.

Exporting signal programs

1. Choose **Signal programs** in the Navigator.

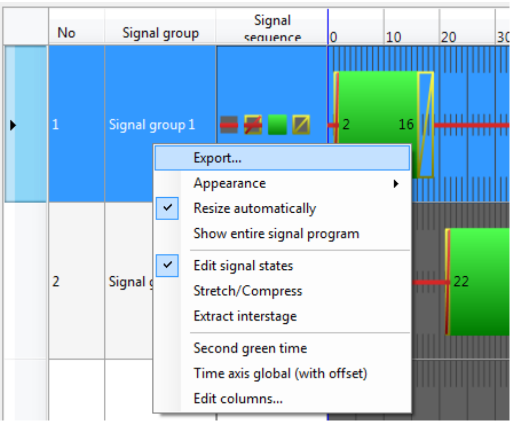
2. Select a program in the list.
3. Right-click it.
4. From the context menu, choose **Export** entry.

   The **Save as** window opens.
5. Choose the desired folder.
6. Specify a **File name** for the graphic file.
7. Select a **File type**.
8. Click on **Save**.

Exporting interstages

1. Choose **Interstages** in the Navigator and select an interstage in the list.
2. Right-click it.

   The context menu opens.



3. Choose **Export** in the context menu.

   The **Save as** window opens.
4. Choose the desired folder.
5. Specify a **File name** for the graphic file.
6. Select a **File type**.
7. Click on **Save**.

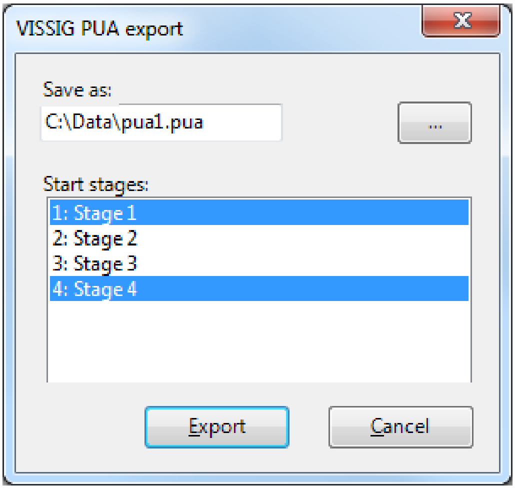
Exporting *.*pua* file

Note: You will need the add-on module Vissig.

The **.pua* file is an output file of Vissig and an input file for a VAP control logic. Interstages can be exported in the **.pua* file format.

1. Choose the menu **File** > **Export** > **PUA**.

   *The **VISSIG PUA export** window opens.*



Note: Settings that do not correspond to the conventions of the **.pua* file format are displayed in the **Export** window, for example, spaces in the file name of a signal group. The correction is made in accordance with the conventions. The details are displayed in the **Export** window. If you close the **Export** window, the export will be continued.

2. In the **VISSIG PUA export** window, click the button  next to **Save as**.

   *The **Save as** window opens.*

3. Choose the desired folder.
4. Specify a file name.
5. Click on **Save**.
6. Select the **Start stages** in the **VISSIG PUA export** window.
7. Click on **Export**.

*The *.pua file is generated and saved in the selected folder.*

Export to Microsoft™ Excel™ (only with Vissig)

All information of the fixed time controller can be exported in an Excel workbook.

1. Choose the menu **File** > **Export** > **Excel workbook**.

   *The **Save as** window opens.*

2. Choose the desired folder.

3. Specify a **File name** for the Excel file.
4. Click on **Save**.

Note: The Excel file is saved as an **.xlsx* file, the format of Microsoft™ Excel™ 2007. The file can also be opened with Microsoft™ Excel™ 2003, but must be converted to the older format for this purpose. You must install the Microsoft Office Compatibility Pack in this case. You can download this in the Microsoft™ Download Center (www.microsoft.com).

Tip: You can also open the **.xlsx* file with the program Calc (OpenOffice.org). Before you make changes to the file in Calc, save the file under a new name in the Calc format.

6.14.6 Linking SC

You can link SCs, which support communication with other control devices. In each case, an output of an SC is linked to an input of another SC. Depending on the control procedures, for example, marks can thus be exchanged between several SCs. The number of links between two light signal controls is unlimited.

1. From the **Signal Control** menu, choose **Signal Controller Communication**.

   *The **SC Communication** list opens. If there is no link, only the column titles are shown.*

   By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button ✚ .

   A new row with default data is inserted.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

3. Make the desired changes:

Element	Description
FromSC	SC from which an output is linked
FromChn	Number of the output channel of the SC
ToSC	SC from which an input is linked
ToChn	Number of the input channel of the SC

In the next time step, the data that was written from the signal controller to the output channel is transmitted to the linked input channel, where it can be read by its control.

Example with VAP control procedures

Definition of SC Communication from SC 1, output 7, to SC 3, input 5. In control logic, the following commands can be used for transmitting and receiving data:

- Within the control logic of SC 1, the following command sets the output value of output 7 to 1:

  ```
  Marker_Put( 7, 1 )
  ```

- In the next time step of the simulation, the control logic of SC 3 can read this value from input 5 using the following command. The variable for the input must be numeric:

  ```
  Value := Marker_Get( 5 )
  ```

*The user-defined variable **Value** is set to 1 and can be used in the subsequent program run.*

6.14.7 Modeling railroad block signals

You can use Vissim to model the operation of railroad block signals and railway traffic, including train protection systems according to the German regulations for tram construction and operation (BOStrab).

Trains are not driven "on sight". The drivers rely on signals and may only drive within blocks released for passage. A block is defined as the area between two block signals. Only after a block has been left by one train is the next train allowed to enter it. Entry to the blocks is controlled via block signaling.

In Vissim you model blocks via signal heads. Signal heads defined as block signals do not belong to a signal group or signal control.

1. Place the signal heads on the link at the desired block distance (see "Defining signal heads" on page 579).
2. For the signal heads, select the **Block signal** option (see "Attributes of signal heads" on page 579).
3. Into the **Amber speed** box, enter the desired speed (see "Attributes of signal heads" on page 579).

Every second, block signals identify the status of their adjacent two blocks downstream:

- If a vehicle is identified in the next block, the signal switches to red. The block is closed for entry.
- If a vehicle is identified two blocks away, the signal switches to amber: A vehicle passing the amber signal (reduced speed area), is allowed the "desired speed". This speed is kept until the vehicle passes a green block signal further downstream.
- If no vehicle is identified in either of the blocks, the signal shows green: i.e. free passage at the desired speed specified.

If, by default, signal heads are assigned to a signal control, block signals treat them as delimiters. They are, however, not controlled via block signals.

6.15 Using static 3D models

A static 3D model is a realistic model of a building, a plant or another static object. In the Network editor, you can insert and edit static 3D models in 2D and 3D mode.

By default, Vissim saves 3D models to the *..\exe\3DModels* directory during the installation.

The depiction of 3D models requires more computation time. This is why switching to the 3D mode might take a few seconds.

You can add 3D models of the following file formats:

Supported file format	Model file
***.v3d**	Vissim-3D
***.skp**	SketchUp- files **.skp* can be imported, edited and loaded into the network. SketchUp files need not be converted via V3DM into a **.v3d* file. ➤ In Vissim 7, in the 32-bit version only, you can add Sketchup models up to and including version 2014. ➤ From Vissim 8, in the 32-bit version, Sketchup models up to version 2014 can be added. In the 64-bit version, Sketchup models up to version 2015 may be added. ➤ For Vissim 10, in the 32-bit version, Sketchup models up to version 2014 can be added. In the 64-bit version, Sketchup models up to and including version 2017 may be added. ➤ For Vissim 11, in the 64-bit version, SketchUp models up to and including version 2017 may be added.
***.3ds**	Autodesk 3ds Max
***.dwf**	Autodesk Design Web Format

You can use the add-on module **V3DM** (Vissim 3D Modeler) to convert static 3D models in the Autodesk 3ds Max format **.3ds* into the Vissim 3D format **.v3d*. **V3DM** also enables you to create simple 3D models, e.g. buildings. Using textures for the faces, allows you to give them a realistic design.

6.15.1 Defining static 3D models

You can insert static 3D models in the 2D and 3D mode.

1. In the Network Objects toolbar, click on the button for **Static 3D Models**.
2. In the Network Editor, right-click the position in which you want to insert the 3D model.
3. From the context menu, choose the entry **Add New Static 3D Model**.

   *The **Open** window opens.*
4. Select the desired directory, for example the installation directory of your Vissiminstallation *..\exe\3DModels\Static*.

By default, static 3D models are saved to the ..\Static directory. For example models of benches, traffic cones, plants, buildings, overhead lines, tracks, road block equipment, sky texture, stop lights, stops, garbage cans.

5. Double-click the desired file: **.v3d*, **.skp*, **.3ds* or **.dwf*.

   A symbol for the 3D model is displayed in the Network Editor.

   *The **Add static 3D model** window opens. In the section on the left, a preview of the 3D model is displayed.*

6. Make the desired settings for the attributes (see "Attributes of static 3D models" on page 675).

   The window also provides the following commands:

Element	Description
Preview in section on the left	3D display of the selected 3D model. ▸ Zoom: Turn the mouse wheel. ▸ Rotate the model: Left-click and keep the mouse button pressed while moving the mouse pointer in the desired direction.
3D model file	Path and file name of the selected 3D model file
	Opens the **Open** window for selection of a file with a static 3D model
	Resets the Preview to default settings. Does not reset the values of attributes in the **Angle** section.
	Adjust visualization: Show options that allow you to display additional elements in the preview. For example, **Show ground plate**: ☑ Select this option to display a gray, transparent ground plate below the 3D model. This view supports the orientation during rotation and tilting of the 3D model in the preview. ☐ If you do not select this option, no ground plate is displayed. This view corresponds to the view provided by the Network editor.
Angle	Angle of rotation around the three axes (see "Attributes of static 3D models" on page 675)

In the 2D mode, in the Network Editor, the contour or outline of the 3D model is displayed and selected.

In the 3D mode, in the Network Editor, the 3D model is displayed and selected.

You can edit the static 3D model (see "Editing static 3D models in the Network Editor" on page 676).

6.15.2 Attributes of static 3D models

Attributes of static 3D models may also be changed during a simulation run.

1. Under the **Lists** menu, select **Graphics & Presentation** > **Static 3D Models**.

*The **Static 3D Models** list opens.*

By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains, amongst others, the following attributes:

Short name	Long name	Description
ModelFilename	**Model file-name**	Name of **.v3d* file of the static 3D model
Level	**Level**	Level at which the static 3D model lies
CoordX	**Coordinate (x)**	Coordinate (x) of the position of the static 3D model in the network
CoordY	**Coordinate (y)**	Coordinate (y) of the position in the network
CoordZOff	**Coordinate (z-Offset)**	Base height of the static 3D model across the level
Scale	**Scale**	Zoom in or out of the static 3D model in the Network editor, default value 1
YawAngle	**Yaw angle**	Yaw angle: angle of rotation around z axis
PitchAngle	**Pitch angle**	Angle of rotation around the y-axis
RollAngle	**Roll angle**	Angle of rotation around the x-axis
State	**State**	Number of the desired state in which the static 3D model should be displayed, if the static model has different states (see "Defining 2D/3D models" on page 220).

6.15.3 Editing static 3D models in the Network Editor

You can edit static 3D models in Network Editors.

6.15.3.1 Moving 3D model in network level

You can move 3D models in 2D or 3D mode incrementally in steps of 22.5 degrees. This may simplify positioning.

1. Click the 3D model in the Network Editor, hold the mouse button and the SHIFT key pressed, and drag the 3D model to the desired position.
2. Release the keys.

6.15.3.2 Rotating 3D model around its own axis

You can rotate 3D models in 2D or 3D mode.

1. Click the 3D model in the Network Editor, hold the mouse button and the ALT key pressed, and rotate the 3D model.
2. Release the keys.

6.15.3.3 Scaling 3D model

You can scale 3D models in 3D mode. In the Network Editor toolbar, the icon **2D/3D** must be shown (not).

1. Select the 3D model in the Network Editor.
2. Press and hold the SHIFT key, right-click the 3D model in the Network Editor, and hold the right mouse button pressed.
3. Drag the mouse to the right if you want to make the 3D model smaller.
4. Drag the mouse to the left if you want to make the 3D model larger.
5. Release the keys.

6.15.3.4 Changing the vertical position of a 3D model

You can change the Z offset of 3D models in 3D mode. In the Network Editor toolbar, the icon **2D/3D** must be shown (not).

1. Select the 3D model in the Network Editor.
2. Press and hold the ALT key, right-click the 3D model in the Network Editor, and hold the right mouse button pressed.
3. Drag the mouse upward if you want to raise the position of the 3D model.
4. Drag the mouse downward if you want to lower the position of the 3D model.
5. Release the keys.

6.16 Modeling sections

Using sections, you can record data of network objects during a simulation, e.g. of pedestrians on pedestrian areas or of vehicles on links. You can save this data to animation recordings and save the pedestrian data to an evaluation. (see "Recording a simulation and saving it as an ANI file" on page 1166), (see "Evaluating pedestrian areas with area measurements" on page 1041). A section lies on a level and can extend over parts of the Vissim, e.g. over several network objects links, connectors and construction elements.

Areas which are not covered by construction elements do not become walkable areas due to sections. The measurement includes only pedestrians from the part of the pedestrian area that is covered by the section.

When the size of the walkable pedestrian area is used in a measured value, for example, the density, the percentage of the walkable area of Vissim is calculated: Thus covered areas and areas without a construction element are deducted as obstacles.

You can define sections as polygons or rectangles (see "Defining sections as a rectangle" on page 678), (see "Defining sections as a polygon" on page 679).

If during hybrid simulation of your Vissim network, you want to choose microscopic simulation for parts of the network, you will need to define sections for those parts (see "Using add-on module for mesoscopic simulation" on page 801). You select these sections in the simulation parameters for mesoscopic simulation (see "Selecting sections for hybrid simulation" on page 838).

The following applies for modeling sections in mesoscopic simulation:

- The borders of these sections must not lead through a meso node.
- The borders of these sections must not intersect with certain network objects. These e.g. include parking lots, conflict areas, priority rules, public transport stops and reduced speed areas.
- Ensure that the section borders are at a sufficient distance from existing meso nodes, including those automatically generated, in order to avoid artifacts in the behavior of vehicles in the transition area.

6.16.1 Defining sections as a rectangle

1. On the Network objects toolbar, click **Sections**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button ▾ is shown.

2. Click **Sections** again.
3. Select **Rectangle**.
4. In the Network editor, using the mouse pointer, point to the desired position of the first corner point of the section.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

5. Press the CTRL key, hold down the right mouse button and drag the mouse to the desired end point of the link.
6. Release the keys.

7. Drag the mouse pointer sideways to the desired width.
8. Double click.

   *The section is shown in color in the Network editor. The **Sections** list opens.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

   A new row with default data is inserted.

9. Edit the attributes of the section (see "Attributes of sections" on page 679).

6.16.2 Defining sections as a polygon

1. On the Network objects toolbar, click **Sections**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Sections** again.
3. Select **Polygon**.
4. Press the CTRL key and right-click the start point of your choice.
5. In order to insert multiple polygon points, repeat the next two steps.
6. Press the CTRL key and drag the mouse pointer in the direction of your choice.
7. Right-click the next desired point.
8. If you do not wish to insert a further polygon point, double-click the left mouse button.
9. Release the keys.

   The last polygon point is connected with the first polygon point. The polygon is closed.

   *The section is shown in color in the Network editor. The **Sections** list opens. A new row with default data is inserted.*

10. Edit the attributes of the section (see "Attributes of sections" on page 679).

6.16.3 Attributes of sections

The **Sections** list opens automatically when you insert a network object and have selected to automatically open a list after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Sections list is opened.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

1. Make the desired changes:

Element	Description
No	Unique number of the section
Name	Name of the section
Display type	Color of section (see "Defining display types" on page 320)
Level	For modeling of multistory buildings: level at which the section lies.
AddLvl	**Additional Level**: If an additional level is selected, only vehicles on those connectors and/or pedestrians those on ramps are recoded that connect the level with the additional level of the section. The same applies when the level and additional level of the section are identical. Vehicles on links and pedestrians in areas are not recorded when an additional level is set. The selection of an additional level affects: - Animation recordings (see "Recording a simulation and saving it as an ANI file" on page 1166) - Area measurements (see "Defining an area measurement in lists" on page 1009), (see "Generating area measurements in lists" on page 1010), (see "Evaluating pedestrian areas with area measurements" on page 1041) - Vehicle record (see "Saving vehicle record to a file or database" on page 1031) - Pedestrian record (see "Saving pedestrian record to a file or database" on page 1053) - Mesoscopic simulation (see "Using add-on module for mesoscopic simulation" on page 801) If no layer is selected in the attribute **Additional Level**, no vehicles on connectors and/or pedestrians on ramps are recorded that connect the two levels. If no additional level is selected, the section recording also includes ramps and connectors whose beginning and end lie on the level of the section. Those whose do not, meaning those who have only one end or none lying on the level of the section, are not recorded.
GeometryType	**Geometry type**: rectangle or polygon: - Rectangle - Polygon

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Area measurements (see "Defining an area measurement in lists" on page 1009), (see "Generating area measurements in lists" on page 1010)
- Points: edit coordinates of the corners

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

6.17 Using the 3D information signs

3D info panels display information about network objects in the network editor. In the network editor, you can insert 3D information signs and assign them to static and/or dynamic network objects (see "Defining 3D information signs" on page 682).

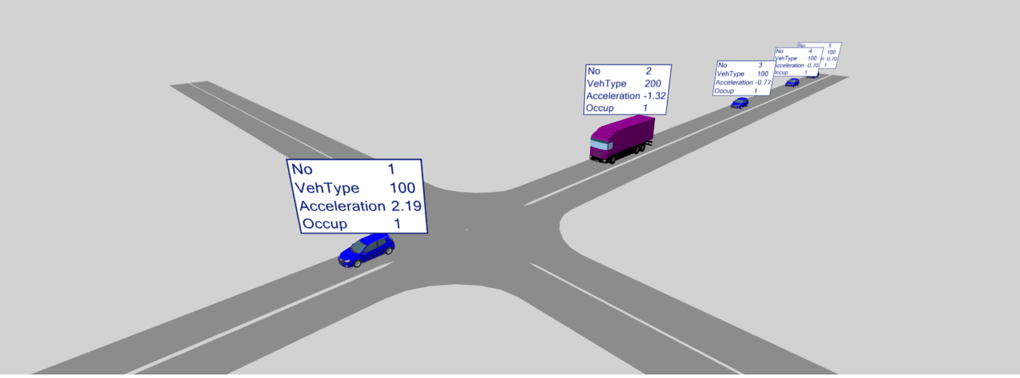
Content of 3D information signs

3D information signs may contain the following information:

- Text that you enter
- Attribute value
- Attribute title
- Attribute title and attribute value
- Text and attribute value

Displaying 3D information signs

You can display 3D information signs in the network editor during simulation and animation.



To align a 3D information board to the camera position, use the **Alignment** attribute (see "Attributes of 3D information signs" on page 683).

In the network editor, In 2D mode, the following symbol indicates a 3D information sign: . Point the mouse pointer to the symbol to show the name of the 3D information board.

Note: 3D information signs reduce display and loading speed of Vissim. Use 3D information signs to display information, not to label network objects. Limit the number of 3D information signs you want to save in the **.inpx* file to a maximum of a few dozen and avoid a larger number.

6.17.1 Defining 3D information signs

In the network editor, in 2D and in 3D mode, you can select a network object and define a 3D information sign for it. The 3D information sign is then automatically assigned to this network object and you can configure its display and content. To define a 3D information sign in the **3D Information Signs** list, as **Reference object type**, select the network object type and the desired network object at which you want to display the 3D information sign.

Note: 3D information signs reduce display and loading speed of Vissim. Use 3D information signs to display information, not to label network objects. Limit the number of 3D information signs you want to save in the **.inpx* file to a maximum of a few dozen and avoid a larger number.

6.17.1.1 Defining 3D information signs in the network editor

1. On the network objects toolbar, click **3D Information Signs**.
2. Right-click the network object to which the 3D information sign is assigned.
3. From the shortcut menu, choose **Add 3D information sign**.

   *The **3D Information Signs** window opens.*
4. Edit the attributes (see "Attributes of 3D information signs" on page 683).
5. Confirm with **OK**.

*The attributes are saved in the **3D Information Signs** list.*

6.17.1.2 Defining 3D information signs in the 3D information signs table

1. From the **Lists** menu, choose > **Graphics & Presentation** > **3D Information Signs**.

Tip: Alternatively, from the **Presentation** menu, choose > **3D Information Signs**.

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

   *The **3D Information Signs** window opens.*
3. Edit the attributes (see "Attributes of 3D information signs" on page 683).

4. Confirm with **OK**.

*The attributes are saved in the **3D Information Signs** list.*

6.17.2 Positioning 3D information signs

You can specify the position of the 3D information sign:

- In 2D mode, the 3D information sign symbol is displayed.
- In 3D mode, the 3D information sign is displayed.

1. Click the symbol of the desired 3D information sign and keep the mouse button held down.
2. Drag the mouse pointer to the desired position.

   A dashed yellow line connects the original and current position. xOffset and yOffset of the 3D information sign are adjusted. In 2D mode, the symbol of the 3D information sign is displayed at the new position. In 3D mode, the 3D information sign is displayed at the new position.
3. Release the mouse button.

6.17.3 Attributes of 3D information signs

1. From the **Lists** menu, choose > **Graphics & Presentation** > **3D Information Signs**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

*The **3D Information Signs** list opens.*

The list in the tab contains, amongst others, the following attributes:

Short name	Long name	Description
Name	**Name**	Name of the 3D information sign
ObjType	**Object type**	Identifier of the network object type labeled with the 3D information sign
ObjTypeName	**Object type name**	**Reference object**: Name of network object type labeled with the 3D information sign
ObjKey	**Object key**	**Reference object**: Number of network object labeled with the 3D information sign
Level	**Level**	Level on which the network object labeled with the 3D information sign is positioned

Short name	Long name	Description
x-off	**x-offset**	relative horizontal position of the 3D information sign on the x-axis
y-off	**y-offset**	relative horizontal position of the 3D information sign on the y-axis
z-off	**z-offset**	relative vertical position of the 3D information sign on the z-axis
BackgroundColor	**Background color**	Background color of the 3D information sign area
FrameColor	**Frame color**	Color of the outer edge of the 3D information board
FrameWidth	**Frame width**	Width of the outer edge of the 3D information board. Default 0.1 m
FontFamily	**Font family**	Font used for text. Default value Microsoft Sans Serif
FontSlopeItalics	**Font slope italics**	Display text in italics. Not selected by default
FontStyleBold	**Font style bold**	Display text in bold. Not selected by default
FontSize	**Font size**	Size of the text. Default 0.8 m
FontColor	**Font color**	Color of the text
Alignment	**Alignment**	Automatic alignment of 3D information sign to camera position when camera position changes: ▸ **Fixed orientation**: The 3D information sign is always aligned to the direction of the rotation angle. ▸ **Rotation to camera**: The 3D information sign always rotates towards the camera position. The tilt remains constant. ▸ **Rotation and tilt to camera**: The 3D information sign always rotates and tilts towards the camera position.
RotAngleZ	**Rotation angle (around z-axis)**	3D information sign displayed in 3D, with angle of rotation to the south.
WidthMax	**Width /Maximum)**	Maximum width of 3D information sign. If the 3D information sign text is longer, it is wrapped and the sign increases in height.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click > **Content**.

The right list contains the attributes and attribute values of the content of the 3D information sign that was selected in the left list (see "Using coupled lists" on page 119).

3. Enter the desired data.

The data is allocated.

6.18 Visualizing turn values

You can visualize turn values in 2D mode. Turn values represent the values of result attributes or user-defined attributes along the turn relations in a node. In the Network editor, you can visually compare turn values that arise in one or multiple nodes.

Selecting attributes for visualization

In the graphic parameters of the network object type **Node**, when you configure the display settings for turn values, you select two attributes (see "Configuring turn value visualization" on page 687):

- An attribute whose value Vissim is visualized using the color of the turn value. Each color is assigned to a class. You can select the value range for each class.
- An attribute whose value Vissim visualizes using the width of the turn value. You can select the scaling of the width. Vissim can also display the attribute value numerically, at the beginning and end of each turn value. In this case, th following elements must be selected large enough. Otherwise not all or no numerical values are displayed:
  - the section of turn value visualization
  - the value for **Turn value bar width (maximum):**
  - the magnification in the Network editor

  In addition to numeric attribute values, you can show total values (see "Configuring turn value visualization" on page 687). Circular segmental arches separate the total values from the numeric attribute values.

Activate turn value visualization

To visualize turn values in the Network editor, in the graphic parameters of the network object type**Node**, select **Turn value visualization**.

The node attribute **Object visibility** also controls the visibility of turn values.

In addition, you can activate turn value visualization for each node of your choice (see "Activate turn value visualization" on page 690). This allows you to compare turn values that arise in one or multiple nodes during or after a simulation run. During the simulation run, turn values are visualized based on the **interval** set for the evaluation of the **node**.

Selecting node evaluation

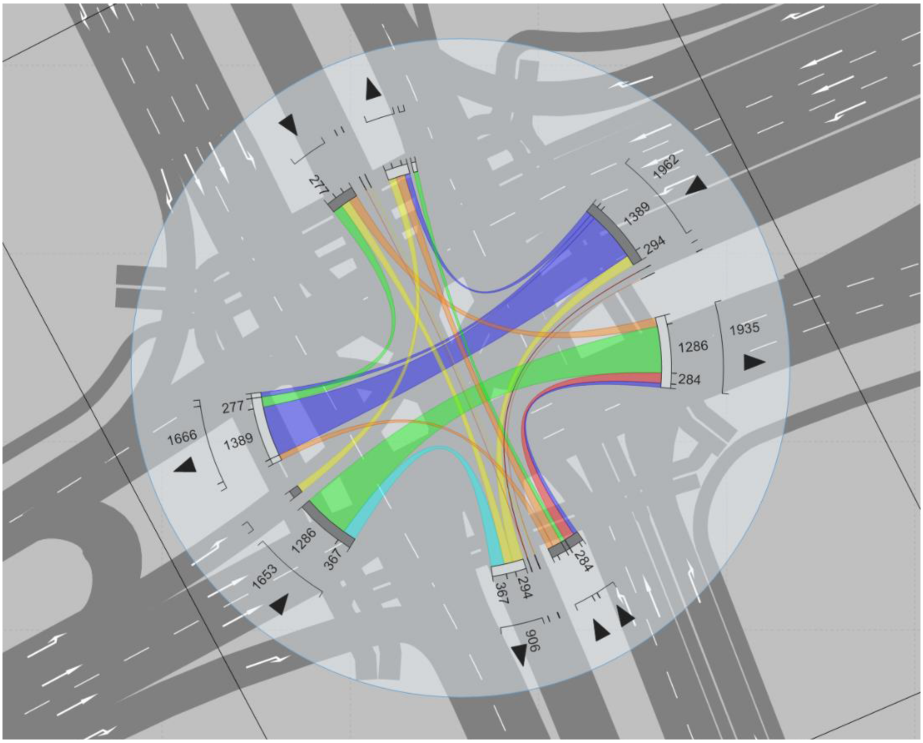
To perform a **Node** evaluation, first select the attribute **Collect dataa** (see "Evaluating nodes" on page 1057). For each node whose turn values you want to visualize, select the **Use for evaluation** attribute.

The evaluation graph must have been generated (see "Generating a node-edge graph" on page 718).

Graphical display of turn value visualization

Elements from outwards tow inwards:

Light blue circle with blue outline	Defines the range of the turn value visualization. You can enlarge or reduce this area (see "Editing the size of turn value visualization for a node" on page 690).
Arrow tip	Shows the direction of travel.
Value next to the arrow tip	If the light blue circle is sufficiently large: Sum of the attribute values of turn value bands for each input and output
Circle segment arc	Separates arrow tips and sums from attribute values.
Circle segment	➤ Dark gray: highlights turn values of turn relations that lead into the node. ➤ Light gray: highlights turn values of turn relations that lead out of the node.
Value between circle segment and turn value band	If the light blue circle is sufficiently large: numeric attribute value at the beginning and end of each turn value band.
Turn value band	Each turn relation in the node, from entry to exit, is represented by a turn value band. The width at the beginning and end and the color are based on the configuration of turn value visualization and the values of the selected attributes (see "Configuring turn value visualization" on page 687). In order to avoid overlapping turn value bands during visualization, their width is reduced in the middle. The width of that part of the band is not used to visualize any values. Width and color of the turn values displayed already change during simulation, as they are based on values determined using the **interval** set for **node** evaluation.



Example:

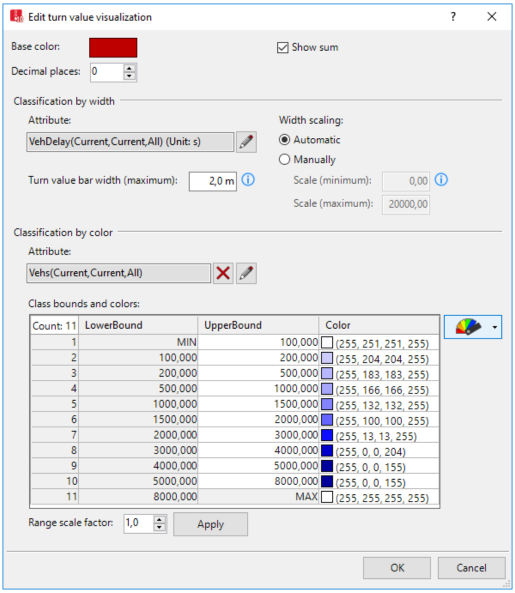
The graphic shows a turn value visualization with the following settings:

- Color of turn values: The values of the attribute **Vehicle delay** (**VehDelay**) have been classified based on the color scheme **Relative lost time**.
- Width of turn values: Used to visualize the values of the attribute **Vehicles** (**Veh**).
- The number at the beginning of a turn value shows the attribute value **Vehicles** (**Veh**) that Vissim determines at the beginning of a turn relation in the node. Accordingly, the number at the end shows the attribute value that Vissim determines at the end of the turn relation in the node. Vissim automatically shows or hides the figures depending on the selected enlargement of the network in the Network editor.

6.18.1 Configuring turn value visualization

1. On the Network object toolbar, next to **Nodes**, click the **Edit graphic parameters** button (see "Editing graphic parameters for network objects" on page 158).
2. Select the option **Turn value visualization**.
3. Next to **Turn value visualization**, click the symbol.

*The **Turn value visualization** window opens.*



4. Make the desired changes:

Element	Description
Base color	Line color of turn relations in the node, if for the node, Show turn value visualization is selected, there are movements, but no values available yet that could be visualized as turn values. This may for instance be the case, if no vehicle in the simulation has reached the end of turn relation yet.
Decimals	Indicates the numeric attribute values, with the selected number of decimal places.
Show sum	Displays the total of values of the attribute selected for **Classification by width** at the beginning and end of each movement. For Vissim to be able to show numeric attribute values and sums, several elements of the program interface must be set large enough (see "Visualizing turn values" on page 685).
Classification by width	**Attribute** : The width of the turn value bar and the numeric attribute values displayed at the beginning and end of it are based on the values of the attribute selected and the following settings: **Width scale**: - **Automatic**: Vissim specifies the width. - **Manually**: For the width, you can enter a minimum value **Scale (minimum):** and a maximum value **Scale (Maximum)**. If the value of the attribute is smaller than the specified minimum value, the turn value bar is not drawn. If the value of the attribute is greater than the spe-

	cified maximum value, the turn value bar is drawn with its maximum width. ➤ **Turn value bar width (maximum):**:Maximum width for turn value bars with automatically adjusted width. If the message **Visualization is not possible because of overlapping.** is displayed, the turn value bars take up too much space to be displayed next to each other. Reduce the width of turn value bar.
Classification by color	**Attribute** : The color of the turn value bars is based on the values of the selected attribute and the following settings: **Class bounds and colors**: ➤ **Lower bound** column: Value that represents the lower bound of the selected attribute within this value range. Based on value range after MIN, on upper bound of the row above. ➤ **Upper bound** column: Value that represents the upper bound of the selected attribute within this value range. The upper bound belongs the value range. **Range scale factor**: ➤ Enter a factor for the upper bound and lower bound ➤ **Apply** button: Multiply values of upper bounds and lower bounds by a factor : Select a pre-defined color scheme that contains class bounds and colors. The color schemes vary in color and class bound: ➤ **Red-yellow-green**: 11 classes, class size by default 0.500, 11 colors ➤ **Speed**: 11 classes, 11 colors from pink to red, yellow, green to MAX = white ➤ **Density**: 11 classes, 11 colors from white to light blue, blue to MAX = white ➤ **Volume**: 11 classes, 11 colors from white to light blue, blue to MAX = white ➤ **Relative delay**: 11 classes, 11 colors from white to light blue, blue to MAX = white ➤ **Level-of-service value**: six classes, six semi-transparent colors from blue to green and yellow to MAX = red

5. Confirm with **OK**.

To visualize turn values, first make the following settings:

- Activate turn value visualization for the node of your choice (see "Activate turn value visualization" on page 690).
- For the **Nodes** evaluation, select **Collect data** (see "Evaluating nodes" on page 1057).
- For each node whose turn values you want to visualize, select the **Use for evaluation** attribute (see "Attributes of nodes" on page 709).

- The simulation parameters are set and the simulation can be started (see "Defining simulation parameters" on page 840), (see "Selecting the number of simulation runs and starting simulation" on page 845).

6.18.2 Activate turn value visualization

1. Repeat the following steps for all nodes for which you want to visualize turn values
2. In the Network editor, right-click the node.
3. From the shortcut menu, choose **Activate turn value visualization**.

Tip: Alternatively, in the **Nodes** list, click the attribute **Show turn value visualization** (**ShowTurnValVisual**) (see "Attributes of nodes" on page 709).

To visualize turn values, first select all the desired nodes. Then make the following settings:

- For the **Nodes** evaluation, select **Collect data** (see "Evaluating nodes" on page 1057).
- For each node whose turn values you want to visualize, select the **Use for evaluation** attribute (see "Attributes of nodes" on page 709).
- Turn value visualization must be configured according to your requirements (see "Configuring turn value visualization" on page 687).
- The simulation parameters are set and the simulation can be started (see "Defining simulation parameters" on page 840), (see "Selecting the number of simulation runs and starting simulation" on page 845).

6.18.3 Editing the size of turn value visualization for a node

1. On the Network objects toolbar, select the network object type **Node**.
2. In the Network editor, click the node whose turn value visualization size you want to change.

   The polyline of the node turns yellow. The corner points are displayed.
3. Point the mouse pointer to the blue circle line of turn value visualization.

   The mouse pointer changes to a double arrow.
4. Click the blue circle line and hold down the mouse button.
5. While holding down the mouse button, drag the mouse in the desired direction.
6. Once the desired size is reached, release the mouse button.

You can use the size of the current turn value visualization for all other turn value visualizations (see "Setting active turn value diagrams to the same size" on page 690).

6.18.4 Setting active turn value diagrams to the same size

You can adopt the size of one turn value diagram of a node for all other turn value diagrams displayed.

1. In the Network editor, right-click the node.

2. From the shortcut menu, choose **Set all active turn value diagrams to this size**.

7 Using the dynamic assignment add-on module

Note: You must have a license for the add-on module.

When using dynamic assignment traffic demand is not specified by using vehicle inputs on selected links with a given traffic volume but in the form of one or more origin-destination matrix/matrices. Thus you can simulate a road network without having to manually create routes and vehicle inputs. In origin-destination matrices, you specify the starting and end points of trips and the number of trips between these locations. Thus the dimension of an origin-destination matrix is the squared number of zones.

In Vissim, the dynamic assignment is done by an iterated application of the traffic flow simulation.

Dynamic assignment glossary

The following terms are used in relation to the dynamic assignment:

- **Routes**: manually entered static link sequences
- **Paths**: edge sequences computed using dynamic assignment
- **Costs**: the exact financial costs. This is the component of the general cost that is neither travel time nor travel distance. The term "costs" is also used for general costs if the context precludes any chance of confusion.
- **General cost**: is the weighted sum of travel time, travel distance and financial cost. The general cost serves as a basis for the utility function of path selection in the decision model.
- **Travel time**: the average time the vehicles needs for passing a path or an edge in the current simulation.
- **Smoothed travel time**: is computed by exponential smoothing of the travel times measured in the course of iterations. The smoothed travel time is the one that is used in the general cost function.
- **Expected travel time**: is used if we want to express the difference between the travel time that is actually measured in the simulation run and the travel time that is expected in the path selection decision.

During dynamic assignment, you can use microscopic or mesoscopic simulation to simulate the entire network (see "Using add-on module for mesoscopic simulation" on page 801). If you choose to use dynamic assignment with mesoscopic simulation, you can also simulate one or multiple sections of your Vissim network microscopically (see "Hybrid simulation" on page 837).

Dynamic assignment takes the toll pricing calculation into account.

Examples

- You will find a simple use case of dynamic assignment in the *3 Paths.inpx* file, in the directory:

  ..\Examples Training\Dynamic Assignment\3 Paths

- You will find a simple use case of dynamic assignment with a detour in the *detour.inp* file, in the directory:

  ..\Examples Training\Dynamic Assignment\Detour

- You will find a simple use case of dynamic assignment for a drop-off zone at an airport terminal or a railway station in the file *Drop-off zone.inpx*, in the directory:

  ..\Examples Training\Dynamic Assignment\Drop-off Zone.Trip Chains.inpx

- You will find a use case of dynamic assignment with real parking lots in the file *Parking Search - Real Parking Spaces.inpx*, in the directory:

  ..\Examples Training\Dynamic Assignment\Parking Search - Real Parking Spaces.inpx

7.1 Quick start dynamic assignment

The Quick Start- contains the most important steps and settings required for dynamic assignment, after you have modeled the Vissimnetwork. The principle of dynamic assignment is illustrated in a flow chart (see "Flow diagram dynamic assignment" on page 696).

1. Define the nodes required for dynamic assignment (see "Modeling nodes" on page 705), (see "Defining nodes" on page 708):
   - in the areas whose dynamic assignment data you want to evaluate or whose turn values you want to visualize
   - at the beginning and end of edges on which there are parking lots for zone connectors
2. Ensure that for these nodes, the attribute **Use for dynamic assignment** is selected.
3. Define the zones between which you want the vehicles to drive in the Vissimnetwork (see "Defining zones" on page 704).

   The number of zones defines the dimensions of the OD matrix.
4. Define the OD matrix (see "Modeling traffic demand with origin-destination matrices" on page 721), (see "Defining an origin-destination matrix" on page 722).
5. For the OD matrix, select the attributes **Time from** and **Time to**.
6. In the matrix edtior, for the OD matrix, enter the volumes for each zone (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).
7. In your Vissim network, define the parking lots for zone connectors.

   Vehicles drive from and to the these parking lots (see "Defining parking lots for dynamic assignment" on page 700).

8. Assign each of these parking lots a zone of your choice (see "Defining parking lots for dynamic assignment" on page 700).
9. In the parameters of dynamic assignment, select the desired matrix or matrices (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
10. Generate the node-edge graph (see "Generating a node-edge graph" on page 718).
11. If desired, configure evaluations (see "Performing evaluations" on page 1001), (see "Overview of evaluations" on page 1002).
12. If desired, visualize turn values (see "Visualizing turn values" on page 685).
13. Ensure that the desired simulation parameters have been set (see "Defining simulation parameters" on page 840).
14. Start the simulation (see "Selecting the number of simulation runs and starting simulation" on page 845).

7.2 Differences between static and dynamic assignment

In the static assignment, the vehicles follow routes in the road network which you have manually defined. Therefore, the drivers in the simulation have no choice which path to follow from their start point to their destination. For a lot of traffic flow simulation applications this is an appropriate way of modeling.

When the simulated road network grows, there are usually several options the drivers can choose to go from one point in the road network to another. The simulated traffic must be realistically distributed among these alternatives. Using the traffic assignment a given traffic demand is distributed among the various paths in the road network. Traffic assignment is one of the basic tasks in the transport planning process. It is essentially a path selection model of transport users, for example drivers of motorized and non-motorized vehicles.

For such a model, first a set of possible paths is determined. These alternatives must be assessed appropriately. A representation follows on how the drivers decide on the basis of this assessment. This path selection decision model is a special case of the general problem of decision based on discrete alternatives (discrete choice). A lot of theory behind traffic assignment models originates from the discrete decision theory.

The most common assignment processes in transport planning belong to the class of static assignments. *Static* means that neither the traffic demand, indicating how many trips should be made in the network, nor the road network change. This does not correspond to reality. The traffic demand can vary significantly during the day. The road network can have time-dependent characteristics, such as when different signal programs run throughout the day at the signalized nodes and thus create time-dependent capacities for the individual flows. Dynamic assignment takes these temporal fluctuations into account.

The motivations to model the path selection in a Vissim simulation model:

- With the increasing size of the simulated road network, it will become more and more difficult to enter all paths from sources to destinations by hand, even if no alternative paths are considered.

- The path selection behavior can itself be the subject of your investigation, if the effects of measures are to be judged. This would also affect the path selection.

7.3 Base for calculating the dynamic assignment

The principle of dynamic assignment is illustrated in a flow chart (see "Flow diagram dynamic assignment" on page 696).

The Quick Start- contains the most important steps and settings required for dynamic assignment, after you have modeled the Vissimnetwork (see "Quick start dynamic assignment" on page 693).

The dynamic assignment is calculated in Vissim based on the iterated simulation. Thereby the modeled road network is simulated not only once but repetitively. The drivers choose thereby their paths through the network based on their experiences from the preceding simulations. The modeling of this "learning process" consists of the following subtasks:

- Vissim takes into consideration in searching for the paths from sources to destinations, that not every driver chooses the best path. Some drivers will use less attractive paths. That means that not only the best paths must be known for each origin-destination relation but also a set of paths. Ideally, the number of the k best paths would be used. Since there is no efficient method for direct calculation of this quantity, which would be useful for traffic assignment, the following approach is used in Vissim:

  In each iteration of the simulation, the best paths are calculated respectively. More than one optimal path would be found, because traffic conditions change from iteration to iteration. In the course of the iterations the number of different paths and the archive of known paths, from which the drivers can choose, increases (see "Path search and path selection" on page 738).

- For the paths found an evaluation must be calculated, on which the drivers base their choice. In Vissim, the so-called generalized costs are computed for the paths. These comprise travel time, travel distance and other costs (for example, tolls) are calculated. Travel distance and costs are defined directly in the network model whereas the travel time can be determined only with the help of the simulation. Therefore travel times will be measured in individual parts of the network during a simulation run in Vissim, and can be considered for the path selection in subsequent simulations.

- The selection of a path from a set of possible paths is a special case of the discrete decision problem (discrete choice problem). From the set of paths and their generalized costs, the percentage of drivers that choose the path is calculated. The mathematical function which is by far the most commonly used to represent this type of selection is the Logit function. Also Vissim uses a variant of the Logit model for the path selection (see "Method of path selection with or without path search" on page 741).

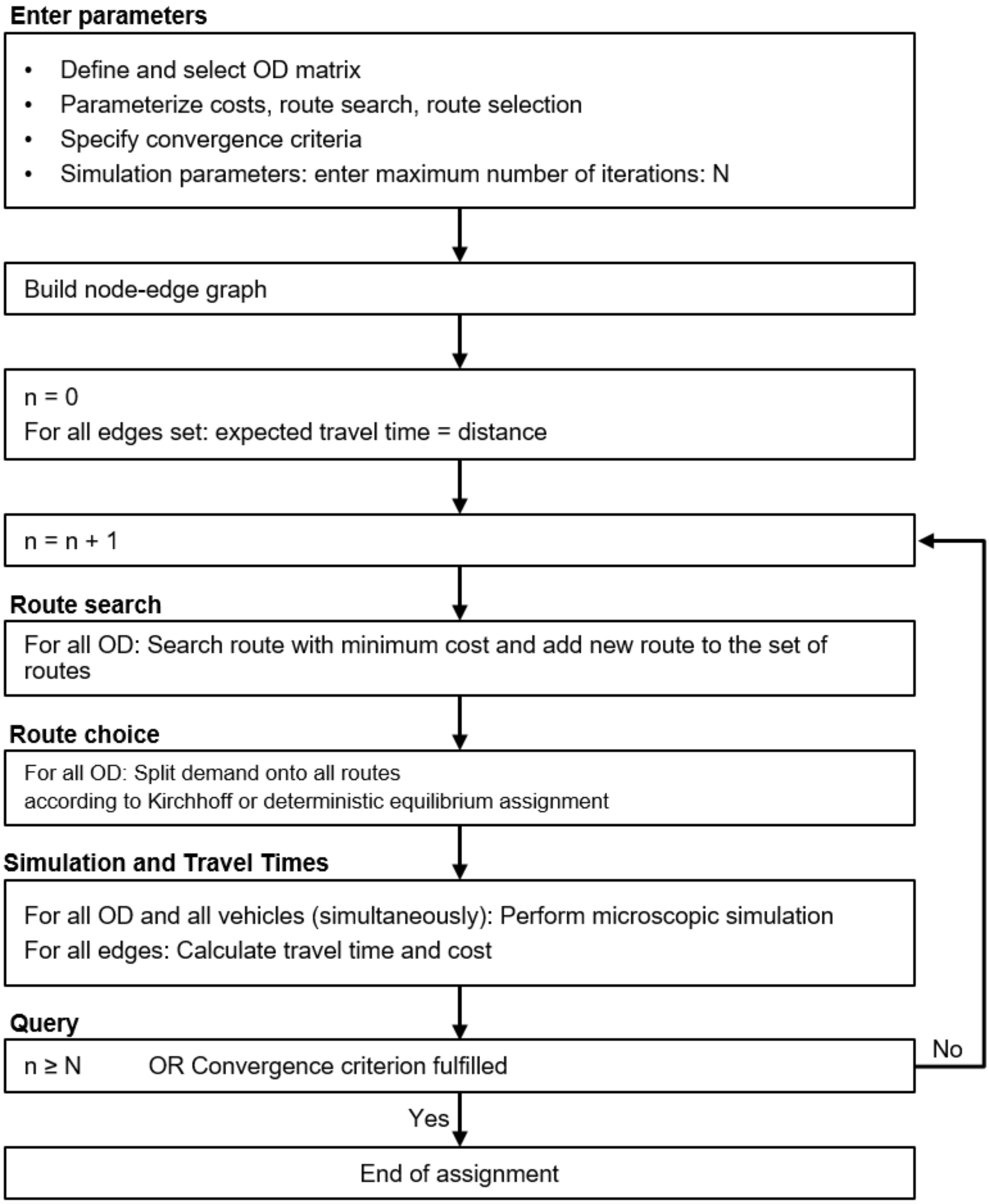
The road network is modeled in Vissim in great detail to provide a reproduction of the traffic flow as precisely as possible, in high temporal and spatial resolution. However, this detailed modeling is not necessary for any of the three subtasks listed above. For example, the decision which path through a city is chosen does not depend on which lanes the vehicles travel, or how the junctions on the path look like exactly. For the assignment it is enough to reference an abstract description of the road network, where the junctions are nodes and the links between the junctions are edges of an abstract network graph. On this abstract network graph, the assignment procedures can work much more efficiently. The abstract network graph also correlates with the human understanding: For example, to describe a path to someone, it is sufficient to mention the sequence of junctions and to add if he must turn there; a detailed description is not necessary.

In Vissim an abstract network is built for dynamic assignment. To do so, in the detailed model, highlight the parts you want to use as abstract nodes by drawing in network objects of the type **node**. For dynamic assignment with microsimulation, these will normally be network sections that correspond to real intersections (see "Building an Abstract Network Graph" on page 697).

The simulation is iterated until there are no more significant changes from one iteration to the next in the congestions and travel times in the network. This situation is called in Vissim as convergence. You can set the criteria for convergence (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

7.4 Flow diagram dynamic assignment

The following flow diagram illustrates the principle of the dynamic assignment:



7.5 Building an Abstract Network Graph

The dynamic assignment creates an abstract network graph as a basis for the calculation. An abstract network graph links the network objects relevant for the dynamic assignment. This includes the parking lots, from which the vehicles enter the network, or which are the destination of the vehicles, as well as nodes and edges. When you start dynamic assignment, Vissim generates an abstract network graph based on these network objects. When you perform dynamic assignment with mesoscopic simulation, Vissim additionally generates a meso-specific network graph (see "Mesoscopic node-edge model" on page 804).

7.5.1 Modeling parking lots and zones

In the dynamic assignment the start and destination points of the vehicles must be located in parking lots. One parking lot is always assigned to one zone only. Trips originating from this zone or ending in this zone can begin or end at this parking lot. A zone can have more than one parking lot. You define the originating traffic with a distribution to the desired parking lots. The distribution of destination traffic across multiple parking lots of a zone is computed by a parking lot choice model (see "Defining the destination parking lot selection" on page 756). As the start and destination points of vehicles lie on parking lots that are allocated to zones, Vissim distinguishes between origin and destination zones. You can show the relation between an origin zone and a destination zone in the **OD Pairs** list (see "Attributes of OD pairs" on page 705). Result attributes that are created between origin and destination zones during a simulation run can be entered in the evaluation **OD pairs** and displayed in the list **OD Pair Results** (see "Displaying OD pair data in lists" on page 1027).

If you want vehicles to drive from a parking lot of a certain district to another parking lot of the same district and Vissim finds a path, the simulation includes the district's entire traffic for this OD relation. If there is no other parking lot the same district, vehicles do not drive back to their start parking lot.

Traffic starting at a parking lot behaves similarly to the traffic generated by vehicle inputs (see "Modeling vehicle inputs for private transportation" on page 454). However, the vehicle composition is not specified at the parking lot, but arises from the vehicle compositions of the source-destination matrices that produce traffic in this parking lot.

The desired vehicles speeds are not taken from the distribution specified with the vehicle composition in the matrix, but from the default desired speed distributions which you choose from the parking lot attributes. In addition, you can assign individual vehicle classes a defined desired speed distribution.

With desired speeds at parking lots you can take into account different speed limits in the parking lot area.

Vissim uses vehicles in the network from simulation start. This allows you to assign a vehicle a path via the COM interface when it leaves its origin parking lot.

The number of vehicles inserted into an origin parking lot corresponds to the value listed in the origin-destination matrix. The origin parking lot is chosen based on whether the **Use volume (old)** attribute on the **Choice** tab of the parameters for dynamic assignment is selected:

- If the **Use volume (old)** attribute is deselected, the origin parking space is chosen based on the **Rel. volume** attribute of the parking lots in the zone where the vehicle starts.

- If the **Use volume (old)** attribute is selected, the origin parking lot is chosen based on the volume specified in the **.weg* path file. The **Rel. volume** attribute is then not taken into account. The time intervals of path volumes may differ from the time window defined for the origin-destination matrix in the **From-time** and **To-time** attributes. This is why path volumes adopted from the **.weg* path file are scaled using the part of the time interval that lies within the valid period of the origin-destination matrix. These scaled path volumes are added up for all time intervals and paths leading to the destination zone of the OD pair. This equals the relative volume of a parking lot. The total of relative volumes for all parking lots of the origin zone of an OD pair is the normalization factor required to calculate the probability of selection of a parking lot. If the total of probabilities equals 0, then no parking lot is selected and no vehicles are used in this zone.

  The relative volume for a parking space is 0 in the following cases:

  - When there is no path between the parking lot and a parking lot in the destination zone
  - When the capacity of the parking lot is exhausted

When the origin parking lot is of the type **Real parking spaces** and the capacity of the parking lot has been reached, the vehicle is not be added to the network.

7.5.1.1 Using parking lot types

You can use the following parking lot types in the dynamic assignment. Parking lot types differ in the behavior of the vehicle when driving in and driving out.

- **Zone connector**: Vehicles drive in without delay and are taken out of the simulation in the middle of the parking lot without stopping. The entry capacity is non-limited and corresponds to the capacity of the links to be driven. Use the zone connector in order to model the vehicle origins (origin parking lots) and destinations (destination parking lots) in the network. This is often useful on the edges of the modeled networks. Do not map real parking lots with zone connectors.
- **Abstract parking lot**: A vehicle decelerates when it approaches a parking lot and comes to a stop in the middle of the parking lot. It is then parked and is deleted. The next vehicle can drive in. Use abstract parking lots if the network model is detailed enough to model many realistic parking lots together, e.g. in an underground garage or parking garage. Due to the modeling of stoppage processes when driving in, the entry capacity of an abstract parking lot is limited to approximately 700 vehicles per hour and lane.
- **Real parking spaces**: For real parking spaces, Vissim by default creates a parking routing decision 50 m before the parking lot. If the graphic parameter **Object visibility** is selected for vehicle routes of the type **Parking lot**, the parking routing decision is displayed in the Network editor during the simulation. If the **Parking routing decision** list is opened during a simulation run, it lists the parking routing decision.

  The calculated routes of the dynamic assignment may also go via parking lots of the **Real parking spaces** type.

7.5.1.2 Defining parking lots for dynamic assignment

You can define different parking lot types for dynamic assignment (see "Using parking lot types" on page 699).

Conditions for placing parking lots and calculating paths

Parking lots must always be placed on an edge between two nodes or within a node.

Parking lots whose entire length does not lie on an edge cannot use the same edge as a start edge for departing vehicles or a destination edge for arriving vehicles.

An edge or turn relation might run via a link with adjacent, real parking spaces on several lanes. Within a node, parking lots may be located on several turn relations. Vissim can calculate paths based on these edges.

A route cannot lead via an edge with a parking lot that vehicles cannot drive past. This is the case in the following situations:

- if on an edge, there is a zone connector or abstract parking lot
- if on the edge, on a link with only one lane, there is a parking lot with real parking spaces

The origin parking lot and/or destination parking lie on such an edge.

No additional parking lot may lie on an edge with a zone connector, or on an abstract parking lot, or on a parking lot with real parking spaces on a link with only one lane.

If a parking lot is only placed on a link for destination traffic, which leads out of the network, so that from this link no other parking lot can be reached, the relative volume for this parking lot must be set to 0. Even these parking lots must be located between two nodes.

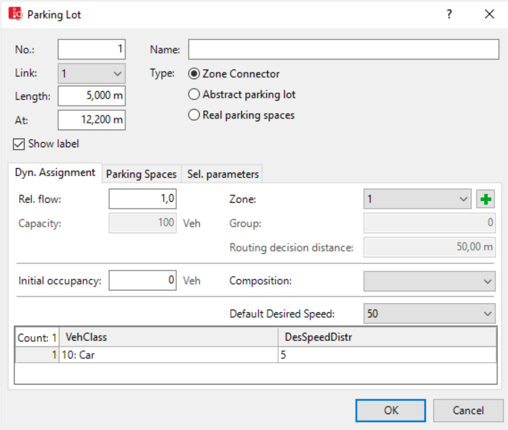
The costs of an edge, on which a parking lot is located, is determined as the average of the costs of all vehicles, which drive in and out of the parking lot.

When the parking space decision is traversed and all destination parking lots only contain blocked parking spaces, the most attractive blocked parking space is chosen, if it is not occupied.

When a vehicle is on a route, it can only choose a parking space at a parking space routing decision that allows it to continue its route downstream of the parking space decision. Otherwise the parking space routing decision is ignored. If a vehicle is not on a route, it will generally choose a parking space when one is available.

1. Define the parking lot of your choice (see "Defining parking lots" on page 499).

   *The **Parking Lot** window opens.*



2. Select the tab **Dyn. Assignment**.
3. If the parking lot is meant to serve as an origin parking lot or destination parking lot with a zone connector, as **Type**, select **Zone Connector**.
4. Make the desired changes:

Element	Description
Rel. flow	**RelFlow**: only relevant in the following cases: - When a zone has multiple parking lots: share at emergence of the zone. - When in the **Dynamic Assignment: Parameters** window, on the **Choice** tab, the **Use volume (old)** attribute is deselected (see "Attributes for path selection" on page 779) Example value 0 = no originating traffic for a parking lot on a link, which departs the network. For multiple parking lots, the sum of the value of a zone = 100 %. From this, the percentage share per zone is determined. You can set the relative volumes of all parking lots to the volume totals (see "Setting relative volumes from all parking lots on the volume totals" on page 703)
Capacity	Only relevant for parking lots with the **Abstract parking lot** attribute: Maximum number of vehicles. For **Real parking spaces** the value is yielded from the length of the parking lot and the length per parking space. The capacity cannot be changed.

Element	Description
Zone	Zone number from origin-destination matrix for the parking lot. Multiple parking lots can belong to a zone. The number of zones defines the dimensions of the OD matrix (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724). : Open **Add zone** window and define a new zone (see "Defining zones" on page 704): - **Number**: Number of the zone. Vissim shows the next number available. - **Name** : Desired name of the zone
Group	Only relevant for parking lots with the **Real parking spaces** attribute: number of the parking lot group to which the parking lot shall be allocated. Without the number, the parking lot does not belong to a group. An allocated parking lot is allocated to all parking space decisions of a parking lot group. Thereby, the parking space decisions are not located any further from the allocated parking lot than the total of the distance to the parking space decision and the maximum distance allowed for grouping parking space decisions (by default *50 m + 50 m*). For the parking lots of a group, the automatically generated parking space decisions are automatically combined, if they are distanced less than 50 m from each other. If a vehicle's destination is a parking lot of the group, it can select any parking lot in the group. Accordingly, the **criterion** for which a routing decision from type **Dynamic** selected also applies to all parking lots of the group to which the parking lot belongs to, for example, **parking lot full =** All parking lots of the group are occupied. Parking space decisions are automatically generated at the start of the simulation (see "Using parking lot types" on page 699). Parking space decisions cannot be displayed in lists or edited.
Routing decision distance	**RoutDecDist**: Distance of the parking space decision to the parking lot
Initial occupancy	**InitOccup**: only relevant for **Abstract parking lots** and **Real parking spaces**. Initial occupancy in vehicles specifies which occupancy of the parking lot should be calculated at the beginning of the simulation for the destination parking lot selection. This also applies for dynamic routing decisions. The initial occupancy serves to determine the time when the capacity of the parking lot has been reached. Do not enter the number of vehicles which arrive at the parking lot and then later drive away during a simulation. These can be included in the origin-destination matrices. For **Real parking spaces**, enter the composition of the initial occupancy.

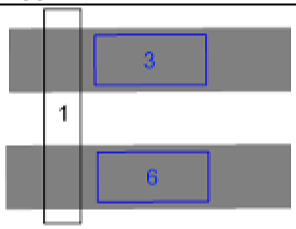
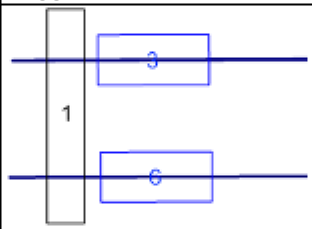
Element	Description
Composition	**InitOccupComp**: only relevant for **Real parking spaces**: Choose the appropriate vehicle composition for the **Initial occupancy** (see "Modeling vehicle compositions" on page 452).
Default Desired Speed	**Desired speed distribution default** (**DesSpeedDistrDef**): allocated desired speed distribution and desired speed per vehicle class. The default-desired speed distribution and desired speed are used for all vehicles whose type does not belong to any of the vehicle classes displayed in the list below.

5. Confirm with **OK**.

*The attributes are saved in the list of **Parking Lots** (see "Attributes of parking lots" on page 500).*

Example for the modeling of a zone connector on the edge of a network

The figure shows a zone connector of the example file *..\dynamic assignment\Drop-Off zone.Trip Chains\Drop-Off Zone inpx*:

Toggle wireframe off	**Toggle wireframe on**
	
➤ The node is placed at the beginning of the upper link at the end of the lower link. The beginning and end could also lie in the node. ➤ Origin parking lot **3** and destination parking lot **6** are placed outside of the node.	

Setting relative volumes from all parking lots on the volume totals

If you have exported a network from Visum and are using a different volume scenario with a new matrix file and a new path file, the relative volumes of the output parking lot are more suitable.

- Press the key combination CTRL+SHIFT+C.

  *The relative volumes of all of the parking lots are set to the volume totals of their paths in the current path file *.weg.*

Avoiding errors when modeling parking lots

An error message such as `The origin parking lot 1 is part of several different edges` can mean that at least one node is missing or not positioned correctly. Thereby multiple paths can be found between both nodes between which the parking lot is located. For each of these paths, separate costs are determined. This can lead to different link costs for a link sequence (in reality, a street) and therefore cause an incorrect vehicle distribution.

In order to avoid this problem, when positioning the parking lots in the Vissim network, ensure the following:

- In the movement direction, the starting point of the path must be located before the parking lot part of the node.
- In the movement direction, the end point of the path must be located behind the parking lot part of the node.
- Both of these nodes, between which the parking lot is located, must be correctly modeled and the beginning and end points of the path must be located between two nodes.

7.5.1.3 Defining zones

1. From the **Lists** menu, choose > **Network** > **Zones**.

   The list of defined network objects for the network object type opens.

   By default, you can edit the list (see "Using lists" on page 93).

   In the list, you can define a new zone.

2. Right-click in the list.

3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

4. Enter the desired data.

Element	Description
No	Unique identification number of the zone
Name	Name of zone

You can assign the zone to a parking lot (see "Defining parking lots for dynamic assignment" on page 700).

Tip: Alternatively, add a zone in the **Parking lot** window > **Dyn. Assignment** tab > **Zone** box (see "Defining parking lots for dynamic assignment" on page 700).

7.5.1.4 Attributes of zones

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Zones**.

   *The **Zones** list opens.*

The list on the left may include the following attributes:

Element	Description
No	Unique number
Name	Name of zone
Center	Location of a zone calculated from the mean of the coordinates of zone parking lots
ParkLot	**Parking lot**: Numbers of the zone parking lots

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- OD pairs (see "Attributes of OD pairs" on page 705)
- Parking lots: (see "Attributes of parking lots" on page 500)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

7.5.1.5 Attributes of OD pairs

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **OD pairs**.

   *The **OD Pairs** list opens. All relations between origin zones and destination zones are displayed. This also includes relations within a zone.*

The list on the left may include the following attributes:

Element	Description
OrigZone	**Origin zone**: Number and name
DestZone	**Destination zone**: Number and name

7.5.2 Modeling nodes

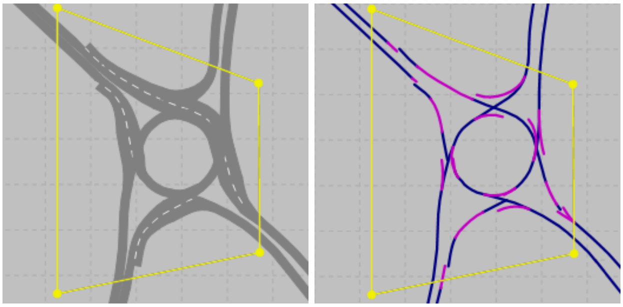
The geometry of the road network is modeled in a very detailed manner in Vissim. This exactness is not necessary for the decision of a driver for a specific path through the network. The exact traffic routing at the node is not relevant; instead, the directions on the nodes which can be turned are relevant.

In order to reduce the complexity of the network model and therefore also the calculation time and memory required, you can identify parts of the network as nodes. These positions are at the minimum the positions in which the paths merge together, or the positions which branch out in different directions. Normally these are the network sections which represent a real junction. Do not group larger network sections, containing multiple intersections, into a node.

Nodes for evaluations, dynamic assignment and mesoscopic simulation

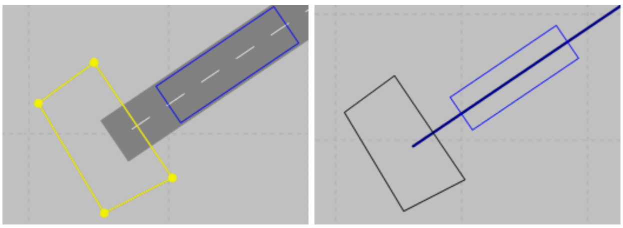
In the attributes of the node, you select whether you want to use the node for evaluations and/or dynamic assignment and/or mesoscopic simulation. Depending on the particularities of the network, a node can be used for evaluations, dynamic assignment and mesoscopic simulation. However, certain particularities of a network might require you to model additional nodes for mesoscopic simulation (see "Mesoscopic node-edge model" on page 804).

To perform dynamic assignment, you only need one node for each roundabout or complex intersection. For these nodes, select the attribute **Use for dynamic assignment**. It is not necessary to define a separate node for each conflict of two movements. Example file *..\Examples Demo\Roundabout Schenectady.US\Roundabout Schenectady.inpx*:



Nodes at the boundary of a network

For dynamic assignment, nodes are required at the boundaries of the network where links in the Network editor begin or end. Example file *..\Examples Training\Dynamic Assignment \Detour\Detour.inpx*:



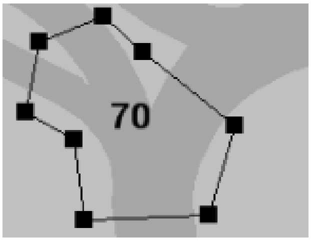
Notes:

- Meso network nodes may intersect with nodes of dynamic assignment.
- Nodes of dynamic assignment must not intersect with each other.
- Meso network nodes must not intersect with each other.
- If you open a network file **.inpx* with an overlapping node or you add a node which overlaps the adjacent node, an error message appears. The error message and the numbers of the overlapping nodes are shown in the **Messages** window (see "Showing messages and warnings" on page 1178).
- Mesoscopic simulation distinguishes between different node types (see "Mesoscopic node-edge model" on page 804).

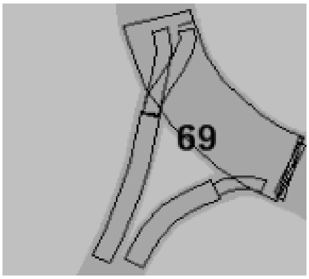
7.5.2.1 Using polygon nodes and segment nodes

Vissim distinguishes between polygon nodes and segment nodes:

Polygon nodes: in figure, node 70. By default, polygon nodes are used in Vissim because they can be easily defined and edited. You always add a node as a polygon node in Vissim (see "Defining nodes" on page 708).



Segment nodes consist of link segments, as shown in figure node 69.



A polygon node can be converted into a segment node and vice versa (see "Converting polygon nodes" on page 715), (see "Converting segment nodes" on page 716).

During the ANM import of external data, nodes in Vissim are always generated as link segments (see "Importing ANM data" on page 366).

The definition of nodes as link segments allows a more detailed editing of the node (see "Selecting nodes, polygons or segments" on page 714).

Examples:

- If you have defined a node manually and in a node polygon, a fast road e.g. continues as a bridge above the node, you can convert the polygon node into segments and remove all segments on the fast road from the node so that the fast road is no longer part of the node.
- If you have imported a node as a link segment, you can convert it to a polygon, in order to change its spatial extent at the polygon level to the segment level.

Notes:

- Attributes of the polygon nodes and segment nodes are identical.
- The colors from polygon nodes and segment nodes are identical.
- In the dynamic assignment, the polygon nodes and segment nodes are considered in the same manner.

7.5.2.2 Defining nodes

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Nodes**.
2. Hold down the CTRL key and right click on the network editor on the desired corners of the polygons in the area of the node.

Note: Alternatively you can select **Add New Node** from the context menu in the network editor. A node with four corners is added. The **Nodes** window opens, if you have selected that you want the program to open the Edit dialog automatically after object creation (see "Right-click behavior and action after creating an object" on page 152).

---

3. Once you have added all of the corners of the polygon, double click.
4. Release the keys.

   *The node has been added. The window **Node** opens.*
5. Edit the attributes (see "Attributes of nodes" on page 709).
6. Confirm with **OK**.

*The attributes are saved in the list **Nodes**.*

Notes:

- Do not define overlapping nodes.
- A polygon node can be converted into a segment node and vice versa (see "Converting polygon nodes" on page 715), (see "Converting segment nodes" on page 716). For each conversion the definition of the node is adjusted correspondingly. You can edit the node definition.

Tip: Alternatively, you can import nodes via ANM import (see "Importing ANM data" on page 366). Nodes are always imported as segment nodes.

7.5.2.3 Attributes of nodes

The **Nodes** window opens when you insert a network object and have selected to have the Edit dialog automatically opened after object creation (see "Right-click behavior and action after creating an object" on page 152). By default, only the Nodes list is opened.

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

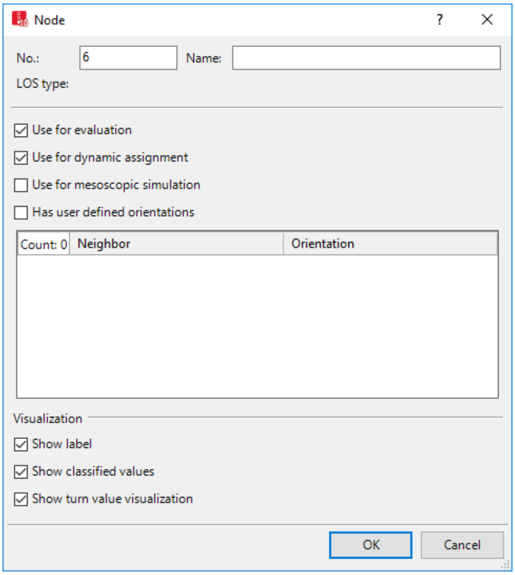
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



1. Make the desired changes:

Element	Description
No.	Unique node number
Name	Designation of the node
LOSType	**Level-of-service scheme type**: Basis for determining the appropriate LOS scheme for result attributes **LOS(All)** and **LOSVal(All)** in node evaluation (see "Evaluating nodes" on page 1057). The **LOSTyp** is defined Vissim based on the node type when the simulation run is first started: - **Signalized**: If at least one signal head of an active SC has been defined for at least one movement - **Non-signalized**: If no signal head has been defined or the SC is inactive The **LOSType** is only calculated in the following cases: - For nodes, whose **Use for evaluation** attribute is selected - For movements with the direction **Total** - For evaluation edges that are entry edges into the node When you edit a node or insert a new node, Vissim defines the **LOSTyp** for all nodes based on the node type the next time you start a simulation.
Showing label	If this option is not selected, the label for the respective node is hidden when label for all nodes is selected.

Element	Description
Dynamic assignment	**Use for dynamic assignment** (**UseForDynAssign**): ☑ If the option is selected, the nodes for the network graphs are taken into consideration in the dynamic assignment. When starting a simulation from COM, the node-edge graph is always constructed because it is not possible to determine at the start of the simulation whether the curvature of the simulation should add another path to the vehicle, which originates at the COM script. Even without COM the construction of the node-edge graph takes time if you call up the list **Edges** because there are many edges between two nodes, particularly for low node density. In the list **Edges**, only the nodes are shown in which the option **Dyn. Assignment** is selected (see "Editing edges" on page 717). ☐ If the option is not selected, the nodes for the network graphs are ignored in the dynamic assignment. If you select the attribute **Use for evaluation**, the node can still be considered for node evaluation (see "Evaluating nodes" on page 1057). If the nodes are not used for the dynamic assignment, deactivate the option. This saves you calculation time.
Use for mesoscopic simulation	**UseForMeso**: ☑ If this option is selected, the meso network node is taken into account for generation of the meso graph (see "Modeling meso network nodes" on page 809). ☐ If this option is not selected, the node in not taken into account for mesoscopic simulation. Note: This setting is ignored for the microscopically simulated sections during hybrid simulation.
User defined orientations	**Has user defined orientations** (**HasUserDefOrient**): ☑ If this option is selected, the **Orientations** list is enabled. The list displays the determined orientation of the intersection between the polygon of the node and the edge to the adjacent node. You can overwrite these in the list when they do not correspond with the real orientation. - Column **Neighbor**: List of nodes adjacent to the edited node in the network. - Column **Orientation**: Select a direction for this adjacent node: N, NE, E, SE, S, SW, W, NW Select a direction in particular when multiple edges lead to an adjacent node and Vissim has determined an unrealistic direction. By default, in the case of multiple edges, Vissim uses the direction which occurs the most often; in the case of only two edges, it uses the direction which, in the list of available directions, occurs first (at the top, in clockwise direction).

Element	Description
Note: The evaluation graph for the calculation of orientations based on the current direction **North** in the network. The evaluation graph only considers the nodes which fulfill the following conditions: For both adjacent nodes, you must select **Use for evaluation**. If this is not the case for all network nodes, the graph is not complete. The adjacent nodes cannot be more than 500 m from each other. In the case of larger distances, an additional node can be inserted.	
Use for evaluation	**UseForEval**: ☑ If the option is selected, the node is considered for the node evaluation, if the node evaluation is selected (see "Evaluating nodes" on page 1057).
Show classified values	**Show classified values** (**ShowClsfValues**): ☑ Select this option to show classified values, not to show the display type selected. To show classified values, in the graphic parameters for nodes, select a color scheme and an attribute (see "Assigning a color to nodes based on an attribute" on page 191).
Show turn value visualization	(**ShowTurnValVisual**: ☑ Select this option to graphically show, in the node, along the turn relations, the values of the attribute selected for turn value visualization (see "Visualizing turn values" on page 685). Only active if the **Use for evaluation** attribute is selected, because only then can movements exist. Tip: Alternatively, in the Network editor, right-click the node. Then from the shortcut menu, choose **Activate turn value visualization**.

2. Confirm with **OK**.

The network object has additional attributes that you can show in the Attributes list. In the Attributes lits, the following is displayed by default:

Element	Description
TurnValVisualSize	**Turn value visualization size**: Radius [m] of the outer circle of the the turn value visualization

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Turns (evaluation): Attributes of edges in the node that are turn relations in the node-edge graph of node evaluation (see "Generating a node-edge graph" on page 718)
- Turns (dynamic assignment): Attributes of edges in the node that are turn relations in the node-edge graph of dynamic assignment (see "Generating a node-edge graph" on page 718)
- User defined orientations
- Movements (see "Evaluating nodes" on page 1057). To show result attributes of movements, you first need to generate the node-edge graph for evaluations (see "Generating a node-edge graph" on page 718).
- Edges (evaluation) (see "Generating a node-edge graph" on page 718), (see "Attributes of edges" on page 718)
- Edges (evaluation, entering): All edges of the evaluation node that enter the node and/or end at the node
- Edges (dynamic assignment) (see "Generating a node-edge graph" on page 718), (see "Attributes of edges" on page 718)
- Conflict areas (see "Attributes of conflict areas" on page 565)
- Meso turn conflicts (see "Attributes of meso turn conflicts" on page 835)
- Meso turns (see "Attributes of meso turns" on page 833)
- Points: edit coordinates of the corners
- Link segments: Attributes of the link segments in segment nodes

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

7.5.2.4 Meaning of node color and line style

In the graphic parameters of the network object type **Nodes**, you can define a fill style, fill color, border line style and border line color for nodes (see "List of graphic parameters for network objects" on page 161).

How visualization is implemented also depends on the following settings and factors:

A node is selected.

The border of the selected polygon node or segment node is a solid yellow line. Filled circles are displayed at the corner points of the polygon node.

Multiple nodes are selected.

The border of each node selected is a solid yellow line. There are no filled circles are displayed at the corner points of the polygon node.

No node is not selected.

The **Drawing mode** attribute of the network object type **Node** defines the color:

- **Use constant colors**: The border of a polygon node or segment node is drawn as a solid line. The color depends on the node type:
  - Polygon node: The **Border color** attribute defines the color of the border.
  - Segment node: The color of the border is dark gray.
- **Color by function**: For polygon nodes and segment nodes, the border and color depend on the node type:

Color and style of the border	node type selected
solid white	no node type
solid green	Use for evaluation
solid red	Use for dynamic assignment
solid black	Use for evaluation and for dynamic assignment
white, dashed	Use for mesoscopic simulation
not white, dashed	Use for mesoscopic and other simulations

7.5.2.5 Selecting nodes, polygons or segments

You can select and edit nodes in a Network editor. For a single node, you can open the **Node** window and edit the attributes. For multiple nodes, you can edit the attributes in the **Nodes** list (see "Attributes of nodes" on page 709). The **Nodes** list contains all nodes, regardless of whether they are currently represented as a polygon or displayed in the segment definition.

You can select and delete single or multiple nodes.

1. On the Network objects toolbar, click **Nodes**.

Note: Do not define overlapping nodes.

2. In the Network Editor, click the network object:

   - in the desired node
   - in the desired polygon
   - on one of the segments
   - in the hidden rectangle around the segments

   The selected node is highlighted *(see "Meaning of node color and line style" on page 713).*

Tip: You can select multiple nodes in a Network Editor by drawing a rectangle or by holding down the CTRL key and clicking the node.

7.5.2.6 Editing node polygons

You can move a node polygon in a network editor and call up different functions for editing via the context menu.

Moving polygon

- Click on the network editor on the desired polygon, hold down the mouse button and move the polygon.

  The courses of the road in the network remain unchanged.

Inserting polygon point

1. On the Network objects toolbar, click **Nodes**.
2. In the Network editor, right-click the desired position of the new polygon point on the line of the polygon.

   The polygon point is displayed.

Moving the polygon point

1. On the Network objects toolbar, click **Nodes**.
2. In the Network editor, click the polygon point, hold down the mouse button and move the polygon point to the desired position.
3. Release the mouse button.

Deleting the polygon point

1. On the Network objects toolbar, click **Nodes**.
2. In the Network editor, click the polygon point, hold down the mouse button and move the polygon point onto an adjacent polygon point.
3. Release the mouse button.

7.5.2.7 Converting polygon nodes

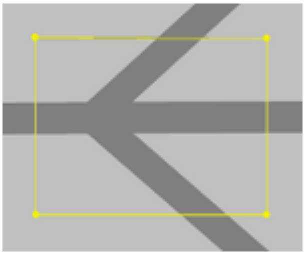
You can convert polygon nodes to segment nodes. When you convert nodes several times, the expansion of the node polygon may increase. Additional segments are added to the display of segments or the size of the segments is adjusted.

Note: Do not define overlapping nodes.

---

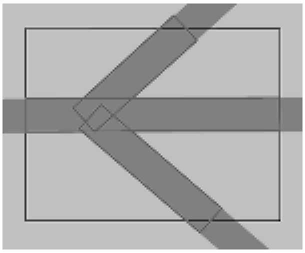
1. On the Network objects toolbar, click **Nodes**.
2. In the Network Editor, right-click the polygon node.

   The polygon node is marked.



3. In the context menu, choose the entry **Convert Node**.

   The segments of the node are shown. The segments are arranged according to the expansion of the original node polygon on the links. When you convert a polygon node to a segment node, only the segments which include the polygon are generated.



7.5.2.8 Converting segment nodes

When you convert a segment node to a polygon node, a rectangular polygon is generated. Its location and size result from the location and size of the segments. The polygon includes all segments. The lower horizontal line is drawn directly under the lowest point of the segments. The right line is drawn immediately to the right of the right-most point of the segments. The segments within this polygon do not necessarily correspond to the original segments.

When you convert nodes several times, the expansion of the node polygon may increase. Additional segments are added to the display of segments or the size of the segments is adjusted.

Note: Do not define overlapping nodes.

---

1. On the Network objects toolbar, click **Nodes**.
2. In the Network Editor, right-click the segment node.

   The segment node is marked.

3. In the context menu, choose the entry **Convert Node**.

   Frame and polygon points of the node are shown. The area of the polygon increases.

7.5.2.9 Creating nodes per level

From a segment node on one level you can create additional segment nodes lying on other levels. This can be useful when you convert a polygon node, with a multi-story parking garage including links and connectors, into a segment node, but you need segment nodes for each level of the parking garage. Vissim will then only generate segment nodes for the levels that had network objects in the the original node.

1. Make sure that in the network editor, you have selected the segment node of your choice (see "Moving network objects in the Network Editor" on page 356).
2. In the Network Editor, right-click the segment node.

   The segment node is marked.
3. From the shortcut menu, choose **Create nodes per level**.

   The segment node is duplicated for each level that has network objects in the segment node. The new segment nodes are listed in the ***Nodes*** *list.*

7.5.2.10 Deleting nodes

You can delete an individual node in a network editor or in the **Nodes** list (see "Deleting network objects" on page 356). You can also delete multiple nodes.

1. On the Network objects toolbar, click **Nodes**.
2. Press the CTRL key and click the nodes.
3. Press the DEL key.

7.5.3 Editing edges

At the start of the dynamic assignment, Vissim automatically generates an abstract network graph based on the user-defined node. You can also generate this node-edge graph via a function (see "Generating a node-edge graph" on page 718). The node-edge graph may consist of the following edges:

- Turn relations: edges within a node
- Edges from node to node. An edge starts at the border of node and ends at the border of a node.

7.5.3.1 Differences from standard network graph

The topology of the node-edge graph is only slightly different from the standard network graph in traffic flow models:

- There may be more than one edge between two Vissim nodes.
- The turn relations within nodes are not just abstract entities, but are represented by edges which have a real length in Vissim.

7.5.3.2 Use of edges

Edges are the elementary components of the path search because paths are sequences of edges. At the edge level, travel times and costs are recorded in the simulation and made available for the path selection in the next iteration.

If for the attribute **Blocked vehicle classes for dynamic assignment**(**Connector closed to**) of a connector, the number of vehicle classes selected results in all vehicle types being closed to the connector, this connector is not used by network graph edges.

7.5.3.3 Generating a node-edge graph

You can create a node-edge graph for nodes:

- For dynamic assignment: The nodes must have been activated for dynamic assignment (see "Attributes of nodes" on page 709).
- For node evaluation and thus for the relation **movements** of nodes (see "Evaluating nodes" on page 1057), (see "Attributes of nodes" on page 709). Vissim automatically generates the nodes-edges graph for evaluations when the **Node evaluation results** list is opened.

1. Select **Network** > **Edges** in the **Lists** menu.

   The list ***Edges*** *opens. If no node-edge graphs were generated, only the column titles are shown.*

2. Right-click in the list.
3. From the shortcut menu, choose the desired entry:
   - **Create Dynamic Assignment Graph**
   - **Create Evaluation Graph**

   The edges and their attributes are shown in the ***Edges*** *list (see "Attributes of edges" on page 718).*

   When you delete network objects that affect edges, Vissim updates the ***Edges*** *list. You then might have to create a new graph.*

7.5.3.4 Attributes of edges

Edges and their attributes are generated with node-edge graphs (see "Generating a node-edge graph" on page 718).

1. Select **Network** > **Edges** in the **Lists** menu.

   The list ***Edges*** *opens. If no node-edge graphs were generated, only the column titles are shown and you must generate node-edge graphs (see "Generating a node-edge graph" on page 718).*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Element	Description
No	Unique edge number
FromNode	**From node**: Name of the node, on which the edge begins
ToNode	**To node**: Name of the node, on which the edge ends
Type	Type of edge is either **Dynamic assignment** or **Evaluation**
IsTurn	**Is turn**: ☑ If this option is selected, the edge is a turn relation.
Closed	**Closed**: ☑ If this option is selected, the edge is closed. The edge is not taken into consideration in the dynamic assignment. The edge is highlighted in red in network editors if the **Synchronization** icon has been selected in the **Edges** list: 
Length	Length of the edge in the network in meters
ConvTravTm	**Converged (travel time)**: ☑ If this option is selected, the travel time is converged. The edge fulfills the convergence criterion **Travel time on edges** for all completed time intervals (see "Attributes for achieving convergence" on page 782).
ConvVol	**Converged (volume)**: ☑ If this option is selected, the volume is converged. The edge fulfills the convergence criterion **Volume on edges** for all completed time intervals (see "Attributes for achieving convergence" on page 782).

Notes:

- The results of the last iteration are only shown in the list **Edges** when the cost file and the path file were saved in this iteration.
- An edge between nodes is ignored in dynamic assignment if it returns to a previously visited link more than three times (for example, if the edge contains more than three parking bays or stop bays). Path files and cost files from older Vissim versions, which contain such edges, can no longer be used.
- An edge in the dynamic assignment is ignored when it does not contain a parking lot which spans over all of the lanes, i.e. zone connectors and abstract parking lots on all links and real parking spaces only on links with one lane (see "Modeling parking lots and zones" on page 698). Path files and cost files from older Vissim versions, which contain such edges, can no longer be used.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- To edges
- Link sequence: Numbers of links and connectors via which the edge leads.
- From edges

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

7.5.3.5 Excluding edges from dynamic assignment

If a node-edge graph has been generated, you can block edges to exclude them from dynamic assignment. If a toll route leads via a blocked edge, the corresponding managed lanes routing decision does not take effect.

1. Select **Network** > **Edges** in the **Lists** menu.

   *The list **Edges** opens.*

2. Activate the option **Closed** (**Closed**).

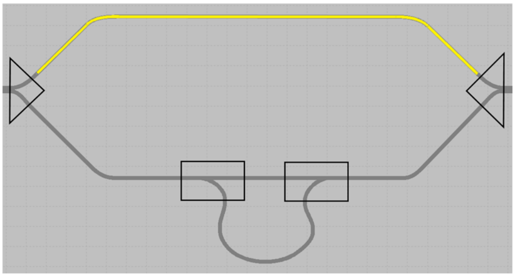
The edge is not taken into consideration in the dynamic assignment. The edge is marked red in the network editor.

*After blocking or unblocking an edge, Vissim regenerates the node-edge graph of dynamic assignment (see "Building an Abstract Network Graph" on page 697), (see "Generating a node-edge graph" on page 718). When you choose the **Undo** or **Redo** command, the node-edge graph of dynamic assignment is deleted and must be regenerated (see "Generating a node-edge graph" on page 718). You cannot use the **Undo** command to undo the regenerated graph.*

7.5.3.6 Visualizing edges

Edges are represented in network editors in the following colors:

- Yellow: open for dynamic assignment (in figure above)
- Red: blocked for dynamic assignment



Displayed values based on the last iteration in which the path file was updated.

Displayed costs based on the last saved cost file.

7.5.3.7 Deleting segments of an edge from segment nodes

1. Ensure that the relevant node is a segment node and not a polygon node.
2. Select **Network** > **Edges** in the **Lists** menu.

   *The list **Edges** opens.*
3. Ensure that a node-edge graph has been generated (see "Generating a node-edge graph" on page 718).

   *In the **Edges** list, the edges of the segments nodes are displayed in the Vissimnetwork.*
4. Right-click the entry of your choice.
5. From the context menu, choose **Delete node segments**.

*All segments of the turn edge are deleted from the **Edges** list and the network editor.*

7.6 Modeling traffic demand with origin-destination matrices or trip chain files

The traffic demand for the dynamic assignment is modeled with origin-destination matrices. Beyond that it is also possible to model the traffic demand with a trip chain file. You can also combine both options.

You can use both options in combination with input flows and static routes, for example for pedestrian flows. In this process, static traffic cannot be taken into account by the dynamic assignment.

7.6.1 Modeling traffic demand with origin-destination matrices

An OD matrix defines the travel demand based on the number of trips between the zones (see "Defining an origin-destination matrix" on page 722). The zones are the starting points and end points of the trips. The number of trips applies to each pair of districts for a given time interval. You define the time interval in the matrix attributes (see "Matrix attributes" on page 724). You edit the number of trips of the OD matrix in the matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724). Origin-destination matrices are also called OD matrices, demand matrices or trip matrices.

You can specify multiple origin-destination matrices for a simulation with dynamic assignment. Each origin-destination matrix can contain a different vehicle composition or apply to a different time interval. The time intervals can overlap arbitrarily because the traffic generated at any time is always the result of the total traffic from all matrices that include this point in their validity interval.

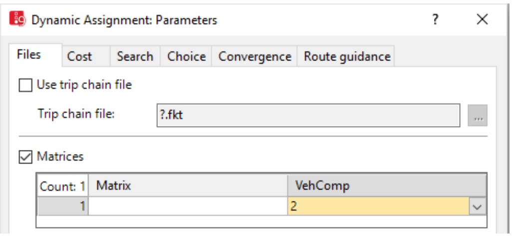
7.6.2 Defining an origin-destination matrix

You can define an OD matrix in the parameters of dynamic assignment or in the **Matrices** list. Matrices are saved to the **.inpx* file. In the Matrix editor, you enter traffic demand data into the OD matrix (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).

Defining an OD matrix in the parameters section of dynamic assignment

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the option **Matrices**.
3. Right-click in the list.
4. From the shortcut menu, choose **Add**.
5. Move the mouse pointer to the new cell and click the symbol.

   Depending on the matrices already defined, the box contains the next consecutive number available for the new matrix.

6. In the **VehComp** column, select the desired vehicle composition.
7. Edit the attributes of matrix (see "Matrix attributes" on page 724).
8. Edit the number of trips in the matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).

Defining an OD matrix in the Matrices list

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Matrices**.
2. In the list, on the toolbar, click the **Add** button .

A new row with default data is inserted.

The Matrix editor is opened. The matrix dimension automatically depends on the number of zones defined. The dimension is displayed in the top left box.

3. Edit the attributes of matrix (see "Matrix attributes" on page 724).
4. Edit the number of trips in the matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).

7.6.3 Selecting an origin-destination matrix

You can select one or multiple OD matrices for the dynamic assignment. And you can select a vehicle composition for each matrix. The vehicles of this vehicle composition operate between the zones that are defined in the OD matrix. The selection of vehicles is effected randomly.

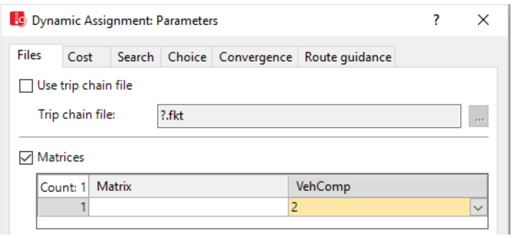
The desired speed of the vehicle is not taken from the desired speed distribution which is defined for the vehicle composition; instead, it is taken from the desired speed distribution which is defined for the parking lot, from which the vehicle begins its trip (see "Attributes of parking lots" on page 500).

In Vissim, you can edit OD matrices in the Matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724)

Matrices are exchanged between Vissim and Visum via Visum data export from Vissim and ANM export from Visum.

1. Ensure that an OD matrix has been defined (see "Defining an origin-destination matrix" on page 722).
2. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*



3. Select the option **Matrices**.
4. Move the mouse pointer to the **Matrix** column and click the symbol.
5. Select the desired matrix.

   *In the **Matrix** column, number and name of the matrix are displayed.*

6. In the **VehComp** column, select the desired vehicle composition (see "Modeling vehicle compositions" on page 452).

You can edit additional attributes of dynamic assignment (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

7.6.4 Matrix attributes

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Matrices**.

   *The **Matrices** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains the following attributes:

Element	Description
No	Unique number of the matrix
Name	Name of matrix
FromTime	Start of the time interval from 0:00 o'clock for which the matrix applies
ToTime	End of the time interval from 0:00 o'clock for which the matrix applies

7.6.5 Editing OD matrices for vehicular traffic in the Matrix editor

In the matrix editor, enter the number of journeys for the vehicle traffic between the zones for an OD matrix. You can open multiple Matrix editors to edit their values.

In Viswalk, you edit OD matrices for pedestrians in the **Pedestrian OD Matrix** (see "Pedestrian OD matrices" on page 977).

7.6.5.1 Opening the Matrix editor and entering the number of trips

1. Ensure that at least one matrix has been defined (see "Defining an origin-destination matrix" on page 722).
2. Make sure that the parking lots of the type **Zone connector** are defined and that they are allocated to a zone (see "Defining parking lots for dynamic assignment" on page 700).
3. From the **Traffic** menu, choose > **Dynamic Assignment** > **Matrices**.

   *The **Matrices** list opens.*
4. Select the desired entry.
5. On the list toolbar, click **Edit object** .

Tip:

- Alternatively, in the **Matrices** list, in the row of the desired matrix, double-click the header.
- Alternatively, in the **Matrices** list, double-click the row of the desired matrix and select **Edit** in the context menu.

The Matrix editor is opened. The matrix, the name of the zones (red) and the sum (green) of trips between zones are displayed.

Matrix Editor (Matrix '2')

5 x 5			1	2	3	4	5
	Name		Zone 1 North	Zone 2 East	Zone 3	Zone 4	Zone 5
		Sum	2560,00	2250,00	1460,00	1650,00	1950,00
1	Zone 1 North	3150,00	900,00	900,00	300,00	450,00	600,00
2	Zone 2 East	2050,00	600,00	750,00	100,00	500,00	100,00
3	Zone 3	1300,00	300,00	400,00	200,00	200,00	200,00
4	Zone 4	750,00	60,00	80,00	60,00	200,00	350,00
5	Zone 5	2620,00	700,00	120,00	800,00	300,00	700,00

The matrix dimension automatically depends on the number of zones defined. The dimension is displayed in the top left box.

In the next step, you can also define several fields and enter values.

6. Into the white and blue boxes, enter the number of trips between zones.

7.6.5.2 Copying and pasting the number of trips

In the Matrix editor, you can select one or multiple cells, copy the values and paste them into other cells. If you copy values of multiple cells, they must be selected according to one of the following patterns:

- The cells or rows are immediately next to each other
- The cells or columns are immediately under each other
- 2 x 2 cells or multiple thereof

1. Select the desired cells.
2. On the Matrix editor toolbar, click .

   In the next step, to paste the cells, follow the same pattern used to copy them.
3. Select the desired target cells.
4. On the Matrix editor toolbar, click .

7.6.5.3 Editing graphic parameters

1. To edit graphic parameters in the matrix, on the Matrix editor toolbar, click **Matrix editor graphic parameters** .
2. Make the desired changes:

Element	Description
Column width	Column width for all columns in pixels
Decimals	Number of decimal places, default value 2
Row height	Row height in pixels for all rows, default value 20

7.6.5.4 Defining column width

1. Right-click the column header of the Matrix editor.
2. Choose the desired entry from the context menu.

Element	Description
Set Optimum Width for All Columns	Adjusts column width for alls column to accommodate the longest column title and longest cell entry
Adjust Column Widths To Window Size	Adjusts column width for all columns to the window width

7.6.6 Using OD matrices from previous versions

From version 9 and later, Vissim manages OD matrices in the **Matrices** list (see "Matrix attributes" on page 724), (see "Modeling traffic demand with origin-destination matrices" on page 721). You can view and edit the matrix content in the matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).

For the dynamic assignment, Vissim up to version 8 requires at least one OD matrix that is saved to an **.fma* file. You can choose the **.fma* file from the dynamic assignment parameters in Vissim up to version 8.

You cannot copy the OD matrix from the **.fma* file and paste it immediately into the matrix editor in Vissim.

You can export an OD matrix from a previous version to Vissim version 9 and higher. The following applications and solutions are distinguished:

- In the matrix editor, **Read from file**
- In the **Matrices** list > **Read from file**
- Open the **.inpx* file from Vissim 6,7 or 8, in Vissim 9 or higher.
- Copy OD matrix and paste it into Vissim version 9 or higher.

7.6.6.1 Reading from file in the matrix editor

1. In the matrix editor, make sure that the origin-source matrix of the desired matrix in the **Matrices** list is displayed.
2. Make sure that the dimensions of the OD matrix in the matrix editor correspond to the dimensions that result from the number of zones in the **.fma* file.
3. In the matrix editor, click the **Read from file** icon.

   *The **Read matrix from file** window opens.*
4. Select the **.fma* file of your choice.

5. Click the **Open** button.

*The data is inserted into the matrix editor. In the **Matrices** table, the values of the **TimeFrom** and**TimeUntil** attributes are adjusted on the basis of the values of the *.fma file.*

7.6.6.2 Reading from file in the Matrices list

1. Make sure that the dimensions of the OD matrix in the matrix editor correspond to the dimensions that result from the number of zones in the **.fma* file.
2. In the **Matrices** table, right-click the matrix of your choice.
3. From the shortcut menu, choose **Read from file**.

   *The **Read matrix from file** window opens.*
4. Select the **.fma* file of your choice.
5. Click the **Open** button.

*The data is inserted into the matrix editor. In the **Matrices** table, the values of the **TimeFrom** and**TimeUntil** attributes are adjusted on the basis of the values of the *.fma file.*

7.6.6.3 Open the **.inpx* file from Vissim 6,7 or 8, in Vissim 9 or higher.

If you have access to the following files, to the Vissim versions and meet the requirements, you can open an older network file in Vissim 9 or higher and create an OD matrix:

- **.fma* file selected in another **.inpx* network file
- You have so far used the **.inpx* network file in Vissim 6, 7 or 8. You can also open an **.inp* network file from Vissim 5.40 in subsequent versions and save it as **.inpx* file. Network files used in Vissim 5.40 and previous versions cannot be opened in Vissim version 6 or above.
- Vissim from version 9

You have the following options:

- Open the **.inpx* network file in Vissim 9 or higher and save it.
- Only for **.inpx* files from Vissim version 9 and higher: Additionally read the **.inpx* file into Vissim. In the **Read additionally** window, choose only the **Matrices** entry from (see "Reading a network additionally" on page 361).

  *Vissim creates the matrices in the **Matrices** list (see "Matrix attributes" on page 724). They are saved to the *.inpx file. The *.fma files are thus no longer required. You can edit the content of any matrix in the matrix editor (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).*

1. In the attributes of the parking lots of the **Zone connector** type that you need for the dynamic assignment, make sure that you have selected the desired zones.
2. In the list **Matrices**, make sure that in the **TimeFrom** attribute the desired start time of the interval and in the **TimeUntil** attribute the desired end time of the interval are defined(see "Matrix attributes" on page 724).

3. Before starting the simulation, make sure that the dynamic assignment parameters are as follows:
   - On the **Files** tab, the **Matrices** option is selected.
   - The desired matrix is selected in the list below.

7.6.6.4 Copy OD matrix and paste it into Vissim version 9 or higher.

If you have access to the following files and to Vissim version 9 or higher and meet the requirements, but do not have Vissim version 6, 7 or 8, you can copy the data from the **.fma* file and paste it into a spreadsheet program or a similar program:

- **.fma* file with OD matrix
- Vissim version 9 and higher
- Spreadsheet program or similar program
- **.inpx* network file with or without Vissim network and with or without defined zones

Copy the data from the spreadsheet program or similar program to the clipboard and paste it into Vissim version 9 or higher, in the open network file, into the fields of the matrix editor.

Making sure that the zones of your choice are defined

From Vissim version 9 and higher, the number of zones defines the dimensions of the OD matrix. Therefore, before copying the data, make sure that in Vissim the zones are defined which match the copied data:

1. To open Vissim version 9 or higher.
2. If you want to paste the copied data into an existing network file, open this **.inpx* network file.
3. From the **Traffic** menu, choose > **Dynamic Assignment** > **Zones**.

   The number of necessary zones depends, for example, on your planning targets, the parking lots of the ***Zone connector*** *type and/or the OD matrix that you want to insert.*
4. Make sure that the number of defined districts corresponds to the desired dimension of the OD matrix (see "Defining zones" on page 704).
5. From the **Traffic** menu, choose > **Dynamic Assignment** > **Matrices**.

   The ***Matrices*** *list and the matrix editor open.*
6. If no matrix is defined in the **Matrices** list, define a matrix(see "Defining an origin-destination matrix" on page 722).
7. If matrices are defined in the **Matrices** list, make sure that the matrix attributes, that you want to copy to the trips, meet your requirements(see "Matrix attributes" on page 724).
8. In the **Matrices** list, double-click the matrix whose dimension you want to check and which you want to use at a later point in time in order to insert data.

   In the matrix editor, the associated OD matrix is displayed and selected. The dimension of the OD matrix is automatically based on the number of zones defined. The dimension is *displayed in the top left box (see "Editing OD matrices for vehicular traffic in the Matrix editor" on page 724).*

9. If you want to change the dimension of the OD matrix, change the number of zones (see "Defining zones" on page 704).
10. From the **File** menu, choose > **Save**.
11. In Vissim, keep the network file and matrix editor open, so that you can insert the data after the next steps.

Copying an OD matrix

1. Open the **.fma* file in a spreadsheet program.

   Some spreadsheet programs are supported by a wizard through which you can configure the distribution of the copied data across the individual cells.

2. Make sure that each value, that indicates a number of trips, appears in a cell.
3. If desired, save the file.

   If you then select data in the spreadsheet program, you can select one or more cells. If you select multiple cells, these must cover a regular range of contiguous cells, for example 1 x 4, 5 x 3, or 6 x 6 cells.

   If you select more than one cell, make sure that the range you selected corresponds to the dimension of the OD matrix in the matrix editor of Vissim that you want to copy the data to.

4. In the spreadsheet program, select the desired range.
5. Press CTRL+C.

Inserting an OD matrix

1. Switch to Vissim.
2. Make sure that:
   - The network file of your choice is open.
   - In the matrix editor, the OD matrix is displayed for the desired matrix.
   - The OD matrix in the matrix editor has the desired dimension.

   In the next step, make sure that you select a range in the matrix editor, which can include the range from the clipboard:

   - The range you copied must not be larger than the dimension of the OD matrix in the matrix editor.
   - If you select more than one cell in the matrix editor, the range you selected must not be smaller than the range you copied.
   - If you select a single cell, the range you copied must not be larger than the range available based on the cell you selected.

3. In the matrix editor, select the desired number of cells.
4. Press CTRL+V.

The data is inserted into the matrix editor based on its volume and the number of selected cells. If the selected range is larger than the range you copied, the data will be inserted multiple times.

5. In the attributes of the parking lots of the **Zone connector** type that you need for the dynamic assignment, make sure that you have selected the desired zones (see "Defining parking lots for dynamic assignment" on page 700).
6. In the **Matrices** list, make sure that in the **TimeFrom** attribute the desired start time of the interval and in the **TimeUntil** attribute the desired end time of the interval are defined(see "Matrix attributes" on page 724).
7. In the menu **Traffic** > **Dynamic assignment** > **Parameters**, go to the **Files** tab and select the ☑ **Matrices** option (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
8. In the list below, in the **Matrix** column, select the matrix of your choice.

7.6.7 Modeling traffic demand with trip chain files

In addition to origin-destination matrices, the traffic demand can also exist in trip chain files **.fkt* for dynamic assignment. A trip chain is defined as a result of a trip. In contrast to origin-destination matrices, a trip chain file delivers detailed data regarding trips of individual vehicles for the simulation. Therefore the generation of trip chain files is more complex compared to that of origin-destination matrices (see "Modeling traffic demand with origin-destination matrices" on page 721).

Trip chains are only used internally in Vissim. Trip chains are generated from origin-destination matrices. Therefore demand data in the form of origin-destination matrices and trip chains can be combined for a simulation run.

A trip chain file contains information regarding trips per vehicle (trip chain).

A trip chain is made up of one or more trips.

A trip chain is allocated a vehicle and is defined by the following:

- Number of the vehicle
- Type of the vehicle
- Number of the origin zone

One or more trips result from the number and the type. A trip is defined by a group of numbers: four numbers for the data format 1.1. or five numbers for the data format 2.1:

- Departure time
- Number of the destination zone
- World coordinates of the destination (only for format version 2.1)
- Number of the activity
- Minimum dwell time

The departure time of the next trip is calculated from the arrival time in the zone and the minimum dwell time for the activity. The specified departure time of the next trip is only considered when the minimum dwell time is provided for: if the vehicle reaches the zone too late, the departure time is moved accordingly. The minimum dwell time is then added to the current arrival time.

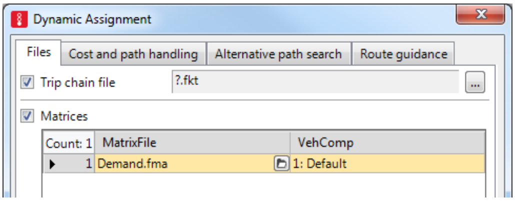
Example trip chain file of a vehicle

```
07:00 departure from zone 1 (home)
to zone 2 (work)
dwell time 9 hours
17:45 departure from zone 2 (work)
to zone 3 (supermarket)
dwell time 30 minutes
6:30 PM departure from zone 3 (work)
to zone 1 (home)
dwell time 11 hours
```

7.6.8 Selecting a trip chain file

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*



2. Select the option **Trip chain file**.
3. Click on the icon **Select file**.
4. Select the desired folder.
5. Select the desired trip chain file.
6. Click the **Open** button.

The trip chain file is displayed. You can edit the attributes of the dynamic assignment (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

7.6.9 Structure of the trip chain file **.fkt*

Each row of the trip chain file contains a trip chain from a series of trips. You have to separate columns with a semicolon.

The first row of a trip chain file must contain the number of the format version used, for example 1.1.

Each subsequent row contains trip chains:

- Column 1: Vehicle number
- Column 2: Vehicle type
- Column 3: Origin zone number
- In the data format version 1.1 every trip is described by four columns, starting from column 4.
- In the data format version 2.1 every trip is described by five columns, starting from column 4. Between destination zone number in column 5 and activity number in column 7 it contains the destination global coordinates:
  - If you do not want to use the center of the destination zone for the determination of the value of "distance from desired zone”, enter the global coordinates of the destination in parentheses.
  - If you want to use the coordinates of the center, enter an empty pair of brackets [] in data format version 2.1.

The format description in BNF (Backus Naur Form)

Specific format version 2.1 entries are highlighted in **bold**.

<trip chain file> ::= <version> {<trip chain>}

<version> ::= <real> <nl>

<trip chain> ::= <vehicle> <vehicle type> <origin> {<trip>} <nl>

*<trip> ::= <departure> <destination>***<coordinates>***<activity> <minimum dwell time>*

<vehicle> ::= <cardinal> <semicolon>

<vehicle type> ::= <cardinal> <semicolon>

<origin> ::= <cardinal> <semicolon>

<departure> ::= <cardinal> <semicolon>

<destination> ::= <cardinal> <semicolon>

<coordinates> = <left parenthesis> <x coordinate> <comma><y coordinate> <right parenthesis> <semicolon>

| <opening square bracket> <closing square bracket> <semicolon>

<x-coordinate> = <real>

<y-coordinate> = <real>

<comma> = ","

<left parenthesis> = "("

<right parenthesis> = ")"

<opening square bracket> = "["

<closing square bracket> = "]"

<activity> ::= <cardinal> <semicolon>

<minimum dwell time> ::= <cardinal> <semicolon>

<nl> ::= new line

<semicolon> ::= semicolon (;)

<cardinal> ::= positive integer (example: 23)

<real> ::= floating-point number (example: 3.14)

Example of a *.fkt file in version 1.1 format

Example of trip chain file with 12 trip chains:

```
1.1
 1;1;10;  1; 20; 101; 117; 211; 30; 101; 169; 732; 20; 101; 171;
 2;1;10;  4; 20; 101; 255; 334; 30; 101; 147; 815; 20; 101; 124;
 3;1;10;  8; 20; 101; 202; 395; 30; 101; 178; 832; 20; 101; 175;
 4;1;10; 12; 20; 101; 216; 703; 30; 101; 162; 533; 20; 101; 208;
 5;1;10; 16; 20; 101; 164; 601; 30; 101; 251;1134; 20; 101; 159;
 6;1;10; 20; 20; 101; 295; 529; 30; 101; 133; 846; 20; 101; 114;
 7;1;10; 25; 20; 101; 248; 262; 30; 101; 256; 987; 20; 101; 117;
 8;1;10; 29; 20; 101; 169; 322; 30; 101; 164; 463; 20; 101; 141;
 9;1;10; 31; 20; 101; 138; 543; 30; 101; 212; 405; 20; 101; 252;
10;1;10; 35; 20; 101; 296; 205; 30; 101; 160; 802; 20; 101; 221;
11;1;10; 40; 20; 101; 270; 622; 30; 101; 244; 604; 20; 101; 175;
12;1;10; 44; 20; 101; 189; 151; 30; 101; 185; 419; 20; 101; 227;
```

Example of a *.fkt file in version 2.1 format

Example of trip chain file with 11 trip chains. The global coordinates for the destination are specified for zone 20 only:

```
2.1
 1; 1; 10;  1; 20; (113.0,157.0); 101; 117; 211; 30; []; 101; 169; 732; 20;
(105.0,159.0); 101; 171;
 2; 1; 10;  4; 20; (102.0,160.0); 101; 255; 334; 30; []; 101; 147; 815; 20;
(128.0,153.0); 101; 124;
 3; 1; 10;  8; 20; (126.0,163.0); 101; 202; 395; 30; []; 101; 178; 832; 20;
(117.0,182.0); 101; 175;
 4; 1; 10; 12; 20; (128.0,153.0); 101; 216; 703; 30; []; 101; 162; 533; 20;
```

```
(103.0,155.0); 101; 208;
 5; 1; 10; 16; 20; (114.0,174.0); 101; 164; 601; 30; []; 101; 251;1134; 20;
(113.0,157.0); 101; 159;
 6; 1; 10; 20; 20; (105.0,159.0); 101; 295; 529; 30; []; 101; 133; 846; 20;
(120.0,172.0); 101; 114;
 7; 1; 10; 25; 20; (117.0,182.0); 101; 248; 262; 30; []; 101; 256; 987; 20;
(102.0,160.0); 101; 117;
 8; 1; 10; 29; 20; (119.0,157.0); 101; 169; 322; 30; []; 101; 164; 463; 20;
(121.0,160.0); 101; 141;
 9; 1; 10; 31; 20; (121.0,160.0); 101; 138; 543; 30; []; 101; 212; 405; 20;
(119.0,157.0); 101; 252;
10; 1; 10; 35; 20; (120.0,172.0); 101; 296; 205; 30; []; 101; 160; 802; 20;
(126.0,163.0); 101; 221;
11; 1; 10; 40; 20; (103.0,155.0); 101; 270; 622; 30; []; 101; 244; 604; 20;
(114.0,174.0); 101; 175;
```

7.7 Simulated travel time and generalized costs

The simulation of the traffic flow is used during the dynamic assignment to determine travel times in the network. The travel times are measured for every edge and every evaluation interval. In contrast to travel times, the spatial lengths of the paths and the financial costs do not depend on the traffic conditions. Thus the spatial lengths of the paths and the financial costs do not need to be determined by simulation but can be read directly from the network model.

7.7.1 Evaluation interval duration needed to determine the travel times

The simulation of the traffic flow is used during the dynamic assignment to determine travel times in the network. The travel times are measured for every edge and every **evaluation interval** (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

In dynamic assignment, as opposed to static assignment process, traffic demand and infrastructure are not assumed to be constant over time. Therefore the traffic condition in the network and as a result the travel times will change during the assignment time period. Therefore to cover these changes the total simulation time is divided into smaller evaluation intervals in which travel times are observed separately. An evaluation interval also specifies the point in time after which the path selection of vehicles changes The appropriate duration of the evaluation interval depends on the dynamics of the traffic demand. Select a time period during which the traffic situation is expected to change significantly. In doing so, take the entire simulation period into account, including possible evaluation intervals.

Evaluation intervals of less than 15 minutes rarely make sense, as fluctuations of the measured values increase with shorter intervals. In many cases, evaluation intervals from 15 to 60 minutes are appropriate.

Especially when signal controls are used the evaluation interval must be significantly longer than the cycle times used.

7.7.2 Defining simulated travel times

During a simulation, travel times are measured for each edge in the abstract network graph. All vehicles that have passed through an edge, report the time they have spent on it on leaving. All travel times delivered on an edge during an evaluation interval are averaged and thus yield the travel time for this edge in this evaluation interval.

When an edge is congested and vehicles have spent more than one evaluation interval on an edge, these vehicles report it at the end of the evaluation interval. The vehicles report this even if they have not managed to leave the edge until the end of the evaluation interval. Thus Vissim gets information also from heavily congested edges, even if due to the congestion no vehicle reaches the end to report about the congestion.

The travel times measured in the current iteration are not used directly for path selection in the same iteration, but are adopted only in the following iterations. This behavior is useful because, for example, for a path selection on a Tuesday between 9:00 to 10:00 the relevant travel time is not on the same day between 8:00 to 9:00, but rather the travel time between 9:00 to 10:00 on the Monday before.

For the simulation of the experience growing with time, not only the travel time of the previous iteration is taken into account, but in particular the travel times of all previous iterations.

You can set the more distant measurements to have less influence. For this the following methods can be used:

- exponential smoothing of the travel times (see "Selecting exponential smoothing of the travel times" on page 735)
- MSA (Method of successive averages) (see "Selecting the MSA method for travel times" on page 736)

7.7.3 Selecting exponential smoothing of the travel times

You can set the more distant measurements to have less importance compared to the recent iterations. With a smoothing factor you set the relative weight of the respective recent iteration.

You can assign less importance to the more distant measurements, using the Method of Successive Averages (MSA) (see "Selecting the MSA method for travel times" on page 736).

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters** (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Cost** tab.
3. Then, in the **Smoothing method** section, select **Exponential smoothing with smoothing factor**.
4. Enter the desired smoothing factor:
   - for the equilibrium assignment `1.0`
   - for the stochastic assignment according to Kirchhoff, the default value is 0.2
5. Confirm with **OK**.

*If the option **Store costs** is selected in the **Files** tab, the expected travel times are saved after every iteration for the next iteration in the Vissim cost file *.bew, from where they are entered into the path selection model.*

After measurement of the new travel times, the smoothed travel time is computed for each edge as the weighted sum of the following:

- the old smoothed travel time from previous iterations
- the newly measured travel time from the current iteration

The new smoothed value represents the travel time that we expect in the next iteration.

$$T_i^{n,\kappa} = (1-\alpha)\cdot T_i^{n-1,\kappa} + \alpha \cdot TO_i^{n,\kappa}$$

Where:

K = index of the evaluation interval within the simulation time

n = index of the iteration

i = index of the edge

$TO_i^{n,\kappa}$ = measured (observed) travel time on edge i for interval k in iteration n

$T_i^{n,\kappa}$ = smoothed travel time on edge i for interval k in iteration n

α = the given constant smoothing factor

This kind of smoothed average contains the information from all preceding iterations. The older an iteration, the less influence has the measured value derived from it. With a smoothing factor of for example 0.5, the current iteration n has a weight of 50 %, iteration $(n-1)$ has a weight of 25 %, and iteration $(n-2)$ has a weight of 12.5 % and so on.

7.7.4 Selecting the MSA method for travel times

With the Method of Successive Averages (MSA), you give each preceding iteration as much weight as the current iteration. This results in the arithmetic mean from all iterations. In this way, the influence of any further iteration becomes increasingly smaller.

The MSA method parameter depends on the cost file **.bew*:

- If you select the option **MSA (Method of Successive Averages)** and there is no cost file **.bew* saved yet by the dynamic assignment, the parameter of the MSA method will be set automatically by Vissim.
- If you have already performed a dynamic assignment and a cost file **.bew* is stored, enter the number of iterations with which the file **.bew* was created. If you enter a smaller value than the actual number of iterations, the subsequent iterations will be weighted higher. Enter a smaller value when the path evaluation shows that the measured travel times deviate significantly from the expected travel times (see "Showing data about paths of dynamic assignment in lists" on page 1109).

You can assign less importance to more distant measurements using **exponential smoothing with smoothing factor** for the travel times (see "Selecting exponential smoothing of the travel times" on page 735).

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters** (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

   *The **Dynamic Assignment: Parameters** window opens.*
2. Select the **Cost** tab.
3. In the **Smoothing method** section, select **MSA (Method of Successive Averages), so far**.
4. When a dynamic assignment was performed and thereby a cost file **.bew* has been stored, enter the number of iterations with which the **.bew* file was created in the field **Iterations**.
5. Confirm with **OK**.

*If the option **Store costs** is selected in the **Files** tab, the expected travel times are saved after every iteration for the next iteration in the Vissim cost file *.bew, from where they are entered into the path selection model.*

After measurement of the new travel times, the smoothed travel time is computed for each edge as the weighted sum of the following:

- the old smoothed travel time from previous iterations
- the newly measured travel time from the current iteration

The new smoothed value represents the travel time that we expect in the next iteration.

$$T_i^{n,\kappa} = \left(1 - \frac{1}{N+n}\right) \cdot T_i^{n-1,\kappa} + \frac{1}{N+n} \cdot TO_i^{n,\kappa}$$

Where:

N = user-defined value for number of existing iterations that shall be considered

K = index of the evaluation interval within the simulation time

n = index of the iteration

i = index of the edge

$TO_i^{n,\kappa}$ = measured (observed) travel time on edge i for interval k in iteration n

$T_i^{n,\kappa}$ = smoothed travel time on edge i for interval k in iteration n

$\frac{1}{N+n}$ = variable smoothing factor from parameter N and the iteration index

7.7.5 General cost, travel distances and financial cost in the path selection

The path selection depends on the following factors:

- Travel time
- spatial length of the paths
- financial cost on the paths, for example for toll

In contrast to travel times, the spatial lengths of the paths and the financial costs do not depend on the traffic conditions. Thus the spatial lengths of the paths and the financial costs do not need to be determined by simulation but can be read directly from the network model.

To account for these three factors in path selection, for each edge the so called general cost is computed as a weighted sum:

General cost = a • travel time + b • travel distance + g • financial cost + Σ cost surcharge 2

Where:

- The **Cost Coefficients** are entered in the attributes of the vehicle type (see "Defining path selection behavior with general cost function" on page 754):
  - α for **travel time**
  - β for**travel distance**
  - γ for **link costs**
- Cost coefficients are specific to vehicle types. This allows the modeling of driver groups with different path selection behaviors, with different time, path and money determinants (see "Defining simultaneous assignment" on page 754).
- The travel distances are determined from the geometry of the links.
- The financial cost of an edge is the sum of the costs of all links that are contained in that edge.
- The cost is computed by multiplying the cost rate per kilometer specified in the link by the length of the route which is contained in the edge. If for a link the attribute **Surch1** contains a value, it is added (see "Attributes of links" on page 409).
- Link attribute **Surcharge 2:**: Additional surcharge for the link. Is added to general link costs without weighting (see "Attributes of links" on page 409).

7.8 Path search and path selection

In the dynamic assignment, several paths are mostly found for an OD pair that a vehicle can take. The decision for one of the paths found via path selection is based on the generalized costs of the paths. The generalized costs consist of the travel time, distance, and link costs (see " General cost, travel distances and financial cost in the path selection" on page 737).

The distribution of demand data to the paths can be done according to the following distribution models:

  - **Use volume (old)**: Base path search exclusively on volumes of previous simulation runs. These may stem from a previous simulation run or an ANM import.
  - **Stochastic assignment (Kirchhoff)**: The distribution is carried out in each iteration according to Kirchhoff based on the smoothed generalized costs of the previous

iteration. Less vehicles use paths with higher costs than paths with lower costs. Paths with the same cost get the same volume, regardless of their capacity. The result of the assignment is the following distribution to the paths:
  - Paths with higher volumes have lower costs
  - Paths with lower volumes have higher costs
  - Paths with identical volumes have identical costs
- **Equilibrium assignment**: Redistributes demand across paths proportionally to costs, from expensive to inexpensive paths, for each OD pair in each iteration: The volume of paths that are more expensive than the average is reduced. The height of the specified volume is proportional to the additional costs of the path. All paths that are cheaper than the average path costs are assigned additional volume. The additional volume is proportional to the cost saving compared with the average path costs (see "Method of path selection with or without path search" on page 741). The result of the assignment are identical generalized costs on all paths of a parking lot OD pair, possibly with very different volumes, dependent on the respective capacity of the path.

Choose whether you want path selection to be based on the travel time measured on paths or on the total of travel times on edges. In doing so, you define the calculation of generalized costs. The desired method of cost calculation can be selected in the parameters of dynamic assignment **Cost for path distribution with Kirchhoff:** (see "Attributes for calculating costs as a basis for path selection" on page 775).

7.8.1 Calculation of paths and costs

A path is a sequence of edges on which a vehicle can move through the road network. In the dynamic assignment, paths start at an origin parking lot and end at a destination parking lot. Since there are usually several different paths between an origin parking lot and a destination parking lot, Vissim must also model the decision of the driver as to the path he selects. In dynamic assignment, during path selection, Vissim identifies the number of paths available between an origin parking lot and a destination parking lot. The software then performs a path search for the path selection methods **Sum of edge travel times** and **Measured path travel times** according to Kirchhoff. The path selection is a special case of discrete selection because the selection probabilities must be calculated for a set of discrete alternatives. To do so, a utility function for evaluating the individual paths as well as a decision function are defined in the traffic assignment, which are based on these evaluations.

The evaluation is calculated in the form of generalized costs from expected travel time, travel distance and financial costs for all edges. The generalized costs of a path are defined as the total cost of edges from which the path is composed:

$$C_R = \sum_{a \in R} C_a$$

These include:

C = the generalized costs

R = a path

a = an edge that occurs in R

7.8.2 Path search finds only the best possible path in each interval

In Vissim it is assumed that not all drivers use only the best route from one parking lot to another, but that traffic is distributed across all known paths. For this, it would be useful to know the n best paths for each origin-destination relation. There are, however, no efficient methods to directly calculate the n best paths in context of a traffic assignment in a useful way.

The shortest path search finds the best path for each origin-destination pair.

Therefore, in each iteration of a simulation, the shortest path search of Vissim searches for the best path for each origin-destination relation. Due to the fact that over the course of an iteration the traffic situation and therefore the travel time on the edges changes until convergence is reached, different best paths can result in the iterations. As long as Alternative path search is not activated, the shortest path search carried out by Vissim never results in more than one best path for an OD pair.

Path file **.weg* saves each best path.

All found paths, which qualify as the best paths in an iteration, are collected in Vissim and saved in the path file **.weg*. These paths are then available for the following iterations.

Best path based on generalized costs

The criteria for the "best" path are the generalized costs. Due to the fact that the weighted coefficient for the generalized costs depends on the vehicle type, different best paths can be found for different vehicle types.

Path search at the beginning of each evaluation interval

The path search takes place at the beginning of the evaluation interval and uses the expected generalized costs which were determined for this evaluation interval in the previous iterations.

First simulation run uses path length

Because the first iteration does not yield any travel time information from the previous simulation, the length of the path [m] is used.

Default travel times for edges not yet used

For the following iterations, Vissim no longer uses path lengths, but enters a fictitious travel time of 0.1 seconds for edges not yet used by vehicles. This results in the use of paths with unused edges to appear attractive when searching according to route. It may be possible that only a few useful paths are found in the initial iterations. However, the collection of known paths (for which travel times were measured and generalized costs calculated) will grow more quickly in the path collection, if drivers are encouraged to try out unknown paths.

Weighting of travel distance helps avoid detours

This "Eagerness to experiment" of the driver may be influenced by a weighting of the distance traveled in the generalized cost functions. This results in long detours being avoided. Generally it is an advantage to find as many paths as possible. When unrealistic paths are found, these can be discarded in a later iteration. This can be defined in the options for path searches (see "Influencing the path search by using cost surcharges or blocks" on page 787).

7.8.2.1 Alternative path search

Optionally, you can carry out an additional search for Alternative path search with stochastic modifications of the edge evaluations or shortest path price increases (see "Performing an alternative path search" on page 749).

7.8.3 Method of path selection with or without path search

You can choose from different procedures for path selection (see "Path search and path selection" on page 738):

- In the parameters of dynamic assignment, in the **Choice** tab, select the procedure **Use old volumes (no path search)** to select a path without carrying out a path search. In this case, the probability of a path being used corresponds to its attribute value **Volume (old)** share in the total of attribute values **Volume (old)**, of all paths of the same OD pair. These attribute values stem from an ANM import or the path file of a previous simulation run.
- If in the parameters of dynamic assignment, on the **Choice** tab, you selected the path choice model **Stochastic assignment (Kirchhoff)** or **Equilibrium assignment**, your path search is followed by path selection. The vehicles are then distributed across the paths depending on the distribution model and based on the distribution formula according to Kirchhoff or for equilibrium assignment.

The following descriptions require that the destination parking lot and potential routes to it are already known. Path search finds only the best possible path in each interval for each OD pair, but all found paths can be used in all intervals (see "Path search finds only the best possible path in each interval" on page 740). For the dynamic assignment, the drivers select the route at the time they depart from the origin parking lot.

7.8.3.1 Calculating utility

One of the basic assumptions in path selection according to Kirchhoff is that not all drivers use the best path, but that all known paths are used and have different costs. However, a large percentage of the traffic should be distributed across the better paths. The quality of paths is evaluated using the generalized costs. Generalized costs are contrary to the "benefit" involved in the theory of discrete decisions. Thus the benefit is defined as the reciprocal of the generalized costs:

$$U_j = \frac{1}{C_j}$$

Where

Uj = the benefit of path j

Cj = the generalized costs of path j

7.8.3.2 Calculating the decision behavior using the Logit function

The most frequently used and thus also the most theoretically analyzed function for mapping the decision behavior is the Logit function:

$$p\left(R_j\right) = \frac{e^{\mu U_j}}{\sum e^{\mu U_i}}$$

Where

Uj = the benefit of path j

$p(Rj)$ = the probability that path j is selected

μ = the sensitivity parameter of the model (>0), **Logit scaling factor** for destination parking lot selection

The sensitivity parameter determines how strongly the distribution responds to benefit differences. A low value would result in a quite similar distribution without any major influence of the benefit, and a high value would result in virtually every driver selecting the best path.

7.8.3.3 Distribution according to Kirchhoff

If the logit function is applied with the cost function defined above, this leads the model to attach the same importance to the difference between 5 and 10 minutes of travel time as the difference between 105 and 110 minutes of travel time because the logit function is translationally invariant and thus considers only the absolute difference of benefits. Obviously, this modeling is not particularly appropriate, because in reality two paths which have a travel time of 105 and 110 minutes are basically considered equally good, whereas paths of 5 and 10 minutes are perceived as significantly different. To approximate the real assessment, the distribution formula according to Kirchhoff is used in Vissim:

$$p\left(R_j\right) = \frac{U_j^k}{\sum_i U_i^k}$$

Where

Uj = the benefit of path j

$p(Rj)$ = the probability that path j is selected

k = the sensitivity parameter of the model

The sensitivity parameter also determines here how sensitively the model responds to differences in the benefits. For Kirchhoff, the ratio of benefits determines the distribution and not the absolute difference of benefits, thus only slight variations arise in the paths with 105 and 110 minutes of travel time, whereas the path with 5 minutes of travel time receives much more traffic than the path with 10 minutes of travel time.

In fact, the Kirchhoff function is also a logit model. It arises from the logit function described above if the logarithmic benefit is used as a utility function:

$$p(R_j) = \frac{U_j^k}{\sum_i U_i^k} = \frac{e^{k \cdot \log U_j}}{\sum_i > e^{k \cdot \log U\, i}} = \frac{e^{-k \cdot \log C_j}}{\sum_i e^{-k \cdot \log C\, i}}$$

Cj are the generalized costs of path j in this case.

7.8.3.4 Distribution with the equilibrium assignment

The equilibrium assignment redistributes demand across paths proportionally to costs, from expensive to inexpensive paths, for each parking lot OD pair.

The volume of paths that are more expensive than the average is reduced. The volume of these less expensive paths is also reduced and part of it assigned to cheaper paths. All paths that are cheaper than the average path costs are assigned additional volume. The cheaper the path, the more volume it is assigned (see "Equilibrium assignment – Example" on page 746).

Assign normalized probability for path selection

As with the procedure according to Kirchhoff, each path j is a assigned a normalized probability $p_{s,n,v}^j$.

where:

s: simulation run

n: time interval

v: vehicle class

The probabilities $p_{s,n,v}^j$ are calculated before each time interval n from the generalized costs $c_{s,n,v}^j$. In equilibrium assignment, the target volume is determined, which is different from the method according to Kirchhoff $g_{s,n,v}^j = f_{s,n,v}^j \; g_{\mathrm{JJ}}$.

Where: $J_{s,n,v}^j$ = $f_{s,n,v}^j$ the **Target volume (relative)** attribute and g_{JJ} is the total volume of the OD parking lot relation. g_{JJ} is iteratively calculated, so that $g_{s,n,v}^j$ is a function γ with the following variables:

- the generalized cost $c_{s,n,v}^j$ of the corresponding path
- the average generalized cost $\overline{c_{s,n,v}}$
- the relative target volumes of the previous simulation run $g_{\text{s-1},n,v}^j$

$$g_{s,n,v}^j = \gamma\left(g_{s-1,n,v}^j,\; c_{s,n,v}^j,\; \overline{c_{s,n,v}}\right)$$

Redistributing volumes proportionally to costs

In the following

$\widehat{J}$: is the number of paths at the beginning of a new time interval for each OD parking lot relation, including newly found paths and excluding previously deleted paths.

$$c_s^j = c_{s,n,v}^j$$

$$\overline{c_s} = \sum_{\widehat{J}} c_s^j$$

This includes:

c_s^j: the costs of path j

$\overline{c}_s = \frac{1}{|\widehat{J}|} \sum_{j \in \widehat{J}} c_s^j$: the average path costs, with $|\widehat{J}|$ the number of paths with the OD relation $\widehat{J}$.

The demand is shifted towards the vector $\vec{d}_s = \left(d_s^j\right)_{j \in \widehat{J}}$

with

$$d_s^j = c_s^j - \overline{c}_s$$

Due to the definition of $\overline{c}_s$ the following applies:

$$\sum_{p \in \widehat{P}} d_s^p = 0$$

The volume is thus redistributed an no additional volume generated.

In iteration s the proportion $\alpha_{tot} = \frac{1}{s_{it}}$ of the total demand for a parking lot relation is redistributed:

Where s_{it} is the content of **CurIterIdx** (**Current iteration index** attribute: index of the current iteration of an equilibrium assignment). The **CurIterIdx** index is incremented at the end of a simulation run, under the following conditions:

- A dynamic assignment has been carried out and matrices or trip chain files have been referenced, and
- the distribution model **Equilibrium assignment** has been selected.

CurIterIdx is saved to the path file **.weg*.

CurIterId is restored when a simulation run is started without a path file.

To redistribute only the desired share α_{tot} of the total volume, the vector d_s has yet to be scaled. For this purpose the scaled direction vector $\vec{n}_s$ is calculated.

$$n_s^{\mathrm{j}} = 2\,\frac{d_s^j}{\sum_{j\in\widehat{J}}|d_s^j|}$$

Thus the following conditions are met:

$$\sum_{j\in\widehat{J},d_s^j>0} n_s^j = 1$$

$$\sum_{j\in\widehat{J},d_s^j<0} n_s^j = -1$$

This means that, just as much volume is taken from paths that are more expensive than the average as is added to paths that are less expensive than the average.

Demand is shifted towards $\vec{n}_s$ so that no negative demand is created on any of the paths:

$$\alpha_{max} = \max_{j\in\widehat{J}}\left\{\frac{\widehat{f^j}}{n_s^j}\forall n_s^j < 0\right\}$$

If $\alpha_{max} = 0$ the algorithm implies that volume is taken from paths which have a volume of 0 already. To carry out the redistribution, these paths are temporarily taken from the set of paths, the OD pair. Volume balancing is restarted and only the temporarily reduced path set is taken into account.

If $\alpha_{max} < 0$ the following is set:

$$\alpha = \min\mathrm{x}\left(\alpha_{tot}, |\alpha_{max}|\right)$$

The new target volume f_s^j is then given by:

$$f_s^j = \widehat{f_{\mathrm{s\text{-}1}}^j} + \alpha n_s^j$$

Thus, a proportion of α in the total demand for the parking lot relation is shifted.

If $\alpha < \alpha_{tot}$ the remaining share of $\alpha_{tot} - \alpha$ must be shifted. The remaining share is redistributed iteratively. For this purpose, paths with a relative target volume $f_s^j = 0$ are temporarily removed from the path set J_{JJ} . The volume balancing is restarted, however with $\alpha_{tot} - \alpha$ instead of α_{tot} and with f_s^j instead of $\widehat{f_{\mathrm{s\text{-}1}}^j}$.

The iterative procedure is stopped when $\alpha_{tot} - \alpha = 0$ is reached.

The new final volume f_s^j is then saved and used for the new time interval, down to vehicle class level, in the new attribute **Target volume (relative)** (see "Attributes of paths" on page 752). **Target volume (relative)** is saved to the path file **.weg*.

If during assignment a path file is read in that does not contain the **Target volume (relative)** and **Current iteration index**, the following values are set:

- **Current iteration index**: 1
- **Target volume (relative)**: empty for all vehicle classes and time intervals

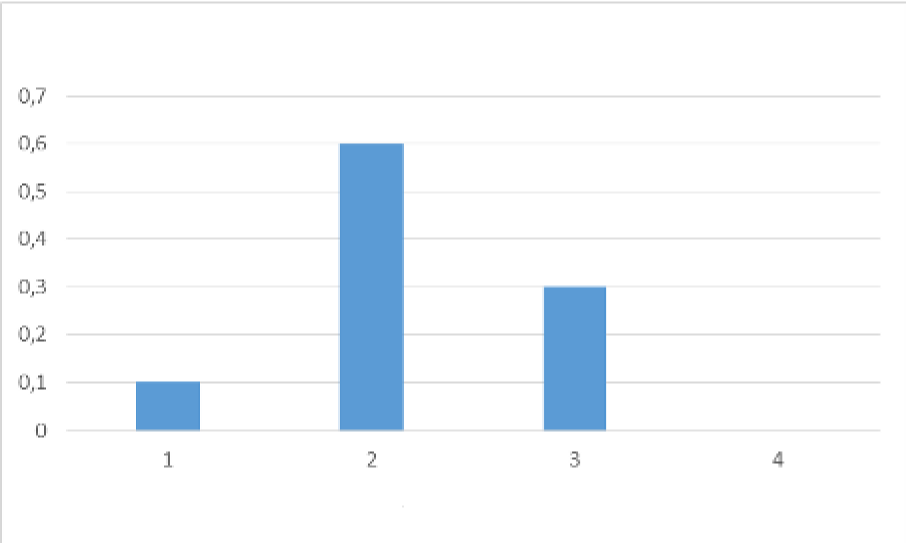
7.8.4 Equilibrium assignment – Example

The example explains the redistribution of relative target volumes in equilibrium assignment based on a single OD pair with only four paths.

Target volumes on the basis of previous simulation runs

The illustration below shows the relative target volumes on the basis of previous simulation runs with equilibrium assignment. Most volumes were distributed to path 2 in the last simulation run completed, followed by path 3 and path 1. Path 4 had no volumes in the previous simulation run.

Relative target volume of previous simulation runs path 1 to 4:

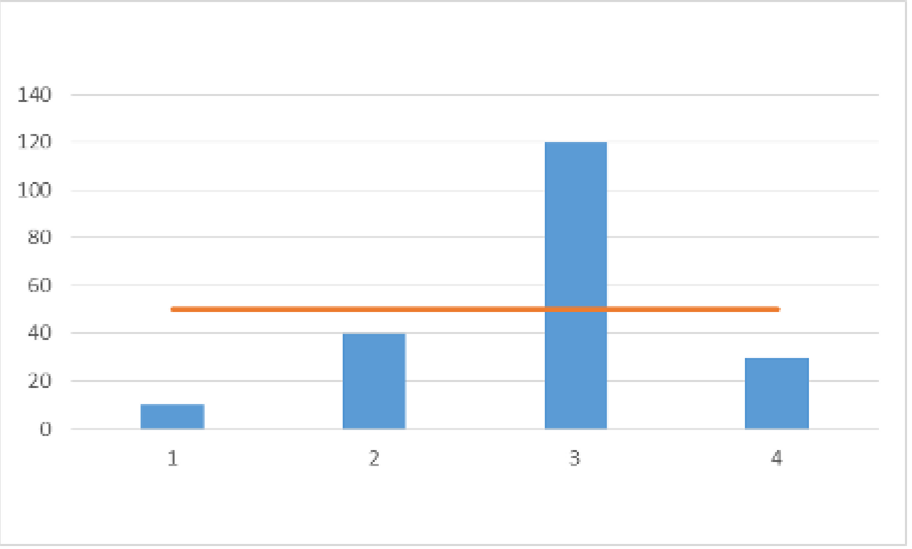


: Relative target volume before

Cost measured in previous simulation run

The illustration below shows the cost measured in the previous simulation run. Path 3 was by far the most expensive path and also the only path that was more expensive than the average cost. Path 2 was closest to the average cost. Path 4 was slightly less expensive and path 1 was the least expensive.

Cost of path 1 to 4:



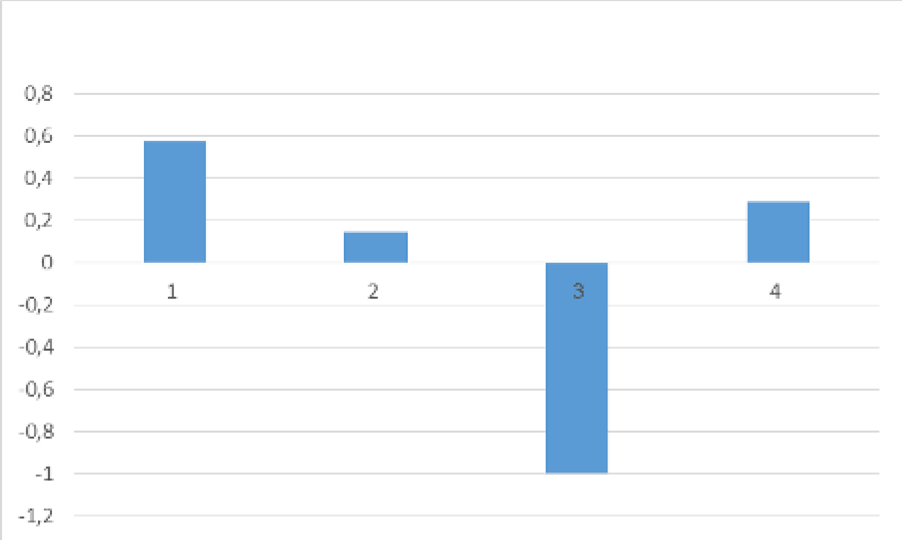
Cost

Average cost

Redistribution of the volume

The volume can be redistributed by means of this data (see "Method of path selection with or without path search" on page 741). From the calculation of the formulas it follows that a share of the total volume, which shall be redistributed, is distributed to the individual paths. This is shown by the illustration below. The entire share of the total volume, which shall be redistributed, is taken from path 3 because path 3 was the only path whose cost was higher than the average cost. The least volume is added to path 2 because the costs of path 2 were closest below the average cost. The most volume is added to path 1, because path 1 was the least expensive path.

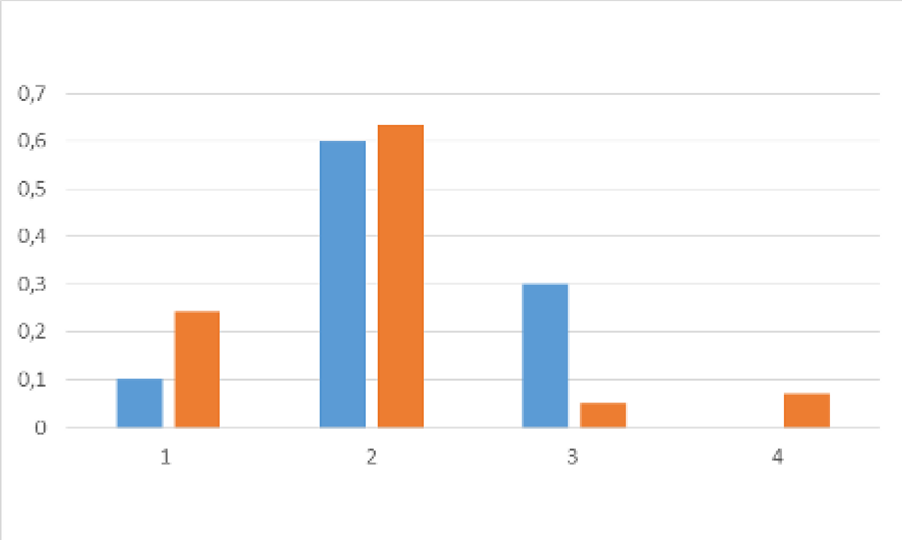
Redistribution path 1 to 4:



Result and comparison of the relative target volume before and after

If the last completed simulation run was the fourth simulation run of this assignment, as in the example, a quarter of the total volume must be redistributed for the OD pair according to the redistribution algorithm. The new target volumes are determined by adding a quarter of the values from the **Redistribution** illustration above to the values from the first illustration above **Relative target volume of previous simulation runs**. The following illustration shows the new relative target volumes. The relative target volume of path 3 has dropped considerably, while the target volume of path 1 has increased significantly. Path 4 now also has volumes. The relative target volume of path 2, by contrast, is nearly unchanged.

Comparison relative target volume path 1 to 4 before and after:



■: Relative target volume before, left bar

■: Relative target volume after, right bar

7.8.5 Performing an alternative path search

You can search via the following functions according to Alternative Path Search:

- Stochastic edge evaluation with maximum dispersion share for each OD relation between all zones and the number of passes
- Shortest path price increase for paths from zones, which you select
- Shortest path price increase for paths from dynamic routing decisions, which you select

Multiple passes of the shortest path algorithm with only slightly modified edge evaluations increase the probability of finding more Alternative Paths. These may have higher total costs than the best path, however, they should still be used.

7.8.5.1 Stochastic edge evaluation with maximum dispersion share

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Search** tab.
3. Select **Search alternative paths**.
4. Click the **Alternative path search** button.

   *The **Alternative path search** window opens.*

5. Make the desired changes:

Element	Description
Search alternative paths with stochastic edge penalization	☑ If the option is selected, the specified number from stochastic passes is carried out at the beginning of an evaluation interval of the dynamic assignment after each normal shortest path search. Before each pass, the calculation of each edge in the network is multiplied by a random factor between (1-x) and (1+x). x is the maximum dispersion share for each OD relation between zones, which you have entered into the **Spread** field.
Spread	Dispersion share for each OD relation between zones
Passes	Number of the stochastic passes

6. Confirm with **OK**.

7.8.5.2 Penalization of the shortest path per zone/OD pair

Vissim runs according to the normal shortest path search as long as additional passes with altered edge evaluations are running until a new path without a route closure is found or the specified maximum number of passes is reached (see "Influencing the path search by using cost surcharges or blocks" on page 787).

Thereby before each pass, the evaluations for all edges of the currently best path are multiplied with the edge cost penalization factor (**EdgCostPenFact**).

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Search** tab.
3. Select **Search alternative paths**.
4. Click the **Alternative path search** button.

   *The **Alternative path search** window opens.*

5. In the **Penalization of the shortest path per zone/OD pair** section, right-click in the row header.
6. From the shortcut menu, choose **Add**.

   The list of attributes opens.

7. Make the desired changes:

Element	Description
FromZone	**From zone**: Number of the origin zone
ToZone	**To zone**: Number of the destination zone
EdgeCostPenFact	**Edge cost penalization factor**
MaxNumPass	**Maximum number of passes**: Maximum number of stochastic passes

8. Confirm with **OK**.

7.8.5.3 Penalization of the shortest path per dynamic routing decision/OD pair

To use this function, you must have inserted dynamic routing decisions into the Vissim network (see "Defining dynamic routing decisions" on page 762). Vissim runs according to the normal shortest path search as long as additional passes with altered edge evaluations are running until a new path without a route closure is found or the specified maximum number of passes is reached (see "Influencing the path search by using cost surcharges or blocks" on page 787).

Before each search, the evaluation of each edge of the currently best path is multiplied with the value of the attribute **Edge costs evaluation factor**.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Search** tab.
3. Select **Search alternative paths**.
4. Click the **Alternative path search** button.

   *The **Alternative path search** window opens.*

5. In the **Penalization of the shortest path per dynamic routing decision/OD pair** section, right-click in the row header.
6. From the shortcut menu, choose **Add**.

   The list of attributes opens.

7. Make the desired changes:

Element	Description
FromRoutingDecision	**From routing decision**: Number of the origin routing decision
ToZone	**To zone**: Number of the destination zone
EdgeCostPenFact	**Edge cost penalization factor**
MaxNumPass	**Maximum number of passes**: Maximum number of stochastic passes

8. Confirm with **OK**.

7.8.6 Displaying paths in the network

You can mark each path in color that was found during the iterations of the dynamic assignment.

1. Select in the menu **Lists** > **Results** > **Paths**.

   *The list **Paths** opens.*

2. Open the network editor.
3. Ensure that the icon **Synchronization** is selected in the **Paths** list.
4. Click in the list on the desired path.
5. If you want to select additional paths, press the CTRL key and click the desired entries.

*The paths are displayed in yellow in the network display. If in the **Paths** list, synchronization is selected, detours are shown in red in the network display.*

Note: The paths result from the last iteration in which the path file was updated. The costs displayed are taken from the previously saved cost file. Hence the results of the last iteration are displayed only if the cost and path files were saved during that iteration.

7.8.7 Attributes of paths

1. Select in the menu **Lists** > **Results** > **Paths**.

   *The list **Paths** opens. When a *.weg file is saved to the directory specified in the parameters of dynamic assignment, in the **Files** tab, in the **Path file** box, it is automatically read in. Depending on the size of the file, this may take a while. A window opens, indicating the loading progress. You can also cancel this process.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Make the desired changes:

Long name	Short name	Description
Is detour	**IsDetour**	☑ If the option is selected and the option **Avoid long detours** is selected, the detours are displayed (see "Using the detour factor to avoid detours" on page 759). Paths are not displayed.
Converged	**Conv**	☑ If this option is selected, the travel time of the path is converged. The path fulfills the convergence criterion **Travel time on paths** for all completed time intervals (see "Attributes for achieving convergence" on page 782). The average journey time of all vehicles is taken into account. This corresponds to the weighted average journey times of all vehicle classes.
From parking lot	**FromParkLot**	Number of the origin parking lot
To parking lot	**ToParkLot**	Number of the destination parking lot
Volume (old)	**VolOld**	Number of vehicles that used the path saved to the path file (*.weg*) during the last simulation run. For blocked edges or blocked connectors in edges, the values are in parentheses. The volume is always output as the sum of all vehicle types. Therefore the value does not change if you select different vehicle types. In the column, vehicles are displayed which have paths that run through a connector, which are blocked only for specific vehicle classes and not for all classes. If the path file is generated via the ANM import, the volumes can also have decimal positions from the assignment with Visum.
Volume (new)	**VolNew**	Number of vehicles using the path during the current simulation run
Volume target (relative)	**VolTarRel**	only for equilibrium assignment: share of the target volume of the path in the total volume of the parking lot OD pair (see "Method of path selection with or without path search" on page 741)

In the network editor, you can display all paths which are found during the iterations of the dynamic assignment (see "Displaying paths in the network" on page 752).

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Edge sequence** (see "Attributes of edges" on page 718).
3. Enter the desired data.

The data is allocated.

7.9 Optional expansion for the dynamic assignment

Vissim allows for the following optional expansions:

- Simultaneous assignment: simultaneous assignment for different classes of road users
- Parking lot selection: find one out of multiple destination parking lots in a zone
- Detour recognition: recognize a detour factor for newly calculated legs, which lengthen newly found paths
- Correcting distorted demand distribution for overlapping paths
- Dynamic routing decisions: Arrange vehicles on a new path selection
- Route guidance: While driving, vehicles search for new paths to a destination parking lot.

7.9.1 Defining simultaneous assignment

You can perform an assignment for different classes of road users, who have an effect upon each other in the road network. Examples of user classes are commuters, business travelers, local drivers, out-of-town drivers etc. These classes differ in the following points:

- in their path selection behavior
- in their access to different parts of the road network

7.9.1.1 Defining path selection behavior with general cost function

To model different path selection behavior, in the attributes of the vehicle type, you can enter **Cost Coefficients** of the general cost function separately for each vehicle type.

- α for **travel time**
- β for**travel distance**
- γ for **link costs**

Thus you can model e.g. drivers who are willing to pay tolls to gain time, and other drivers that do not want to pay and accept longer paths in exchange.

1. From the **Base Data** menu, choose **Vehicle Types**.

   *The **Vehicle Types** list opens.*

2. Right-click the vehicle type of your choice.

   The context menu opens.

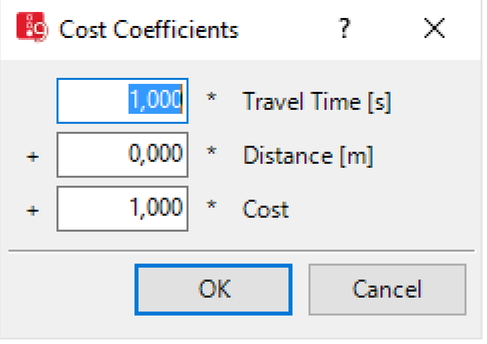
3. Select the **Edit** entry.

   The ***Vehicle Type*** *window opens.*

4. Select the **Special** tab.

5. Click the **Cost Coefficients** button.

   The ***Cost Coefficients*** *window opens.*



The summands must be of the same magnitude as the cost components. Pay attention to the units:

- Travel times in seconds
- Travel distances in meters
- Costs have no implicit unit. You define the unit. So if for example cost unit is **Euro per kilometer**, choose the corresponding coefficient so that the product amounts to the same order of magnitude as the travel time in seconds.

6. Enter the desired values.

7.9.1.2 Defining access to different parts of the road network

The second aspect of simultaneous assignment, besides the different cost sensitivity, is the selective usability of the road network. It allows you to restrict parts of the network for vehicle classes. It allows you to model, for example, that local drivers have a solid knowledge of the network, while out-of-town drivers know only the main road network.

For this connectors are used in Vissim. You can block a connector for vehicle classes. The path search will not use paths with blocked connectors for the vehicles from these vehicle classes (see "Modeling connectors" on page 420). The blocks for the selected vehicle classes are only relevant for the dynamic assignment.

7.9.2 Defining the destination parking lot selection

The traffic demand in the origin-destination matrices refer to the zones for origins and destinations. Zones are represented in Vissim via one or more parking lots. If more than one parking lot is located in the destination zone of a driver, the driver must choose the destination parking lot before choosing his/her path.

Thereby, the destination parking lot selection is an additional example of the problem class Discrete Selection (see "Differences between static and dynamic assignment" on page 694). The reason for this is there are many alternatives: a utility function and a decision function (see "Base for calculating the dynamic assignment" on page 695)(see "Calculation of paths and costs" on page 739). For the vehicle type used by the driver, you may enter a utility function coefficient for each of three decision situations that trigger a parking lot search (see "Entering coefficients for utility function of parking lot" on page 757). Calculation of the utility function is also based on the attributes **Parking fee** and **Attraction** of the parking lots (see "Attributes of parking lots" on page 500).

For destination parking lot selection a Logit formula is used (see "Method of path selection with or without path search" on page 741). The Logit forumula contains a scaling factor that you can specify (see "Defining the Logit function scaling factor" on page 759).

Destination parking lot selection depends on the following settings made in the parameters of dynamic assignment, on the **Cost** tab, in the section **Cost for path distribution with Kirchhoff**. This also applies for destination parking lot selection for dynamic routing decisions (see "Defining dynamic routing decisions" on page 762):

- If **Measured path travel times** is selected, destination parking lot selection uses generalized costs of paths from the second simulation run. For the first simulation run, the total of edge lengths is used, as there is still no data available on the travel time. From the start of the edge, the distance from the end of the origin parking lot to the first node border is taken into account. From the end of edge, only the distance from the last node border to the beginning of the destination parking lot is considered.
- If the **Sum of edge travel times** is selected, the entire length of all edges is used.

7.9.2.1 Definition of the utility function of a parking lot

Coefficient	Formula symbol	Description	Description
α	$C_{Parking}$	**Parking Cost**	Parking fee listed as **Parking fee** attribute of the parking lot
β	$Attraction$	**Attraction**	Attraction listed as **Attraction** attribute of the parking lot
γ	$D_{Destination}$	**Distance from desired zone**	Straight-line distance between the position of the parking lot and the center of the destination zone
δ	D_{Veh}	**Distance from current position**	Generalized costs of the cheapest path from the current location (vehicle position)

Coefficient	Formula symbol	Description	Description
ϵ	fs	**Current parking availability**	Availability of free parking spaces
	k		Index of Vehicle type
	s		Index of Decision Situation, Departure, Routing Decisions, RouteGuidance cycle 1 or RouteGuidance cycle 2

For the utility function the following applies:

- positive coefficient for **Attraction** and for **Current parking availability**
- negative coefficient for **Parking Cost**, **Distance from desired zone** and **Distance from current position**

7.9.2.2 Entering coefficients for utility function of parking lot

You may enter individual utility function coefficients for each vehicle type and each of the three following decision situations. Coefficients are weighting factors for the smallest possible generalized cost of a path, from the current vehcile position to the destination parking lot. You can show the coefficient values entered in the **Vehicle types** list, in the **GenCost** (**generalized costs**) attribute.

- **Departure from Parking lot**: when a vehicle begins its trip in the origin parking lot
- **Dynamic routing decision**: when a vehicle passes a dynamic routing decision
- **Route Guidance Tactic 1**, **Route Guidance Tactic 2**: when a vehicle with a route guidance system receives new information

The number of the permissible destination parking lots to the time of the departure is the number of parking lots which belong to the destination zone and are open at the time of the departure. For decisions, which are triggered by a dynamic routing decision or a route guidance system, the selected quantity depends on the set strategy (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

1. From the **Base Data** menu, choose **Vehicle Types**.

   *The **Vehicle Types** list opens.*

2. Right-click the vehicle type of your choice.
3. From the shortcut menu, choose **Edit**.

   *The **Vehicle Type** window opens.*

4. Select the **Special** tab.
5. Click the **Destination Parking Lot Selection** button.

*The **Destination Parking Lot Selection** window opens. In the next step, to select multiple entries, hold down the CTRL key.*

6. In the list, click the desired decision situation (**DecSituation**).
7. Into the list boxes, enter the desired coefficients.

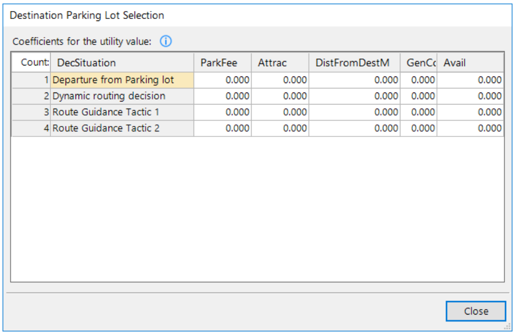
Note: After multiplication by the Logit scaling factor, the difference of the largest and smallest utility function value should be no larger than 4. If the difference of the utility value is 4, the probability of the selection of a better parking lot is 55 times as much as the probability that a worse parking lot will be selected.

Note: Select the negative coefficient for:

- Parking fee
- Distance from the destination
- Generalized cost

Note: Select the positive coefficient for:

- Attraction
- Availability



Warnings:

- For very large utility values, an overflow of numbers can occur.
- Very high negative values can lead to inaccuracy in the calculation because the percentage of **e** yields a value too close to zero.

In both cases, a warning is saved to the **.err* file and the **Messages** window.

When the use of all parking lots has been defined in the selected quantity, the selection probability is calculated with the Logit function.

8. Confirm with **OK**.

7.9.2.3 Calculation of parking spaces currently available

The ratio of available parking spaces of considered parking lots to the largest number of available spaces in the entire selection of parking lots equals the number of parking spaces currently available.

7.9.2.4 Distance from other destination zone

Although parking lots lie in zones, the information **distance from destination zone** is required, for example when all parking spaces of the destination zone are occupied. In such decision situations, parking lots that do not belong to the destination zone may become potential destination parking lots. Then Vissim shall choose a parking lot in a different zone as destination parking lot, preferably one which is close by.

The location of a zone is calculated from the mean of the coordinates of the parking lot of the zone and can be displayed as an attribute **Center** in the list of **Zones** (see "Traffic menu" on page 128).

7.9.2.5 Defining the Logit function scaling factor

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   The ***Dynamic Assignment: Parameters*** *window opens.*

2. Select the **Choice** tab.
3. In the section **Destination Parking Lot Selection**, enter the desired values:

Element	Description
Logit scaling factor	The Logit scaling factor μ determines how strongly the distribution responds to utility differences (see "Method of path selection with or without path search" on page 741).
Logit lower limit	If the selection probability of a parking lot lies below this value, the parking lot is not selected as destination parking lot. This increases the selection probability for the remaining parking lots.

7.9.3 Using the detour factor to avoid detours

In the iterations of the dynamic assignment the drivers are encouraged to try out new paths, which contain links or partial routes that have not yet been traveled (see "Path search finds only the best possible path in each interval" on page 740). This may lead to useless paths in the path collection. A path is considered useless if it is an obvious detour. An obvious detour is a path that can be generated from another, already known path by replacing a section by a much longer section. For this purpose, Vissim calculates the total length of nodes and turn relations, from the position in the node entrance or node exit where the paths differ to the position in the node entrance or node exit where the paths are the same again.

You can define how much longer the leg must be compared to the original leg to qualify as a useless path. For example, Vissim checks with a detour factor of 2 for all paths, whether they are just copies of other paths, in which a section has been replaced by a section which is more than twice as long.

If two paths only differ within a node where two parallel turn relations are used (one from each path), the detour factor is still not fulfilled, even if the two turn relations differ more in length than the detour factor. The paths must differ in at least one edge between two nodes for the detour to be recognized by the software.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*
2. Select the **Choice** tab.
3. In the **Path pre-selection** section select **Avoid long detours**.
4. Enter the desired value in the field **Avoid long detours**. The default value is 2.50.

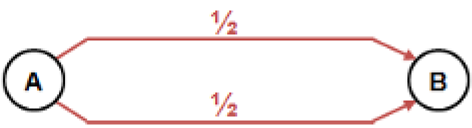
*From now the detour factor is taken into account in the iterations. Detours are shown in the **Paths** list without travel time or volume data (see "Attributes of paths" on page 752). If in the **Paths** list, synchronization is selected, detours are shown in red in the network display.*

7.9.4 Correcting distorted demand distribution for overlapping paths

For every origin-destination relation the whole traffic demand will be distributed to all available paths. The distribution considers the general path costs, calculated from the measured variables and the configured weighting coefficients. A path consists of a sequence of edges. Two paths are different if their sequences of edges are not exactly the same. Two paths may also be considered to be different if they differ only by a small section. In such a case both paths would have about the same weight in the distribution of the traffic volume. This would lead to an overall biased distribution. This problem occurs in all assignment tasks and is called the **blue/red bus paradox**. This is depicted in the following figures:

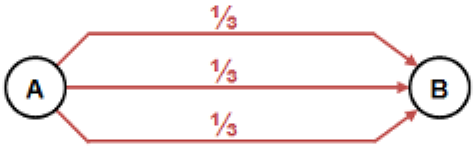
Case 1: Two paths with identical cost

The distribution of trips 50:50 is unproblematic:



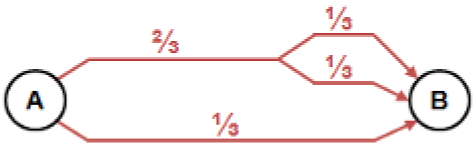
Case 2: Three paths with identical cost

The distribution of traffic on three paths is unproblematic. Each path receives one third of the demand:



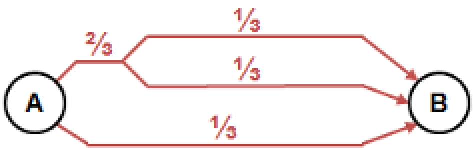
Case 3: A slight variation results in 3 optional paths

Problem: Actually, there are only two quite dissimilar paths. Because of the slight variation in the end, the path search finds three different paths. Result: It is distributed amongst three paths. The overlapping part of the two similar paths receives too much traffic.



Case 4: Common stretches leads to three possible paths

The opposite of case 3: Actually there are 3 different paths but two of them have a small stretch in common. As in case 3, every path gets about one third of the demand. This is much more realistic compared to case 3.



7.9.4.1 Selecting the correction

You can correct the biased distribution of overlapping paths. Thus, the path selection model calculates a degree of commonality for the paths (commonality factor) The commonality factor expresses how much of a path is shared with other paths:

- Higher value: A path has many edges in common with other paths.
- Lower value: A path is largely independent from other paths.

Using this value the distribution function reduces the selection probability of paths with high commonality factor.

Notes: In certain network constellations, the correction of the biased distribution tends to spread traffic over longer paths if these paths have little in common with other paths. This can lead to unexpected results.

In general, the correction of a biased distribution improves the result of the assignment. Use the correction of a biased distribution only in combination with restricting the cost difference between the paths.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Choice** tab.
3. In the **Path choice model** section, select **Correction of overlapping paths**.

7.9.5 Defining dynamic routing decisions

Vehicles that are routed by the dynamic assignment ignore all static routing decisions along their way. With dynamic routing decisions you can trigger a new path selection for these vehicles.

On a dynamic routing decision, a vehicle decides if a specific condition has been fulfilled, for example, if its destination parking lot is full. If the condition is fulfilled, a new parking lot selection and a new path selection are carried out according to the given strategy. The strategy also determines the parking lots which are available for selection.

Path selection for dynamic routing decisions uses the same generalized costs of the edges as path selection of dynamic assignment in the current evaluation interval.

Destination parking lot selection depends on the following settings made in the parameters of dynamic assignment, on the **Cost** tab, in the section **Cost for path distribution with Kirchhoff**:

- If **Measured path travel times** is selected, destination parking lot selection uses generalized costs of paths from the second simulation run. For the first simulation run, the total of edge lengths is used, as there is still no data available on the travel time. From the start of the edge, the distance from the end of the origin parking lot to the first node border is taken into account. From the end of edge, only the distance from the last node border to the beginning of the destination parking lot is considered.
- If the **Sum of edge travel times** is selected, the entire length of all edges is used.

Note: Do not position the dynamic routing decision on an edge which contains a parking lot that cannot be passed.

1. In the Network objects toolbar, click the **Vehicle Routes** button.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button ▾ is shown.

2. Re-click the button **Vehicle Routes**.

   A list box with the variants of the network object type opens.

3. Select the entry **Dynamic**.

4. In the Network Editor, right-click the desired position of the Routing decision section in the desired link or connector.
5. From the context menu, choose **Add New Dynamic Vehicle Routing Decision**.

   A red purple is shown at that position.
6. Edit the attributes (see "Attributes of dynamic routing decisions" on page 763).

7.9.6 Attributes of dynamic routing decisions

- In the **Lists** menu, select **Private Transport** > **Routes** > **Dynamic Routing Decisions**.

  The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number of dynamic routing decision
Name	Name of dynamic routing decision
Link	Number of the link, on which the dynamic routing decision is located
Pos	Distance of dynamic routing decision to the beginning of link or connector
AllVehTypes	☑ If the option is selected, all vehicle types follow the vehicle routing decision or partial routing decision The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	Valid **vehicle classes**
Condition	Condition for new parking lot selection and new path selection: - **Destination is zone of parking lot** - **Destination parking lot full** - **Destination parking lot fewer than**: accounts for the number of free parking spaces - **Parking lot full** - **Destination parking lot fewer than**: accounts for the number of free parking spaces - **Always**
CondParkLot	**Condition parking lot**: Name of parking lot specified in the attribute **Destination is zone of parking lot**.

Column	Description
AvailSpaces	**Available spaces**: Enter number of free parking spaces for the conditions **Destination parking lot fewer than** and **Parking lot fewer than**
Strategy	If the condition has been fulfilled: strategy according to which the new path selection is performed: - Same zone, different parking lot (random): Parking lot is selected evenly from all the parking lots of the zone. - Same zone, parking lot by benefit function - Same zone, different parking lot by benefit function - Any zone, parking lot by benefit function - Any zone, different parking lot by benefit function - New route to destination parking lot - Specified parking lot With the strategies for benefit, parking lot search is based on a choice model (see "Defining the destination parking lot selection" on page 756) Closed parking lots are not taken into account.
StratParkLot	**Strategy parking lot**: Name of parking lot specified in the **Strategy** attribute.
ExclFullParkLots	Exclude full parking lots: Full parking lots are not considered for route choice

Showing and editing dependent objects as relation

The list on the right contains attributes and attribute values of network objects allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119).

Among others, these may include the following attributes:

- Vehicle classes (see "Defining the vehicle class" on page 280)
- Paths: Paths of path file from dynamic assignment If no paths are shown and you have performed dynamic assignment, you can read in the paths from their path file. To do so, read in the paths via the **Paths** list shortcut menu (see "Attributes of paths" on page 752).

1. On the list toolbar, in the **Relations** list, click the desired entry.

2. Enter the desired data.

The data is allocated.

7.9.7 Defining route guidance for vehicles

With the route guidance you can make the vehicles search for new paths to a destination parking lot while driving. This allows you to model the mode of action of the route guidance systems.

By default, in the dynamic assignment the vehicles choose the path to their destination parking lots at the moment of departure from the origin parking lot. Thereby the path selection is based on the general cost from the preceding iterations of the simulation (see "Method of path selection with or without path search" on page 741). In the route guidance, the path selection is based on the current traffic situation in the current simulation iteration. The travel time from the last completed time interval is therefore used.

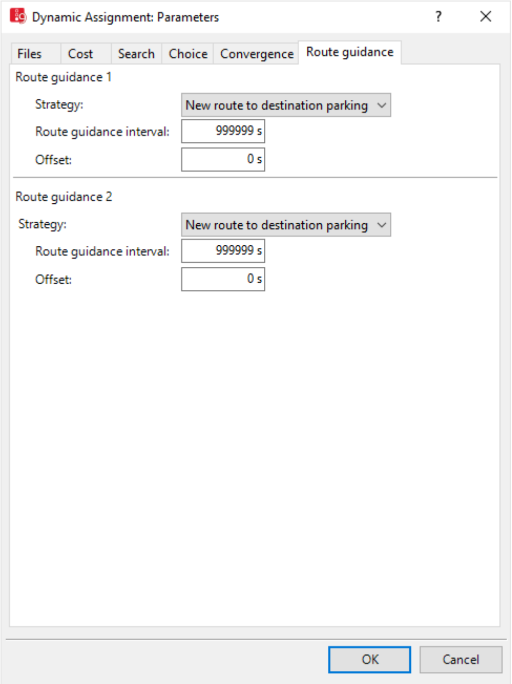
While you need to place dynamic routing decisions at a desired position in the network, the vehicles with route guidance search for new paths with a route guidance interval from their current position. You set the **Route guidance interval**. Thereby the route guidance triggers the search for the best path to the destination parking lot.

You assign the route guidance system to vehicle types (see "Using vehicle types" on page 267).

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Route guidance** tab.



3. Make the desired changes:

Element	Description
Route guidance 1 **Route guidance 2**	You can model two different route guidance systems. For each route guidance system, you can set the strategy, the route guidance interval and the offset time.
Strategy	Select destination: - **New route to destination parking lot** - **New destination parking lot in same zone**
Route guidance interval	Time interval for the new path search in seconds
Offset	Duration of processing times and run times of messages in real route guidance systems. This can cause the travel times of the route guidance to deviate from the travel times of the current evaluation interval.

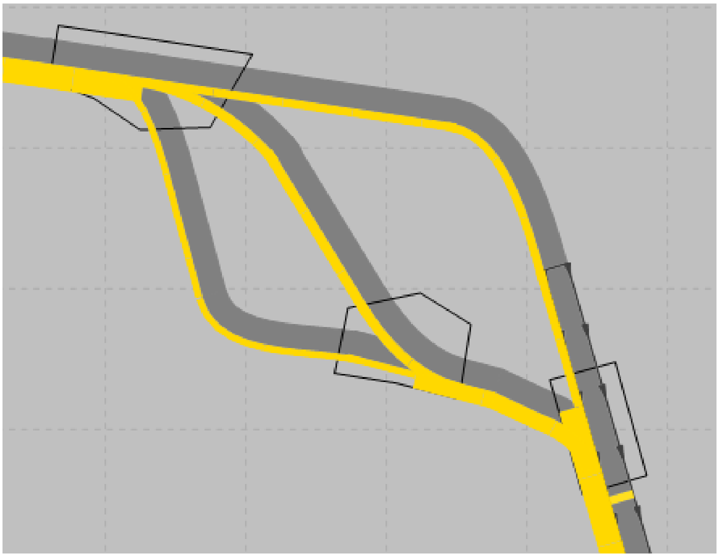
4. Confirm with **OK**.

7.10 Visualizing volumes on paths as flow bundles

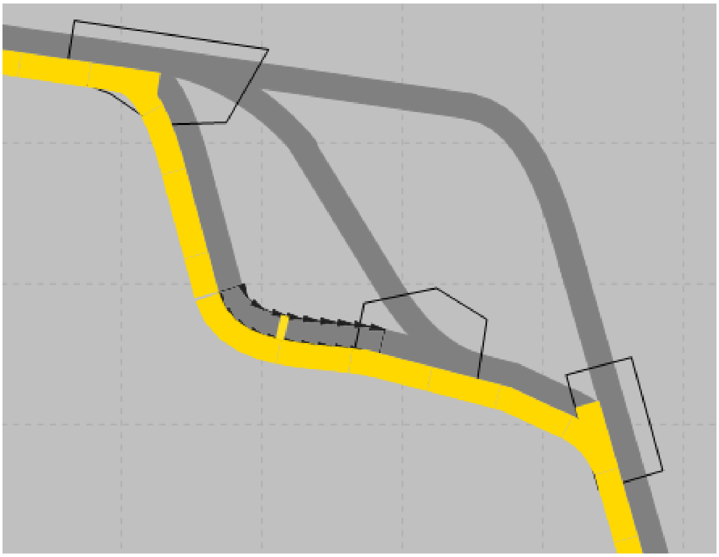
You can visualize the volume on paths generated through dynamic assignment as a flow bundle. To do so, in the Network editor, add filter cross sections of the network object **Flow bundles** to the links or connectors of the paths whose volume you want visualize (see " Defining flow bundles and filter cross sections" on page 768). The flow bundle visualizes the volumes of the last simulation run.

Examples

The figure shows a section of a Vissim network, with path volumes visualized as a flow bundle. The yellow filter cross section of the flow bundle has been placed and selected at the bottom right of the node. Parallel to it, the path volumes on the link are visualized as yellow flow bundle bars. The width varies depending on the path volume.



The figure below shows the same section. This time, the filter cross section of the flow bundle has been placed and selected on the left of the connector. There is only one path that traverses this connector.



Additional options to visualize evaluation results

- Color schemes for links (see "Assigning a color to nodes based on an attribute" on page 191)

- Link bars (see "List of graphic parameters for network objects" on page 161), (see "Attributes of links" on page 409)
- Turn value visualization (see "Visualizing turn values" on page 685)

7.10.1 Defining flow bundles and filter cross sections

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. Ensure that for the Vissim network, at least one simulation run for a dynamic assignment has been completed for which is a path file available.
2. On the Network objects toolbar, click **Flow bundles**.
3. Hold down the CTRL key and in the Network editor, right-click the link or connector for which you want to visualize the volume of its entire path.

   *By default, a yellow **filter cross section** is then inserted and selected at this position. By default, parallel to the path, yellow flow bundle bars depict the volume. The width of the flow bundle bar varies depending on the volume. You can show and hide the flow bundle bars.*
4. To insert additional filter cross sections for this flow bundle, repeat the last step.
5. After having inserted the desired number of filter cross sections, in the Network editor, click an empty space.
6. Release the keys.

The **Flow bundles** list opens if automatic opening of a list after object generation has been selected (see "Right-click behavior and action after creating an object" on page 152). The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

7. Edit the attributes (see "Flow bundle attributes" on page 768).

*The attributes are saved to the **Flow bundles** list.*

7.10.2 Flow bundle attributes

1. From the **Lists** menu, choose **Measurements** > **Flow bundles**.

   *The **Flow bundles** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. Make the desired changes:

Short name	Long name	Description
No.	**Number**	Unique number of the flow bundle
Name	**Name**	Name of flow bundle
Cross section relevance	**CroSecRelev**	Method how the relevant paths for the flow bundle are determined from the filter cross sections, if multiple filter cross sections are inserted: ▸ **All**:The path must traverse all filter cross-sections of the flow bundle selected in the **Flow bundles** list for it to be considered for the flow bundle. ▸ **At leas one**:The path must traverse at least one filter cross-section of the flow bundle selected in the **Flow bundles** list for it to be considered for the flow bundle.
AllVehTypes	**All vehicle types**	☑ If this option is selected, the volume of flow bundles is calculated for all vehicle types. The option **All Vehicle Types** is a virtual vehicle class that automatically includes all new vehicle types and vehicle types that have not been assigned a vehicle class yet.
VehClasses	**Vehicle class**	Vehicle classn as an additional filter for which the volume of the flow bundle is calculated. Make the following settings for the simulation run during which the flow bundle relevant path file is generated: ▸ The only vehicle classes displayed are those selected in the parameters of dynamic assignment, on the **Files** tab, under **Vehicle classes**. ▸ Select **Store paths (and volumes)**.
TmInts	**Time intervals**	Beginning and end of the intervals in simulation seconds (see "Defining time distributions" on page 246). The time intervals refer to the start time of the counted vehicles.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Filter cross sections**:
  - **Link**: Number of the link or connector on which the filter cross section is positioned
  - **Pos**: Distance from the beginning of the link or connector to the position of the filter cross section
- **Paths** (see "Attributes of paths" on page 752)
- **Vehicle classes**

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

7.10.3 Show flow bundle bars

To show flow bundle bars, ensure that the following requirements are met:

- At least one filter cross section must be defined for a link or connector (see " Defining flow bundles and filter cross sections" on page 768).
- The graphic parameter **Show flow bundle bars** of the network object flow bundle must be selected (see "Graphic parameters for visualizing the volume on paths with flow bundles" on page 171).
- For each link at which flow bundles are shown, select the **Link evaluation active** attribute.
- The **Segment length:** attribute of the link defines the subdivision of the link into segments with their own flow bundle volumes. If you change the segment length, you must update the flow bundle volumes. To do so, read in the path file again:
  - From the **Traffic** menu, choose > **Dynamic Assignment** > **Read Paths**.

Selecting a flow bundle or filter cross-section

You have the following options to select a flow bundle or filter cross sections and show the corresponding flow bundle bars:

- Select filter cross-section in the Network editor
- Select filter cross-section in the Attribute list:
  - In the Attribute list **Flow bundles** in the **Relations** list, click **Filter cross sections**.
  - Make sure synchronization for the Relations list **Filter cross sections** is selected.
- Select flow bundle in Attribute list: Make sure synchronization for the **Flow bundles** list is selected.

Defining the display of flow bundle bars

By default, the display of flow bundle bars is based on the graphic parameters of the flow bundle.

Alternatively, you can display flow bundle bars based on a color scheme for link bars:

1. Open the graphic parameters for links (see "Editing graphic parameters for network objects" on page 158).
2. For the graphic parameter Link bar drawing mode, select **Link bars only** or **Links and link bars**.
3. Click **Link bar configuration**.

   *The **Edit Link Bar** window opens.*
4. As **Link bar drawing mode**, select **Segment-based** is selected.
5. Depending on your display requirements, under the **Flow bundle volume** attribute, select the entry of your choice for **Classification by color**.
6. In the **Class bounds and colors** section, specfiy the desired color scheme.

7.11 Controlling dynamic assignment

You must parameterize the dynamic assignment and the files for the control and for saving. In doing so, you e.g. specify parameters for costs, path search and selection as well as convergence criteria.

You can also define the Alternative Path Search and simulate the effectiveness of route guidance systems.

Resetting dynamic assignment

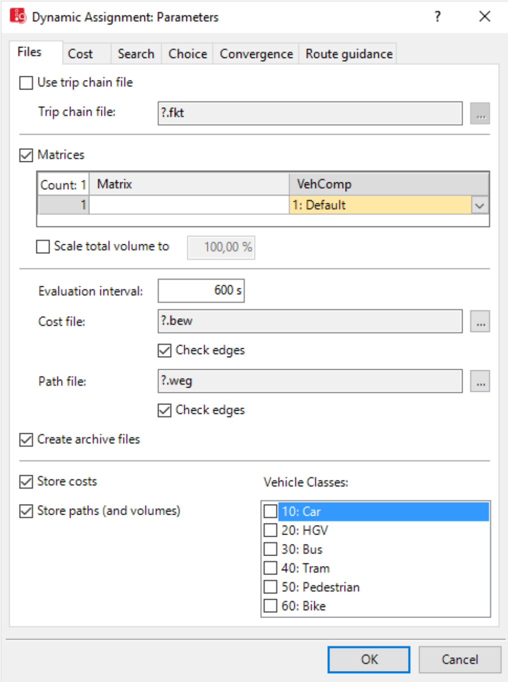
When you perform simulation runs during dynamic assignment, Vissim saves the path file **.weg* and the cost file **.bew* together with the results of dynamic assignment. The path file **.weg* and cost file **.bew* can have an impact on the dynamic assignment of subsequent simulation runs. To reset dynamic assignment, delete the path file **.weg* and cost file **.bew*. Then carry out the simulation runs again without any impact of the results obtained until then on dynamic assignment.

Note: When you delete the path file **.weg* and cost file **.bew*, the results of dynamic assignment from previous simulation runs are lost. If you need these results, e.g. for a comparison with the results of other simulation runs, select **Create archive files**. Vissim then saves the path file **.weg* and evaluation file **.bew* under new files names, when you run additional simulations (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

7.11.1 Attributes for the trip chain file, matrices, path file and cost file

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*



2. Select the **Files** tab.
3. Make the desired changes:

Element	Description
Use trip chain file	If this option is selected, Vissim performs dynamic assignment based on a trip chain file **.fkt* (see "Modeling traffic demand with trip chain files" on page 730)
Trip chain file	**.fkt* file with trip chains from a sequence of trips

Element	Description
Matrices	☑ If this option is selected, Vissim performs dynamic assignment based on one or multiple OD matrices. ▶ **Matrix** column: Select number and name of origin-destination matrix (see "Modeling traffic demand with origin-destination matrices" on page 721) ▶ **VehComp** column: Selecting vehicle composition (see "Modeling vehicle compositions" on page 452)
Scale total volume to	Reduce total demand in all origin-destination matrices to the percentage specified in [%] This value is used as a starting point for scaling the total demand of origin-destination matrices by the value **Scale total volume to** (see "Defining simulation parameters" on page 840). This starting point value of the reduced total demand increases with each simulation run defined in the simulation parameters (in the **Number of runs** box) by the **Dynamic assignment volume increment** until 100 % of the total demand has been reached (see "Defining simulation parameters" on page 840).

Element	Description
Evaluation interval	Time interval in which costs are calculated and paths are searched. The path selection of vehicles can change at these intervals. Can be changed, as long as no **.bew* file has been created (see "Evaluation interval duration needed to determine the travel times" on page 734).
Cost file	File with a current list of the costs for the current paths, plus edge times and volumes of the edges of the abstract network graph from the two previous simulation runs (see "Files of dynamic assignment" on page 1195) ▶ **Check edges**: If you edit the Vissim network, numbers of links and edges may change. ☑ If the option has been selected, when reading paths, Vissim checks if the edges from the cost file match the edges in the open Vissim network and tries to replace the edges from the cost file with the current edges in the Vissim network. The check criteria are described in this table at the bottom of section **Check criteria for the cost file and path file**.
Path file	File with the path archive (see "Files of dynamic assignment" on page 1195) ▶ **Check edges**: If you edit the Vissim network, numbers of links and edges may change. ☑ Select this option to have Vissim check, based on the following criteria, if the edges from the path file match the edges in the open Vissim network and try to replace the edges from the path file with the current edges in the Vissim network.

Element	Description
	Vissim criteria to be checked for the cost file and path file: - Vehicle class or vehicle type not available - Parking lot not available - Dynamic dynamic routing decision not available - Edge not available - Table **Linksanzahl** does not contain the origin parking lot - Table **Linksanzahl** does not contain the destination parking lot - Attribute list **Edges**: attribute **Link sequence** has no values - Attribute list **Edges**: contains nodes that do not exist in the path file or cost file - Time interval not available - Values cannot be read because they contain letters instead of numbers - Not enough entries in a row - The origin parking lot is not located on the first edge. - The destination parking lot is not located on the last edge. - The edges of a path are not connected.

Create archive files	If this option is selected:
	- Before the start of the simulation, Vissim checks whether the following output files are saved in the directory in which the network file **.inpx* is saved:
	  - Path file **.weg*
	  - Cost file **.bew*
	- Before new output files are generated, Vissim renames the existing output files **.bew* and **.weg*. When doing so, Vissim adds an underscore and the three-digit number of the simulation run to the file name, for example, *Routing_001.weg*. If the first simulation run is stared without a path file, there will not be a path file *Routing_001.bew*.
	- Based on the archived files, the changes during dynamic assignment remain traceable. If necessary, you can reset a previous result of an assignment. To do so, replace both the current path file **.weg* and the current evaluation file **.bew* with a version of these files that contains the desired, previous results you want to use to run a new simulation. Rename the files, so that Vissim can access them.
	- For a simulation with multiple runs, the system checks whether the path file and evaluation file exist before the start of the first simulation and for every other simulation run (see "Defining simulation parameters" on page 840).
	- If multiple simulation runs have been defined for the simulation

Element	Description
	parameter **Number of runs**, Vissim adds **.bew* and **.weg* files between the individual simulation runs. ☐ If this option is not selected and in the simulation parameters, in the **Number of runs** attribute, you have chosen to perform only one simulation run, the existing **.bew* and **.weg* files are overwritten when the simulation run is performed. If multiple simulation runs have been defined for the simulation parameter **Number of runs**, Vissim adds the **.bew* and **.weg* files after the last simulation run and not in between them.

Element	Description
Store costs	☑ If this option is selected, the specified file **.bew* is saved. You may select **Vehicle classes** for which additional vehicle class data, namely travel times and volumes, are saved to the evaluation file. You can access these data via the edge attributes and path attributes **Travel time (old)**, **Travel time (new)**, **Volume (old)** and **Volume (new)**. Then select the subattributes **Time Interval** and **Vehicle Class**. The subattribute **Vehicle Class** is based on the vehicle classes selected. For each vehicle with a vehicle type that belongs to a selected vehicle class, vehicle class specific values are used for path search and path selection. Combined with the option **Use volume (old)** (in the **Choice** tab), class-independent assignment results of an **.anmroutes* file, e.g. exported from PTV Visum, can be reproduced with only one stochastic deviation. Travel times are smoothed based on the **smoothing method** selected in the **Cost** tab.
Store paths (and volumes)	☑ If this option is selected, paths and volumes are saved in the **.weg* path file. Next to it, In the **Vehicle Classes** section, you can select vehicle classes for which additional data, apart from the data for all vehicle classes, is saved to the path file.

4. Confirm with **OK**.

7.11.2 Attributes for calculating costs as a basis for path selection

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Cost** tab.
3. Make the desired changes:

Element	Description
Time interval for edge travel times in path search	Time interval of which the smoothed edge travel times shall be used: - **Same time interval, previous simulation run**: Select this option for equilibrium assignment or for stochastic assignment (according to Kirchhoff). Always select this option when using equilibrium assignment (see "Attributes for path selection" on page 779). - **Previous time interval, same simulation run**: May result in less iterations to find paths in dynamic assignment and can thus speed up dynamic assignment. Vehicles respond faster to temporary queues. Do **not** select this option when using equilibrium assignment.

Element	Description
Cost for path distribution with Kirchhoff	Calculation procedure used as a basis to calculate costs for path selection if you have selected **Stochastic assignment (Kirchhoff)** or **Equilibrium assignment** for the distribution model. Existing volumes **Volume (old)** are not taken into account: - **Sum of edge travel times**: The cost calculation is based on the smoothed travel times of the edges. - **Measured path travel times**: The cost calculation is based on the smoothed travel times of the paths. If a path has no travel times yet, the sum of the edge travel times is used for this path. For links with multiple lanes, the **Measured path travel times** option produces more accurate results than the **Sum of edge travel times** option: In both cases, Vissim determines the average travel times on the edges of the paths. However, the average edge travel time is less significant when travel times on an edge before a node strongly differ for different following edges, e.g. because right turn traffic in the node is congested but straight-on traffic is flowing. Therefore, Vissim does not use the sum of the average edge travel times when using option **Measured path travel times**, but the smoothed total travel time of all vehicles that reach the destination parking lot if at least one path travel time is available from a previous simulation run. In an overloaded network, it makes sense to use measured path travel times only, as the total of estimated edge travel times in congested areas may result in completely unrealistic total times.

Element	Description
Smoothing method	Selecting smoothing method for travel times: - **Exponential smoothing with smoothing factor**: Value range 0.01 to 1, 1.0 = no smoothing, default value 0.20. If on the **Choice** tab, for **Path choice model**, you selected **Equilibrium assignment**, enter `1.0` (see "Selecting exponential smoothing of the travel times" on page 735), (see "Attributes for path selection" on page 779). - **MSA-method (method of successive averages), so far**: If a dynamic assignment has been performed and as a result the evaluation file **.bew* is saved, enter the number of iterations used to create the **.bew* file. Default 1 (see "Selecting the MSA method for travel times" on page 736) In the **Files** tab, the **Store costs** option must be selected. If the **Write evaluations** option is not selected, the smoothing of travel times is not performed. Instead, for each simulation run, the evaluations listed in the evaluation file **.bew* are used. The smoothing method is not relevant if you have selected equilibrium assignment (see "Attributes for path selection" on page 779).

4. Confirm with **OK**.

7.11.3 Attributes for path search

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Search** tab.
3. Make the desired changes:

Element	Description
Path search	On the **Choice** tab, in the **Path choice model** section, select the **Stochastic assignment (Kirchhoff)** option or **Equilibrium assignment**. - **Search new paths**: Before path selection, Vissim performs a path search (see "Path search finds only the best possible path in each interval" on page 740). - **Search alternative paths** (see "Performing an alternative path search" on page 749) - **Search path for O-D pairs with zero volume**: A parking lot-parking lot relation is without volumes, if none of the OD matrices contain a demand for this relation. ☑ If this option is selected, all paths for O-D pairs with zero volume will remain unchanged when starting the simulation. If you have also selected the **Search new paths** option, paths for O-D pairs with zero volume and new paths will be searched for. ☐ If this option is deselected, all paths for O-D pairs with zero volume will be discarded upon simulation start. The option will be ignored, if in the **Choice** tab, in the **Distribution model for paths**, the **Use volume (old)** option is selected. No paths will be searched for. All paths will remain unchanged upon simulation start. If in the **Path selection type** section, the **Decide repeatedly** option is selected, no new paths for O-D pairs with zero volume will be searched for.

Element	Description
Path selection type	If there are several paths a vehicle can use to get from the origin parking lot to the destination parking lot, the vehicle can decide on the path it wants to take at the following times: - **Decide at start only**: The vehicle chooses its path to the destination parking lot at the origin parking lot or at a dynamic vehicle routing decision. - **Decide repeatedly**: On the **Choice** tab, in the **Path choice model** section, the option **Stochastic assignment (Kirchhoff)** or **Equilibrium assignment** must be selected. The vehicle chooses its path dynamically while on the road. Before a vehicle reaches a node, Vissim determines the best route to the vehicle's destination parking lot for each turn relation of the node. Based on the Kirchhoff model for generalized costs, the vehicle randomly chooses one of these paths. These costs are aggregated if the path follows a toll route and is therefore a combined path. This procedure is repeated at every node, until the vehicle has reached its destination parking lot. When it comes to lane change, the vehicle accounts for the driving behavior attribute **Look ahead distance** and at least the next three nodes. As soon as the vehicle has reached its destination parking lot, the path used is saved to the path collection. Select this option, and in the **Costs** tab, click **Previous time interval, same simulation run** to perform dynamic assignment with a single simulation run, without any additional iterations. In this simulation run, all possible paths (without loops) can theoretically be used.

4. Confirm with **OK**.

7.11.4 Attributes for path selection

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Choice** tab.
3. Make the desired changes:

Element	Description
Destination parking lot selection	
Logit scaling factor	Sensitivity parameter of the Logit function for the parking lot selection model
Logit lower limit	Lowest selection probability which is still taken into account when choosing a parking lot. Parking lots with lower selection probability are not selected.

Element	Description
Path pre-selection	
Reject paths with too high total cost	When this option is selected, the **Max. extra cost compared to the best path:** box is selected. Vehicles avoid paths identified as too expensive Paths are too expensive if the generalized costs exceed the costs of the best path of the corresponding parking lot relation by the specified factor (in %).
Max. extra cost compared to the best path	Threshold in % for the cost difference as a criterion for path selection.
Limit number of paths	If this option is selected, the option below it **Max. number of paths per parking lot relation:** is activated. Vehicles consider the specified number of paths at most for a parking lot relation. This attribute is only effective during dynamic assignment and not when you read paths (see "Setting volume for paths manually" on page 786).
Max. number of paths per parking lot relation	Maximum number of paths per parking lot relation as a criterion for path selection. Default 999.
Avoid long detours	If the option has been selected, vehicles avoid paths that they identify as detours. A path is a detour if it replaces a section of an existing path with a section which is longer by at least the specified factor (see "Using the detour factor to avoid detours" on page 759). The option **Stochastic assignment (Kirchhoff)** or **Equilibrium assignment** must be selected.

4. In the **Path choice model** section, select the desired procedure:

Element	Description
Use volume (old)	The path is selected without a path search for new paths being carried out. The probability of a path being used corresponds to its attribute value **Volume (old)** share in the total of attribute values **Volume (old)**, of all paths of the same parking lot relation. This volume data for all intervals is read in from the Path file at the beginning of each simulation run. It is then moved to the **Paths** list, to the **Volume (old)** attribute. With the **Volume (new)** attribute for paths, the vehicles are counted that use the respective path during the simulation run(see "Attributes of paths" on page 752). You can set the volume to the paths of a time interval that has not yet started during the simulation interval or to a time interval prior to the simulation run (see "Setting volume for paths manually" on page 786). If ANM routes are imported as paths for dynamic assignment, the ANM import function automatically selects the attribute **Use volume (old)**.
Stochastic assignment (Kirchhoff)	The path is selected based on path search for new paths and an evaluation of the costs according to Kirchhoff. In the **Costs** tab, select the calculation procedure used as a basis to calculate the costs for path selection according to Kirchhoff. - **Kirchhoff exponent:**: Sensitivity parameter of the distribution function for path selection model. Default 3.50. - **Correction of overlapping paths**: If this option is selected, it activates the correction of the distribution of traffic on paths which have legs in common (see "Correcting distorted demand distribution for overlapping paths" on page 760). The attribute **Kirchhoff (edge travel times)** must be selected.

Element	Description
Equilibrium assignment	Redistributes demand across paths proportionally to costs, from expensive to inexpensive paths, for each OD pair in each iteration: The volume of paths that are more expensive than the average is reduced. The amount of the transferred volume is proportional to the additional costs of the path. All paths that are cheaper than the average path costs are assigned additional volume. The additional volume is proportional to the cost saving compared with the average path costs (see "Method of path selection with or without path search" on page 741). The result of the assignment are identical generalized costs on all paths of a parking lot OD pair, possibly with very different volumes, dependent on the respective capacity of the path. ➤ **Current Iteration**: Number of simulation runs performed ▸ On the **Cost** tab, in the section **Time interval for edge travel times in path search**, select **Same time interval, previous simulation run** (see "Attributes for calculating costs as a basis for path selection" on page 775). If you select **Equilibrium assignment**, the smoothing method is not relevant (see "Attributes for calculating costs as a basis for path selection" on page 775). Vissim then uses the smoothing factor `1.0`. This means no smoothing.is performed. ▸ On the **Search** tab, below **Path selection type**, select **Decide at start only**. Ensure that the option **Decide repeatedly** is deselected (see "Attributes for path search" on page 777).

5. Confirm with **OK**.

7.11.5 Attributes for achieving convergence

The iteration of the simulation for the calculation of the dynamic assignment can be ended once a stable state has been reached. This is the case when from one iteration to the next the travel time and corresponding congestion on the edge no longer changes significantly. This stability must be reached for all evaluation intervals. The situation does not change within one evaluation interval from iteration to iteration. The situation can, however, change within an iteration from one interval to the next because the demand and the network control is time dependent.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Convergence** tab.
3. Make the desired changes:

Element	Description
Travel time on paths	If the option is selected, the change of travel time for all paths is considered in comparison to the previous simulation iteration. This convergence criterion is fulfilled if the percentage change for all paths in all evaluation intervals is smaller than the specified threshold. Default value of 15 %. **Required share of converged paths**: weakens the convergence condition. Share of the paths in % which must at least have converged for the convergence criterion to be fulfilled. Default value 95 %, default value when loading network files which do not contain this attribute yet: 100 %. To exclude the impact of particularly short paths, slightly reduce the value of this attribute to approx. 90 to 95 percent.
Travel time of edges	If the option is selected, the change of the travel time for all edges is considered in comparison to the previous simulation iteration. This convergence criterion is fulfilled if the percentage change for all edges in all evaluation intervals is smaller than the specified threshold. Default value of 15 %. **Not for edges shorter than**: Length in meters is a criterion for edges, which is not checked for convergence Default 20 m. **Required share of converged edges**: weakens the convergence condition. Share of the edges in % which must at least have converged for the convergence criterion to be fulfilled. Default value 95 %, default value when loading network files which do not contain this attribute yet: 100 %.

Element	Description
Volume on edges	☑ If the option is selected, the convergence criterion is fulfilled if, on all edges, the absolute number of vehicles (traffic volume) in all evaluation intervals in comparison to the previous iteration has not changed more than the specified number of vehicles. Default 15 vehicles. **Required share of converged paths/edges**: weakens the convergence condition. Share of the paths/edges in % which must at least have converged for the convergence criterion to be fulfilled. Default value 95 %, default value when loading network files which do not contain this attribute yet: 100%. To exclude the impact of particularly short edges, slightly reduce the value of this attribute to approx. 90 to 95 percent.
Required number of consecutive converged simulation runs	Number of converged simulation runs after which Vissim is to end the simulation runs even if a higher **Number of runs** has been specified in the **Simulation parameters**.
Behavior upon convergence:	Convergence may already be reached before the number of simulation runs is completed that has been defined in the simulation parameters, in the **Number of runs** box. In this case, you can choose how to want Vissim to behave. - **Exit**: Do not perform another simulation run - **Ask** (default): Continue to perform simulation runs. If after another simulation run, convergence is reached, a window opens, allowing you to select one of three options. - **Complete all runs**: Perform the defined number of simulation runs. When convergence is reached, do not open a window with behavior options.

4. Confirm with **OK**.

Note: In order for the convergence to be reached, note the following points:

- Select only one of the three convergence criterion.
- Preferably, choose **Travel time on paths**.
- If you select multiple convergence criteria, there is a chance that convergence will never be reached because due to the increasing requirements the convergence criteria could become too strict.
- In particular, if you select the convergence criterion **Volume on edges**, there is a chance that convergence will never be reached because the absolute number of vehicles on the highest volume links fluctuate more than on links with less volume although the percentage deviation is the same.

The test for convergence is carried out at the end of each evaluation interval. When convergence occurs with several simulation runs in a row, the following message opens: **All selected convergence conditions are true. Cancel further simulation runs?** You can select **Cancel** if you do not want to perform any further simulation runs or a specified number of simulation runs. Before the start of the simulation runs, you can specify the number of converged simulation runs after which Vissim shall not run simulations any more using the attribute **Required number of consecutive converged simulation runs**.

7.11.5.1 Checking the convergence in the evaluation file

You can also check the behavior of the convergence during the iteration in the convergence evaluation file. The convergence evaluation file contains a statistical evaluation of the changes from travel time and volume on the edges/paths from one iteration to the next evaluation (see "Saving data about the convergence of the dynamic assignment to a file" on page 1067).

7.11.5.2 Showing converged paths and paths that are not converged

In the **Paths** list, you can show if a path converged or not using the attribute **Converged** (**Conv**). The convergence criterion is fulfilled if the percentage change for all paths in all evaluation intervals is smaller than the specified threshold for the selected convergence criterion.

7.11.6 Attributes for the guidance of vehicles

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Route guidance** tab.
3. Make the desired changes (see "Defining route guidance for vehicles" on page 764).
4. Confirm with **OK**.

7.11.7 Controlling iterations of the simulation

For the calculation of the dynamic assignment, the modeled network is simulated again.

7.11.7.1 Saving data in the path file and cost file

During the iterations, information regarding paths in the network and travel times at the edges in the network is collected. This information is saved in the following files:

- Path file **.weg*
- Cost file **.bew*

These files represent the current state of the assignment. The names of these files can be entered (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771). Thereby you can save the different states of the assignment.

7.11.7.2 Not saving data in the path file and cost file

You can deactivate the output of evaluations and paths during the simulation. This makes sense, for example, when the assignment is converged and the path selection should no longer be changed for the following simulations.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Files** tab.
3. Deactivate the options **Store costs** and **Store paths (and volumes)**.

7.11.7.3 Preventing unrealistic congestion of paths

During iteration, the number of known paths increases. In the first iterations, only a small number of paths are available for each origin-destination relation. This can lead to unrealistic congestion of these paths because the traffic demand cannot be distributed to enough paths. These congestion jams disappear again in the following iterations when more paths are found. Due to exponential smoothing of the travel times, the congestion jams continue to have an effect on some of the iterations, slowing down the convergence of the processes.

In order to prevent these initial congestion jams, it may be necessary in the first iteration to send only a part of the demand to the network. The traffic volume can increase with every iteration until the entire demand is allocated. For the modeling of this procedure, Vissim offers a global scale of all specified demand matrices.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Parameters**.

   *The **Dynamic Assignment: Parameters** window opens.*

2. Select the **Files** tab.
3. Enter the desired value in the field **Scale total volume to**.

7.11.8 Setting volume for paths manually

In the parameters of dynamic assignment, on the **Choice** tab, select **Use volume (old)** to set the distribution of vehicles across the paths during a simulation run or prior to a simulation run.

7.11.8.1 Setting the distribution of vehicles during a simulation run

To set the distribution of vehicles across the paths of a time interval that has not begun yet, use the **Volume (old)** attribute.

1. Start the simulation (see "Running a simulation" on page 840).
2. From the **Traffic** menu, choose > **Dynamic Assignment** > **Paths**.

   *The list **Paths** opens.*

3. Verify that in the list, the column for the **Volume (old)** attribute shows the time intervals for which you want to set volumes.

4. Under **Volume (old)**, enter the values of your choice for the desired time intervals that have not begun yet.

7.11.8.2 Setting the distribution of vehicles prior to a simulation run

When in the attributes of dynamic assignment, in the **Files** tab, in the **Path file** box, a **.weg* file is selected, you can read in the paths from this file and set the volume before starting the simulation.

1. From the **Traffic** menu, choose > **Dynamic Assignment** > **Read Paths**.

Tip: Alternatively, to read paths, in the **Paths** list, right-click them and from the shortcut menu, choose **Read Paths**.

*The **Paths** list is updated.*

2. Verify that in the list, the column for the **Volume (new)** attribute shows the time intervals for which you want to set volumes.
3. Under **Volume (new)**, enter the values of your choice.
4. From the **Traffic** menu, choose > **Dynamic Assignment** > **Write Path File**.

7.11.9 Influencing the path search by using cost surcharges or blocks

Despite ample exact modeling of the road network, the traffic control and the demand, the result of the dynamic assignment can deviate from the situation which is observed in reality. This can result from the fact that the decision model in Vissim cannot account for all influences, due to their abstractness, which affect the driver in reality. In these cases, the use of parts of road networks are changed during the path selection of the dynamic assignment.

7.11.9.1 Modeling cost surcharges

If sections of the road network experience more or less traffic than expected, these can be modeled via the use of cost surcharges. Put cost surcharges on the respective links or connectors. A cost surcharge is added to the total costs of the link every time a path uses a link, independent of the length of the used section of the link. For example, for a link which cuts through two nodes, the surcharge is added three times when the vehicle traverses through the entire link (see "Attributes of links" on page 409).

7.11.9.2 Blocking edges

You can block edges for the path search of the dynamic assignment and thereby displace traffic from specific network sections (see "Excluding edges from dynamic assignment" on page 720). Blocked edges are shown in red.

7.11.9.3 Limiting the number of paths

In principle, the number of paths which are found in the iterations are not limited. By default, all paths which are found are collected in the path archive. The paths are thereby available for the path selection. A consequence of this can be that from a few attractive paths only a few vehicles will be distributed, even when much better paths are found in later iterations and the use of old paths is very unlikely in reality. These paths use up memory space and calculation time. You can enter one of the two following values and thereby limit the number of applied paths per OD relation (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771):

Element	Description
Reject paths with too high total cost **Max. extra cost compared to the best path**	Threshold in % for the cost difference as a criterion for the path search
Limit number of paths **Max. number of paths per parking lot relation**	Maximum number of paths per parking lot relation as a criterion for the path search

The limiting of the absolute number of paths can be unsuitable in the network if there is an alternative path for some origin areas to destination areas, which also should be used and for other relationships for which few paths are possible. In this case the limiting of the cost difference between the best and worst path can be the suitable solution. Thereby the paths which are discarded are those which, via a preallocation of unused edges at 0.1 seconds, appear particularly attractive. If it is later realized, after a number of vehicles have driven there, that these paths are much more expensive than others, they can be discarded again without any adverse consequence.

By default, the path file is recorded at the beginning of every iteration. Within the respective time interval, for each OD relation, none of the paths are used for which a cost difference in comparison to the cheapest path for all evaluation intervals is larger than the specified value. This means these paths can be used in later time intervals and/or iterations, without having to search for them again as a shortest path.

7.11.9.4 Closing paths with vehicle routes of the type Closure

You can also confine the path search for dynamic assignment by marking a specific sequence of links as vehicle routes of the type **Closure** in the network. To do so, in the Network Editor, define a route closure like a static route with a routing decision marker of the type **Closure** for a destination section (see "Defining parking routes" on page 472).

For the dynamic assignment, a path which contains a route closure is only used when it is the only path which is found as a result of the shortest path search for a parking lot OD relation. In the event that no better paths are found, which correspond to a lower generalized cost, vehicles of the dynamic assignment can use a path which contains a route closure.

If in principle a specific sequence of links should not be used, a complete edge or complete turn relation in the node is blocked (see "Editing edges" on page 717). You can change the node polygon and determine the desired link sequence from a complete turn relation from the node entrance up to the node departure or block all turn relations which contain this sequence of links.

Notes:
- Block paths only as a last resort. Instead, define high costs or add reduced speed areas.
- To block turn relations, block the edge of a node.
- Before you block paths or parts of paths, check to make sure there is no other modeling possibility available. Often it is sufficient to block a turn relation when the border of the node is defined.

7.11.10 Evaluating costs and assigned traffic of paths

To evaluate the result of a dynamic assignment, we recommend in particular using the following result lists and files:

- Result list **Paths** (see "Showing data about paths of dynamic assignment in lists" on page 1109)
- Result list **OD pairs** (see "Displaying OD pair data in lists" on page 1027)
- Cost file **.bew* (see "Files of dynamic assignment" on page 1195)
- Path file (see "Files of dynamic assignment" on page 1195)

7.12 Correcting demand matrices

Using Matrix correction, you can adjust the demand in the origin-destination matrix to the count data available in numeric link attributes, vehicle class-specific subattributes of the attribute **Count data** or user-defined attributes. The results of an assignment must be saved to a Path file.

Examples of Matrix correction use cases:

- You have one OD-matrix and assignment results of dynamic assignment that include paths and their volumes as well as a set of count data in link attributes. Using Matrix correction, you automatically adjust the OD-matrix to the new path volumes, creating values that are closer to the count data. The count data cover an integer multiple of the evaluation interval. The OD-matrix does not change when count data is collected.
- You receive an assignment result that includes multiple vehicle classes and the respective OD-matrices. The count data is listed separately by vehicle class. You use Matrix correction successively for the individual OD-matrices.

Alternatively, you can perform Matrix correction via the COM interface.

Method used

Vissim uses the least squares method. The total of squares of the differences between count data and volumes and the total of squares of the differences between the original and corrected matrix values is minimized. The number of iterations is set to 1000. OD relations with a volume of ZERO are not adjusted. The values in the other cells of the matrix can be edited.

Using matrices with realistic values

The matrix correction is not suitable for generating a matrix with realistic values from a "dummy" with unrealistic values. You need a matrix from a demand model, for example from Visum. Use this matrix in Vissim for simulation runs with dynamic assignment until the model converges. Then you will have current paths. Alternatively, adopt the paths from a Visum assignment. Then run the matrix correction with your count data. From a mathematical point of view, the matrix correction always provides a result based on the method of least squares. However, this result is not automatically meaningful and realistic. Make sure you save meaningful matrix values and count data.

Notes:

- Avoid correcting demand matrices in a Vissim network that also uses trip chain files to model transport demand.
- Avoid correcting demand matrices in a Vissim network with vehicles that have been added to the Vissim network via the COM interface.
- You cannot undo Matrix correction. Before using Matrix correction, save a backup of the OD-matrix you want to use as a basis for matrix correction.
- The assignment result may become inconsistent when you use the following elements:
  - dynamic routing decisions
  - real parking lots
  - route guidance vehicles
  - Matrix correction can calculate values that are not integer.
- The OD-matrix must contain integer values. These correspond to the values of the assignment results in the Path file.

7.12.1 Defining and performing Matrix correction

For Matrix correction, select the desired origin-destination matrix, the count data source and the path attributes. The count data must be available as numeric link attributes, vehicle class specific subattributes of the attribute **count data** or as user-defined attributes.

1. Ensure that at least one OD matrix containing demand data has been defined (see "Modeling traffic demand with origin-destination matrices" on page 721).
2. Ensure that dynamic assignment has been performed and a Path file is available.
3. Specify the source of your count data:
   - To use the count data of a numeric link attribute (default setting), make sure that the desired data is available in the link attribute.
   - To enter count data for a specific link and access it via the **Count data** attribute, in the **Links** list, show the vehicle class-specific subattribute of the **Count data** attribute. Then, in the row of the respective link, enter the values.
   - To access count data via a user-defined attribute, define the attribute according to your requirements (see "Using user-defined attributes" on page 210).
4. From the **Traffic** menu, choose > **Dynamic Assignment** > **Matrix correction**.

*The **Matrix correction** window opens.*

5. Make the desired changes:

Element	Description
Input and output matrix	Number and name of OD-matrix whose values you want Matrix correction to correct based on selected count data and volumes.
Counts for links:	Shows the link attributes selected. The count data of a link is made up of the total of the selected attributes.
➕	Opens the window **Links: Select attribute**: Select the desired numeric link attributes and/or subattribute of the **Counted data** attribute. The data of the selected attributes are used as target values during Matrix correction. Under the **Counted data** attribute, you can select the desired vehicle class. If you have created user-defined attributes, you can select them as well.
Volumes on paths:	Shows the path attributes selected. The volume of a path is made up of the total of the selected attributes.
➕	Opens the **Paths: Select attribute**: Select the desired numeric path attributes, for example **Volume (new)**.

6. Click the **Execute** button.

If no paths are available, Vissim will read in the Path file. Based on OD-matrix data, selected count data of links and paths and scaling factors, Vissim calculates new values for the OD-matrix. The scaling factors cannot be changed.

7.13 Generating static routes from assignment

You can convert the current state of the dynamic assignment and thereby also the paths found and their congestions into a Vissim model with static routes. This Vissim model can be used without the dynamic assignment module.

Vehicle inputs and routing decisions are generated based on the contents of the path file **.weg*, the cost file **.bew* and the total demand of matrices.

Note: Ensure that for the dynamic assignment, the simulation time is a whole number, frequently the evaluation interval. Otherwise the path and cost files can be incomplete because you save data per evaluation interval.

To generate routing decisions and vehicle inputs the following data is used:

- The path volumes last saved to the path file (**Volume (new)** attribute, vehicles that departed within the time interval)
- The total origin volume of zones in the OD matrices

Where:

- For each origin parking lot, one vehicle input is generated for each time interval resulting from it whose traffic volume corresponds to the total of path volumes in the path file.
- Next, the traffic volumes of all origin parking lots of a zone are multiplied with a common factor. Their sum then corresponds to the total origin volume of this zone in the matrix or matrices.
- For each parking lot a static routing decision per group of vehicle types is generated, which has an identical path selection behavior in the dynamic assignment, e.g. with the same weighted parameters for the costs, the same accessible road network and the same parking lot selection parameters. The relative volume for each route corresponds to the route volumes from the path file.
- Vissim selects the next higher number available, respectively, for the numbers of the generated vehicle input and the generated routing decisions.

The vehicle compositions are generated from the overlapping of all vehicle compositions allocated to matrices. Vehicle compositions (combination of vehicle type - desired speed) with a relative share < 0.001 (0.1 %) pare not taken into consideration.

If in your Vissim network, the link on which a destination parking lot is placed is connected to additional links via connectors, vehicles will use the connectors and links after static routes have been generated and will remain in the Vissim network.

1. Select from the menu **Traffic** > **Dynamic Assignment** > **Create Static Routing from Assignment**.

   *The **Create Static Routing** window opens. The settings are only available during the run time of Vissim and are not saved.*

2. Make the desired changes:

Element	Description
Limit number of routes	All paths are converted to static routes. All path volumes are defined based on the relative distribution of routes. For each decision section, the absolute and the relative minimum volume as well as the number of routes per destination section are compared with the set parameters. Thereby the number of generated static routes is reduced. All routes, which do not fulfill the criteria, are deleted with their volumes. Decision sections are not deleted, even when all routes on a decision section are deleted.

Element	Description
rel. min. Volume	The static route is deleted if the following applies for the relative volume: *Relative volume in each time interval < current relative minimum volume* Value range [0.00 to 1.00], Default: 0.05, two decimals If the **rel. min. Volume** = 0.00, no static routes are generated because this criterion is discarded. *Relative volume in time interval = absolute volume in time interval /Sum of volumes from all time intervals* If in a time interval the volumes of all routes of a routing decision = 0, the relative minimum volume has not been defined. The time interval is not evaluated.
Abs. Min. Volume	The static route is deleted if the following applies for the absolute volume: *Absolute volume in each time interval < current absolute minimum volume* Value range [0 to 999 999 999], Default value: 2, integer If the **Abs. Min. Volume** = 0, no static route generated is discarded because of this criterion.
max. number of routes (per destination)	All the routes from the start section to the destination section are taken into consideration. For each decision section, the number of routes to destination sections is specified. Multiple destination sections of a routing decision, which are located closely to each other on a link (tolerance ± 1 m), are regarded as one single destination section. If the number of routes per destination section is > current **Max. no.** of **routes**, then the routes with the smallest time interval volume sums are removed. If a very unlikely value is entered, for example, 999,999, no static routes are generated because this criterion is discarded. Value range [0 to 999 999 999], Default value: 10, integer

3. Click on **OK**.

   Create static routing is started.

7.14 Using an assignment from Visum for dynamic assignment

For the dynamic assignment, you can use the results of a static assignment from Visum in Vissim. The goal is to reduce the number of iterations, which are necessary to reach convergence in Vissim. On the basis of the assignment from Visum, the dynamic assignment is carried out by Vissim. You can calculate this dynamic assignment automatically or manually.

7.14.1 Calculating a Visum assignment automatically

You can automatically calculate the dynamic assignment based on a static assignment from Visum. You can then use it to perform simulations.

7.14.1.1 Starting automatic calculation

In Vissim, from the **Traffic** menu, choose > **Dynamic Assignment** > **PTV Visum Assignment**.

*The Visum assignment is calculated with the Visum converter. The Visum converter is a standard part of your Vissim installation. The Visum converter stores the routes from Visum in the Vissim path file *.weg. If a path file with the same name has already been saved, Vissim saves a backup file *.bak. You can change the name of the path file (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).*

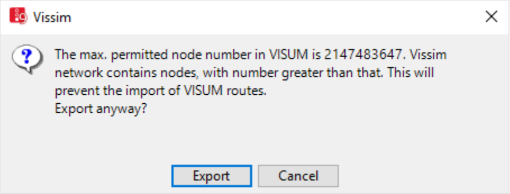
Notes:

- If you run the dynamic assignment multiple times, the files **.weg* and **.bak* are overwritten.
- A static assignment from Visum in Vissim takes place in much less time than a microscopic Vissim simulation of individual vehicles.

7.14.1.2 Running a simulation

After the automatic calculation, you can start the simulation in Vissim without further steps (see "Running a simulation" on page 840). You can also perform several simulation runs (see "Defining simulation parameters" on page 840). In the first iteration the paths computed by Visum are used. After the first iteration, the path files and cost files generated by Vissim are used. The file names are kept.

If the Vissim network contains nodes whose numbers are greater than the maximum allowed Visum node number 2,147,483,647, the assignment cannot be calculated. The **PTV Visum Export** window opens.



1. Change the Vissim node numbers so that they are less than the maximum allowed Visum node number.
2. Select from the menu **Traffic** > **Dynamic Assignment** > **PTV Visum assignment**.

7.14.1.3 Displaying warnings, notes and debug messages for the calculation

In the automatic calculation of the assignment the warnings, notes and debug messages are displayed in the **Message** window.

- In the **View** menu, choose > **Messages**.

7.14.2 Stepwise Visum assignment calculation

You can perform the dynamic assignment from Visum also based on a static assignment step-by-step:

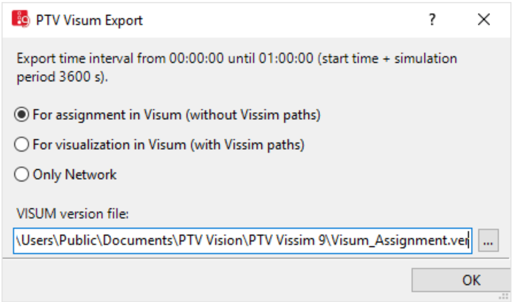
- Export network from Vissim
- Load network data in Visum
- Edit the network in Visum
- Calculate assignment in Visum
- Export routes from Visum
- Import routes into Vissim

- Perform simulation run in Vissim.

  This ensures that for all origin-destination relations with demand> 0 the paths throughout the Vissimnetwork are available.

7.14.2.1 Exporting network data from Vissim

1. From the **File** menu, choose > **Export** > **PTV Visum (Nodes/Edges)**.
2. Export network data and matrix for the assignment in Visum (see "Exporting data" on page 384).
3. Copy the **Export time interval** shown during export. You will need the data later in Visum.



*A version file *.ver will be saved.*

7.14.2.2 Loading the version file in Visum

- Load the version file **.ver*.

7.14.2.3 Modifying network in Visum

- Make the desired changes in the network.

Note: Changes in the network topology can cause the import of routes to fail in Vissim.

The network topology in Visum is not affected by the following modifications:

- Change link attributes (Type, Length, Capacity PrT, v0 PrT)
- Change link polygons
- Change node attributes (Type, Capacity PrT, t0 PrT, Control type)
- Change node geometry
- Change link orientations
- Change node coordinates
- Add signalization using the Node editor

The following modifications in Visum could cause the route import in Vissim to fail:

- Insert and delete nodes
- Edit node number
- Insert and delete links
- Open blocked links (change TSys of links)
- Open blocked turns (change TSys of turns)
- Insert and delete connectors
- Open blocked connectors (change TSys of connectors)
- Insert and delete TSys/Modes/DSegs

7.14.2.4 Calculating assignment in Visum

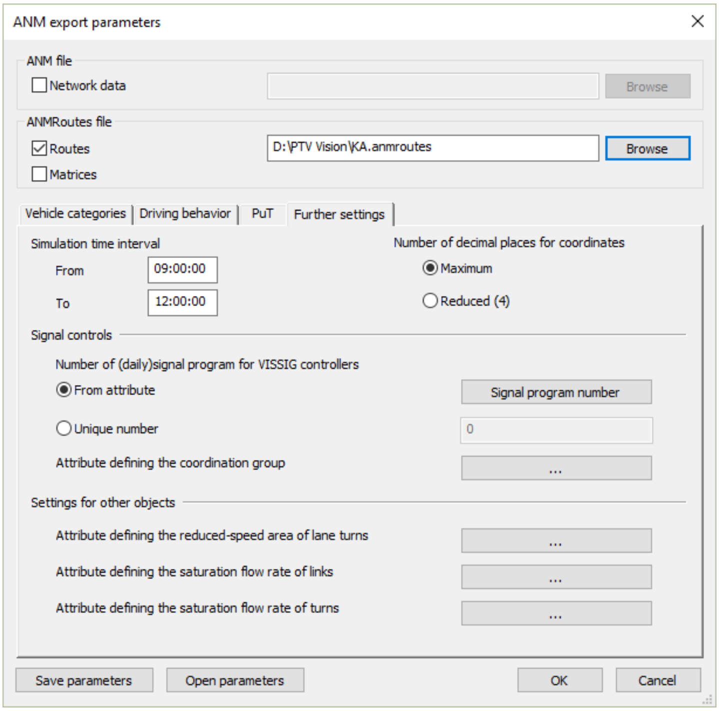
You can find further information in the Visum Help.

7.14.2.5 Exporting assignment routes into Visum

For information on ANM Export from Visum, please refer to the section **Using interfaces to exchange data** of the Visum Help.

Note: Please make sure that only routes **.anmroutes* are exported. Do not export network data **.anm* or matrices.

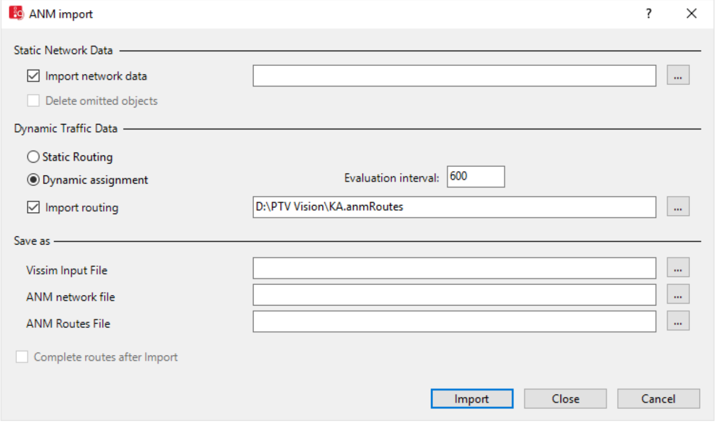
1. Enter the **Export time interval** which you have noted when exporting from Vissim into the **ANM export parameters** window in the **Further settings** tab in the field **From** and **To** in Visum.



2. Export the data from Visum.

7.14.2.6 Importing routes for dynamic assignment into Vissim

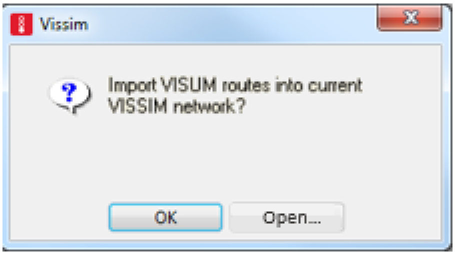
1. From the **File** menu, choose > **Import**> **ANM (Vistro/Visum)**.
2. In the **ANM Import** window, deselect **Import network data**.



3. Select option **Dynamic Assignment**.

4. Select option **Import routing**.
5. Select the **.anmroutes* file that was exported from Visum.
6. Click **Import**.

   The following message indicates that no errors occurred during the import:



7. Confirm with **OK**.

Vissim saves the calculated routes in a path file **.weg*. If a path file with the same name has already been saved, Vissim saves a backup file **.bak*. You can change the name of the path file (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).

Note: The distribution of the source volume of a zone to its parking lots in Vissim is based on the entered relative loads and does not depend on the distribution to the source zone connections in the assignment in Visum. The result of the assignment in Visum is only used for the distribution of vehicles from parking lots from Vissim to the paths Visum has found to the chosen destination parking lot.

7.15 Calculating toll using dynamic assignment:

For path selection, in the Vissim network, dynamic assignment takes link sequences without toll routes as well as link sequences with managed lanes facilities and toll routes into account (see "Defining a vehicle route of the type managed lane" on page 476). This allows you to model, simulate and evaluate the following use cases, for example:

- Impact of toll route pricing on the surrounding traffic situation: How will demand on toll routes, toll-free routes and alternative routes develop if pricing is changed?
- Effects of a future change in demand: How will the capacity requirements for toll routes, toll-free routes and alternative routes change when demand changes?

Decision on toll route is based on a managed lanes routing decision

As with static routing decisions and routes, in dynamic assignment the vehicle decides which toll route to follow when it traverses a managed lanes routing decision. The decision is made based on the decision model and the toll pricing calculation model of the assigned managed lanes facility. (see "Defining managed lane facilities" on page 327), (see "Defining decision model for managed lane facilities" on page 329), (see "Defining toll pricing calculation models" on page 331).

In order for the vehicle to take the managed lanes routing decision and its toll route or toll-free route into account in dynamic assignment, the path search must have considered the managed lanes routing decision and the common destination section of the toll route and toll-free route placed downstream, and these objects must lie on a path. This path does not have to include the entire toll route.

If at the manged lanes routing decision, the vehicle can neither select the toll route nor the toll-free route, it continues on the path assigned to it by dynamic assignment at the start of the journey.

Vissim combines edges and paths

After the dynamic assignment graph has been generated, Vissim combines edges, in particular for path selection and convergence, under the following conditions:

- The edges lie within a node or between two nodes.
- The toll-free route runs over one edge and the toll route runs over the other edge.

For each of the combined edges, Vissim aggregates the volume and travel time for the shortest combined edge. This edge then represents the combined edges. The **Edges** list shows all edges. It also includes combined edges.

For path searches and path selection, Vissim combines the two routes that run along the toll route and toll-free route of a managed lanes routing decision. This path represents the two combined paths. The data represented is displayed in the **.weg* path file.

Modeling of managed lanes routing decisions and toll routes

When modeling the above-mentioned use cases, pay particular attention to the effects of managed lanes routing decisions and the notes on modeling (see "Mode of action of routing decisions of the type Managed Lanes" on page 463).

Closed edges, connectors and routes

- The vehicle does not take into account a managed lanes routing decision positioned on a closed edge.
- If a connector is closed for all vehicle classes, there is no edge traversing the connector that can be taken into account for path search. If a toll route leads via a closed connector, vehicles will not be able to use the toll route.
- If a connector is closed for some vehicle classes only, an edge leads via the connector. If this edge is the shortest of the combined edges, the edge is considered for path search.
- A managed lanes routing decision is not taken into account if a route is closed and the following applies for the toll objects on the route (see "Defining a vehicle route of the type closure" on page 481):

- Managed lanes routing decision and destination section are on the closed route. The closed route corresponds to the toll route or toll-free route between the managed lanes routing decision and the destination section.
- The managed lanes routing decision lies on the closed route. The closed route corresponds to the toll route or toll-free route from the managed lanes routing decision on.
- The common destination section of the toll-free route and the toll route lies on the closed route. The closed route corresponds to the toll route or toll-free route up until the destination section.
- The entire closed route lies on a toll route or on a toll-free route.

Changes to toll objects delete dynamic assignment graph

After a managed lanes routing decision or a toll route has been changed, the dynamic assignment graph must be recreated. (see "Building an Abstract Network Graph" on page 697), (see "Generating a node-edge graph" on page 718).

Dealing with detours

A toll route cannot be a detour of its toll-free route, and vice versa. Even if in the parameters of dynamic assignment, on the **Choice** tab, **Avoid long detours:** is selected, the program will not consider using a toll route as a detour for a toll-free route or vice versa.

8 Using add-on module for mesoscopic simulation

Note: You must have a license for the add-on module.

Mesoscopic simulation is a vehicle simulation that provides faster run speed when compared to microscopic simulated models. Mesoscopic simulation uses a simplified vehicle following model (see "Car following model for mesoscopic simulation" on page 803) for modeling vehicle behavior. It allows for the fast simulation of large networks during dynamic assignment. Mesoscopic simulation is event based. Therefore, in contrast to microscopic simulation, vehicle data is not updated with every time step but only at times when changes occur in the network or vehicle behavior. These so-called events may be triggered by different situations, e.g.:

- Vehicle entering the network
- Change in traffic signal state
- Vehicle entering a node
- Vehicle entering a microscopically simulated area
- Vehicle exiting a microscopically simulated area

Mesoscopic and microscopic simulations can be combined to form a hybrid simulation. To do so, network sections where vehicles and their behavior are required to be simulated microscopically need to be defined. This can be particularly useful when:

- detailed evaluations are only needed for certain parts of the network, but a large network is required to model route choice.
- for parts of the network mesoscopic simulation is not precise enough and conflicts cannot be modeled realistically, e.g. in turbo roundabouts or complex intersections.

For the mesoscopic simulation of your network, the factors dealt with in detail in the following sections need to be considered.

You can find examples of mesoscopic simulation in the directory *..\Examples\Examples Training\Meso*.

8.1 Quick start guide mesoscopic simulation

The quick start guide demonstrates the key steps and settings required for mesoscopic simulation.

Note: When you import a network via ANM import, all network objects required for mesoscopic simulation are created and the parameters are set for mesoscopic simulation (see "Importing ANM data" on page 366), (see "Generated network objects from the ANM import" on page 372). In this case, make sure that the values and settings meet your requirements, e.g. the meso follow-up gap and the meso critical gap (see "Attributes of meso turns" on page 833), (see "Attributes of meso turn conflicts" on page 835).

1. Prepare your Vissim network according to the requirements for dynamic assignment (see "Using the dynamic assignment add-on module" on page 692), (see "Defining parking lots for dynamic assignment" on page 700), (see "Defining zones" on page 704).
2. Select the desired matrix or matrices (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
3. Make sure the simulation duration specified in the matrix and in the simulation parameters is the same (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
4. Ensure that the numbers of the zones in the matrix and the numbers of the zones in the Vissim network are the same (see "Attributes of zones" on page 704).
5. Ensure that the settings for dynamic assignment meet your requirements (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
6. Insert nodes for mesoscopic simulation (see "Defining nodes" on page 708). When doing so, make sure that the modeling rules for meso graphs are applied (see "Modeling meso network nodes" on page 809), (see "Mesoscopic node-edge model" on page 804).
7. For these nodes, select the attributes **UseForDynAssign** and **UseForMeso** (see "Attributes of nodes" on page 709), (see "Attributes of meso nodes" on page 829), (see "Attributes of meso edges" on page 832).
8. Create the meso graph (see "Generating meso graphs" on page 837).
9. To simulate parts of your Vissim network microscopically, make sure they lie in sections (see "Modeling sections" on page 677), (see "Hybrid simulation" on page 837).
10. Select the **Meso** simulation method (see " Selecting simulation method micro or meso" on page 840).
11. If you have defined sections in which you want Vissim to perform microscopic simulation, select the desired sections (see "Selecting sections for hybrid simulation" on page 838).
12. Make sure that the driving behavior parameters in the **Meso** tab meet your requirements (see "Editing the driving behavior parameter Meso" on page 317):
13. Make sure that for links you have selected the **Meso speed model** according to your requirements (see "Attributes of links" on page 409).
14. Make sure that for links, for which you selected the **Meso speed model** attribute **link related**, the **Meso speed** attribute meets your requirements (see "Attributes of links" on page 409).
15. Define the status of conflict areas In the **Conflict areas** list or in the coupled list **Nodes - Conflict areas** (see "Attributes of conflict areas" on page 565), (see "Attributes of nodes" on page 709).
16. Model node control (see "Node control in mesoscopic simulation" on page 807). For intersection control, define a critical gap, using the **Meso critical gap** attribute in the **Meso turn conflicts** list (see "Attributes of meso turn conflicts" on page 835) and a follow-up gap in the **Meso turns** list (see "Attributes of meso turns" on page 833).

17. To perform evaluations for mesoscopic and/or hybrid simulation, configure the following evaluations:
    - Network performance (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085)
    - Link evaluation (see "Showing data from links in lists" on page 1103)
    - Meso edge evaluation (see "Showing meso edges results in lists" on page 1064)
    - Meso lane evaluation (see "Showing meso lane results in lists" on page 1065)
18. Ensure that the desired simulation parameters have been set (see "Defining simulation parameters" on page 840).
19. Start the simulation (see "Selecting the number of simulation runs and starting simulation" on page 845).

8.2 Car following model for mesoscopic simulation

Mesoscopic simulation uses a simplified car following model based on the work of M. Mahut (see "References" on page 1201). The main difference to car following models used in microscopic simulation is that braking and accelerating are not included. A vehicle changes its speed immediately and has only two statuses:

- The vehicle is driving at maximum speed.
- A temporal distance is kept to the rear end of the preceding vehicle. This temporal distance corresponds to the response time.

The precise formulation of a car following model depends on the meso speed model selected. You may select a meso speed model for each link (see "Attributes of links" on page 409):

- **Link-based**: less detailed. Can therefore lead to an improvement in simulation speed.
- **Vehicle-based** (default setting): high level of detail. Vehicles attempt to drive at their individual desired speed.

8.2.1 Car following model for the meso speed model Link-based

The car following model uses fixed link speeds:

$$x_{Follower}(t) = \min_{\epsilon>0} \left(x_{Follower}(t-\epsilon) + \epsilon V_{Link}, x_{Leader}(t-\tau_{Follower}) - \lambda_{Follower} \right)$$

These include:

$x_{Follower}$: Front end position of trailing vehicle on current link

x_{Leader}: Front end position of preceding vehicle on current link

V_{Link}: Meso link speed

$\tau_{Follower}$: Response time of trailing vehicle

λ_{Leader}: Effective vehicle length of preceding vehicle, calculated from the individual vehicle length and the meso standstill distance.

8.2.2 Car following model for the meso speed model Vehicle-based

The car following model uses individual vehicle speeds:

$$x_{Follower}(t) = \min_{\epsilon>0} \left(x_{Follower}(t-\epsilon) + \epsilon V_{Follower}, x_{Leader}(t-\tau_{Follower}) - \lambda_{Leader} \right)$$

In this formulation, V_{Link} is replaced by $V_{Follower}$: Each vehicle tries to drive at its individual desired speed $V_{Follower}$.

8.2.3 Additional bases of calculation

Both car following models use individual vehicle lengths and response times that are based on driving behavior parameters.

By default, the meso speed model is set to **Vehicle-based**. Vehicles therefore drive at their individual desired speed in free flowing traffic. Desired speed decisions and reduced speed areas result in changes to the vehicle’s desired speed. When used the following limitations apply:

- Reduced speed areas and desired speed decisions apply per lane and extend over the entire meso edge (see "Mesoscopic node-edge model" on page 804).
- Multiple reduced speed areas and/or desired speed decisions cannot be used on the same lane within the same meso edge as they are considered to overlap. In this instance Vissim displays a message when the meso graph is created and uses the information of only the object that is furthest downstream.

For the **Link-based** meso speed model the following applies:

Desired speed decisions and reduced speed areas are ignored. Differences in the desired speed of vehicle classes cannot be modeled. This means that different speed limits on different lanes can only be modeled with the vehicle-based model. However, in a city center network, the **Link-based** meso speed model might be sufficient. Use of the Link-based meso speed model in most parts of the network can lead to enhanced simulation speed.

8.3 Mesoscopic node-edge model

Mesoscopic simulation requires a special node-edge model also called a meso graph. Creating a meso graph is a prerequisite for mesoscopic simulation (see "Generating meso graphs" on page 837). Based on the meso graph and evaluations, you can generate meso edge results and meso lane results (see "Performing evaluations" on page 1001).

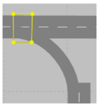
8.3.1 Properties and nodes of the meso graph

The meso graph is different from the abstract network graph for dynamic assignment. It is a combination of all meso nodes and meso edges that are relevant for mesoscopic simulation:

- The meso graph is created based on nodes where the attribute **Use for mesoscopic simulation** has been selected. These nodes are referred to as **Meso network nodes** and displayed in the **Nodes** list (see "Modeling nodes" on page 705). By default, **meso network nodes** can be edited the same way as other nodes.

- The modeling of meso network nodes must comply with the rules for defining meso network nodes in mesoscopic simulation (see "Modeling meso network nodes" on page 809).
- In meso graphs, Vissim does not use the modeled **meso network nodes**, but automatically generates corresponding **meso nodes** when it creates the meso graph. Thus, for each **meso node** modeled, there is a corresponding **meso network node**.
- Vissim assigns this **meso node** the value **Node** in the **Type** attribute, when it generates the meso graph. This is because the **meso node** is based on a modeled network object of the type **Node**.
- However, the **meso nodes** are not shown in the **Nodes** list, but in the **Meso nodes** list (see "Attributes of meso nodes" on page 829).
- A **meso node** cannot be edited like a network object **node**. Only the respective **meso network node** that Vissim used to generate the meso node can be edited. You can show the **Meso nodes** list as a relation of the **Nodes** list, to list each meso network node together with its corresponding **meso node**.
- In addition, when creating the meso graph, Vissim automatically generates additional **meso nodes**, e.g. for parking lots or dynamic routing decisions. It further generates additional nodes where, for mesoscopic simulation, relevant link attributes change, e.g. the number of lanes or the meso speed model. Accordingly, in the **Type** attribute, Vissim assigns each of these meso nodes their corresponding value, e.g. **Parking lot**, **Routing decision** or **Other** (see "Attributes of meso nodes" on page 829).

This allows for a distinction between **meso network nodes** and **meso nodes** during mesoscopic simulation. The table shows the relationships and order in which you define a node, you mark it as a meso network node, Vissim then generates the meso network node at the start of the simulation and the **meso node** for the **meso network node**.

Element	Description
Nodes 	Models network object **Node** in the Network editor (see "Modeling nodes" on page 705), (see "Modeling meso network nodes" on page 809).
Marks node as meso network node	Selects the attribute ☑ **Use for mesoscopic simulation** for the modeled node. The node is displayed in the **Nodes** list (see "Attributes of nodes" on page 709). The **Nodes** list contains all nodes of the network object type **Node**. At the **meso network nodes**, the attribute **UseForMeso** (**Use for mesoscopic simulation**) is selected.
Meso node	When after modeling the network and configuring the simulation, you start the simulation, Vissim generates the meso graph. When generating the meso graph, Vissim automatically generates a **meso node** for each **meso network node** modeled. As this **meso node** is not the **meso network node**, it is not displayed in the **Nodes** list, but in the **meso nodes** list. You can show the **Meso nodes** list as a relation in the **Nodes** list (see "Attributes of meso nodes" on page 829).

8.3.2 Differences between meso network nodes and meso nodes

Meso network nodes

- A meso network node is a network object of the type **Node** for which the attribute **Use for mesoscopic simulation** (**UseForMeso**) has been selected.
- A meso network node can be defined and edited in the same way as any other node of the network object type **Node**. You may define polygon nodes and segment nodes (see "Modeling nodes" on page 705).
- Meso network nodes are shown in the **Nodes** list (see "Attributes of nodes" on page 709).
- However, meso network nodes are not shown in the **Meso nodes** list (see "Attributes of meso nodes" on page 829).

Meso nodes based on meso network nodes

- Meso nodes cannot be edited in the Network editor.
- Meso nodes that are automatically generated based on meso network nodes during creation of the meso graph are shown in the **Meso nodes** list. In the **Type** attribute, they have the value **Node**. The **Node** attribute also contains the number of the corresponding meso network node.

Meso nodes automatically generated by Vissim

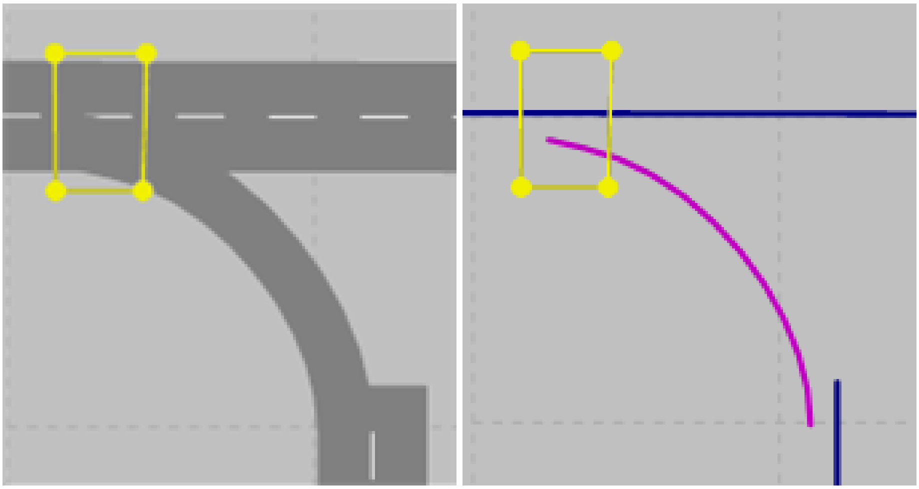
Besides the meso nodes that Vissim generates for the modeled meso network nodes, Vissim generates additional meso nodes when creating the meso graph. For these meso nodes the following applies:

- Meso nodes have no longitudinal extension.
- Meso nodes cannot be edited.
- Meso nodes are only displayed in the **Meso nodes** list (not in the **Nodes**) list.
- The **Type** attribute value of these meso nodes is different than for **Nodes**, e.g. **Parking lot**, **Routing decision** or **Other**.

8.3.3 Meso edges in meso graphs

The meso graph consists of meso nodes and meso edges. The following meso edges are distinguished:

- A **link meso node** connects two meso nodes via links and connectors that lie between the two meso nodes (see "Attributes of meso edges" on page 832).
- Within a meso node, a **meso turn edge** connects meso edges that lead into the meso node with meso edges that lead out of the meso node. The number of lanes and speed of links and connectors where a meso turn edge begins may be different for the meso turn edge itself. For instance, the meso turn edge may start at a double-lane link, but then lead via a single-lane connector and link to another double-lane link:



- The attributes of the **Defining links** of the respective meso turn, however, must not change. If in the meso node, a meso turn edge leads from a double-lane link via a link sequence, consisting of multiple connectors and links that begin with a single-lane connector, the following links and connectors of the meso edge must also be single-lane. This is described in **Rule 3** (see "Rules and examples for defining meso network nodes" on page 810).
- **Connector meso edges** are origin connector edges that lie within parking lots. An origin connector meso edge does not have a preceding edge. A destination connector meso edge does not have a successive edge.
- A **PT line connector** is an edge leading from a meso node of the type **PT line origin** to a meso node of the type **Node** or **Input**.
- A **micro-meso transition** is an edge leading from a meso node of the type **Micro-meso transition** to a meso node of the type **Micro-meso**.

8.3.4 Changes to the network will delete the meso graph

If changes are made to the network that affect the structure of the meso graph, the graph is deleted. The attributes list of meso-specific network objects then no longer displays any data. By creating a new meso graph, you can fill the attributes list with data again.

8.4 Node control in mesoscopic simulation

In mesoscopic simulation there are some rules for modeling signalized and non signalized intersections that differ from the rules that apply for microscopic simulation.

For signalized intersections note that the decisive factors are the stop position and storage capacity. The vehicle always stops at the meso network node. If the real situation cannot be modeled with one meso network node only, model several nodes, e.g. for a separately led right turn. For conflict control at intersections, in mesoscopic simulation, Vissim only looks at the vehicles on the edges that lead into the node.

Signalized intersections

Detectors in the mesoscopic simulation transfer the following data to dynamic controllers:

- The front edge of the vehicle reaches the detector.
- The vehicle is on the detector.
- The rear edge of the vehicle reaches the end of the detector.
- Number of front edges and rear edges
- Vehicle number, vehicle type, vehicle length
- Occupancy
- PT line

If the detector lies within a meso-node upstream of a signal head, the following applies:

  - The detector is treated as if it were directly upstream of the signal head.
  - The signal head is treated as if it were directly upstream of the node entry.

Fixed type signal control only distinguishes between the statuses stop and drive. The signal state sequences of signal groups either cause vehicles to wait at the entry of the meso network node or enter the node:

- The signal states **Red** and **Red flashing** are interpreted as stops.
- **Red/amber** is evaluated with the driving behavior based on the settings made in the **Behavior at red/amber signal** attribute (see "Editing the driving behavior parameter Signal Control" on page 315).
- Vehicles may drive at any of the other signal states. When green times overlap, e.g. for conditionally compatible left turns, conflict areas and attribute values take effect that are relevant for mesoscopic simulation (see "Attributes of meso turn conflicts" on page 835).

Non signalized intersections

In mesoscopic simulation intersection control is modeled only using conflict areas and their status. The network object type **Priority Rule** is not taken into account in mesoscopic simulation. Priority rules should not be used in mesoscopic simulation. As different algorithms are used to model movement of the vehicles, meso-specific attributes with individual value ranges are required. The following parameters apply for the behavior of vehicles in conflicting flows:

  - Critical gap (see "Attributes of meso turn conflicts" on page 835)
  - Follow-up gap (see "Attributes of meso turns" on page 833)
  - Maximum wait time (see "Editing the driving behavior parameter Meso" on page 317)

Default values for critical gap and follow-up gap

During ANM import, Vissim sets the following default values. When modeling nodes, you can use these default values for orientation.

For intersection controlled nodes and u-turns the following applies:

Direction	Critical gap [s]	Follow-up gap [s]
left major - minor	3.5	2.2
right minor - major	5.5	3.3
minor intersects major	6.0	4.4
left minor - major	6.5	3.5
U-turn major	3.5 Same as left major - minor	3.5 Same as left minor - major
U-turn minor	6.5 Same as left minor - major	3.5 Same as left minor - major
Channel island (at the end)	5.5 Same as right minor - major	3.3 Same as right minor - major

For uncontrolled nodes the following applies (priority to right):

Direction	Critical gap [s]	Follow-up gap [s]
right minor - minor	5.5 Same as right minor - major	3.3 Same as right minor - major
straight minor - minor	5.5 Same as right minor - major	3.3 Same as right minor - major
left minor - minor	6.5 Same as left minor - major	3.5 Same as left minor - major

For controlled nodes the following applies:

Direction	Critical gap [s]	Follow-up gap [s]
Left turns with counter flow	3.5 Same as left major - minor	2.2 Same as left major - minor
Right turn on red	5.5 Same as right minor - major	3.3 Same as right minor - major

For roundabouts the following applies:

Direction	Critical gap [s]	Follow-up gap [s]
Entry into roundabout	3.5	3.2
Bypass (at the end)	3.5 (as for entry)	3.2 (as for entry)

8.5 Modeling meso network nodes

You may exclusively define and edit meso network nodes (see "Mesoscopic node-edge model" on page 804). You cannot define or edit meso nodes that were automatically generated by Vissim.

Comparable to nodes for dynamic assignment in microsimulation, meso network nodes must be defined where paths come together or significantly branch out.

For dynamic assignment with microsimulation, it is sufficient to use one node per real intersection. This node may contain multiple conflict areas that are independent from each other. These conflict areas only impact vehicles located precisely within their area.

In mesoscopic simulation, vehicles generally wait outside the meso network node due to conflict areas and then traverse the entire meso network node without stopping. This is why a meso network node must start at every position vehicles are meant to wait. If vehicles heading different turning directions are meant to stop at different positions of an intersection entry, you will need to create multiple separate meso network nodes. A single meso node for the entire intersection will not be sufficient.

In mesoscopic simulation you often need more meso network nodes to model intersections than nodes in dynamic assignment with microsimulation, even if the intersections are of similar complexity. This is particularly true when it comes to roundabouts, non- signalized intersections, intersections with bypasses, etc.

Each meso edge leading into a meso network node must be long enough to create a travel time on the node which is greater than the meso critical gaps of the conflict areas for the following turn relations. Otherwise, instead of the critical gap value defined, the travel time on the edge is used as the critical gap. Therefore it is necessary to define small meso network nodes and make sure the edges are sufficiently long enough. Follow the rules and examples given for defining meso network nodes (see "Rules and examples for defining meso network nodes" on page 810).

8.6 Rules and examples for defining meso network nodes

Meso network nodes must be modeled correctly for Vissim to be able to model conflicts realistically in mesoscopic simulation. The level of correctness has a decisive impact on the result of dynamic assignment in mesoscopic simulation. Therefore, the rules for modeling meso network nodes strictly need to be followed accurately. Before starting to model meso network nodes it is necessary to look at the examples and read the descriptions of correct and incorrect definitions for meso network nodes.

8.6.1 Rules for defining meso network nodes

- **Rule 1**: Meso network nodes must be defined everywhere on a link where more than one connector begins or ends.
- **Rule 2**: For each intersection, at least one meso network node must be defined. Depending on the node geometry, several meso network nodes may be defined.

  For non-signalized intersections the following applies: All conflict areas must be defined. In order to decide which conflict areas shall lie within a separate meso network node, check the following:

  - Where is the vehicle supposed to stop? For all turn conflicts, the vehicle stops at the meso network node. Model the meso network node so that its edge roughly corresponds to a stopping position of the vehicle in reality, e.g. a stop line.
  - Are the incoming meso edges used by vehicles with the right of way to reach the meso network node relevant for all turn conflicts in the meso network node? The size and positioning of the node determine which meso edges are perceived as edges with vehicles that have the right of way (see "Meso conflict relevant and non-relevant edges" on page 814).
  - How long is the travel time on the incoming meso edge used by vehicles with the right of way to reach to meso network node? This travel time should be longer then the meso critical gap of the subordinate flow.

For signalized intersections note that the decisive factors are the stop position and storage capacity. The vehicle always stops at the meso network node. If the real situation cannot be modeled with one meso network node only, model several nodes, e.g. for a separately led right turn.

- **Rule 3**: On turn meso edges, the following properties must not change:
  - the number of lanes
  - the link behavior type
  - the meso speed, if the meso speed model **Link-related** is selected (see "Car following model for mesoscopic simulation" on page 803)

This means the **Defining links** of the meso turn must each have the same value (see "Attributes of meso turns" on page 833).

Note: Please note the limitations and information that apply for defining meso network nodes (see "Defining meso network nodes" on page 828).

8.6.2 Examples of applying the rules for defining meso network nodes

The following examples show how the rules are applied when you model intersections. First, you are shown how the position of a meso network node impacts where at the conflict area a vehicle stops and which edges it perceives as relevant:

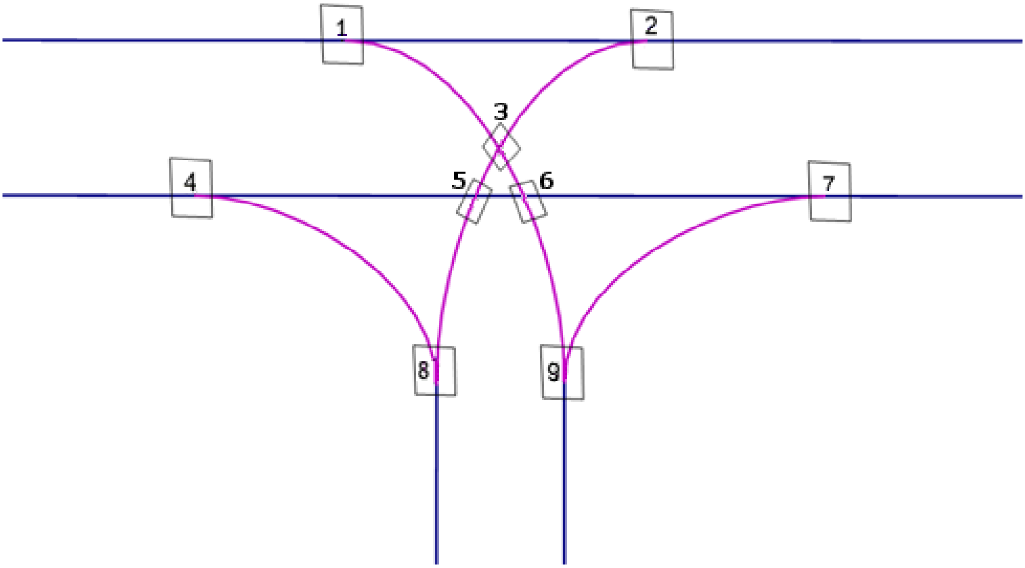
- Consequences of correct and incorrect positioning of meso network nodes
- Meso conflict relevant and non-relevant edges

Then, you are given an explanation of how the rules impact the modeling of different types of intersections. For different network objects, the impact of rules on the meso graph structure and on simulation is demonstrated:

- Nodes in areas where the number of lanes changes
- Modeling connectors in meso network nodes
- Modeling a signalized intersection
- Modeling intersections with lane widening
- Modeling intersections with bypass and channelized turn
- Modeling roundabouts
- Modeling reduced speed areas on links
- Modeling SCs on links

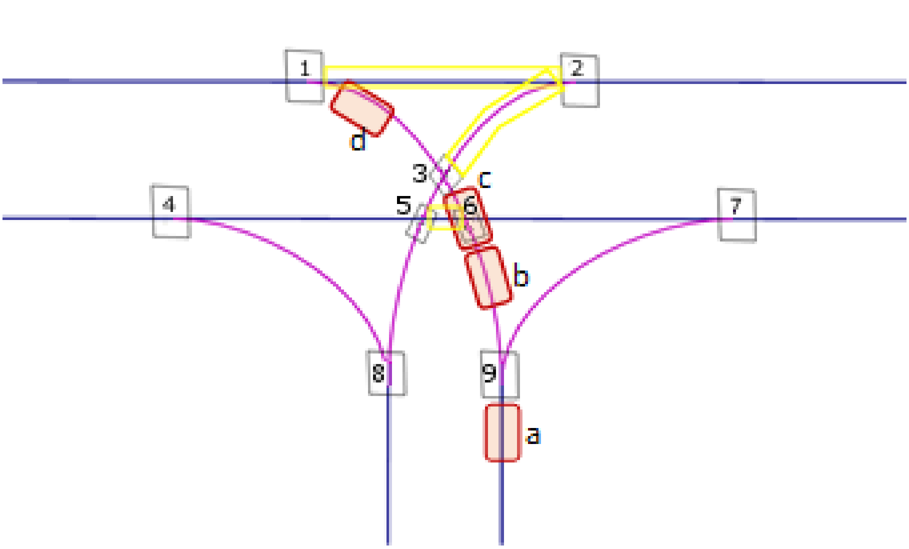
8.6.2.1 Consequences of correct and incorrect positioning of meso network nodes

The following example describes the meaning of travel time as a meso critical gap on an edge for a 3-leg intersection with nine turn conflicts. Nine meso network nodes have been manually defined at the nine turn conflicts (1 to 9):



This type of modeling is not recommended if the travel time on the edge leading into the meso network node is shorter than the meso critical gap of the conflict in the meso network node. This leads to incorrect modeling of the conflicts in mesoscopic simulation. It is illustrated in the following figure and explained in the description given below it.

Situation: The vehicle is coming from below and turns upward left.



The conflicts are not modeled correctly in mesoscopic simulation:

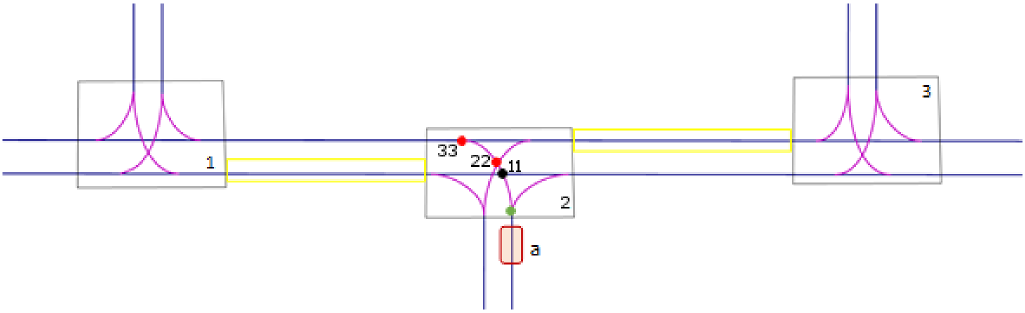
- **Cause**: The travel times on some edges leading into the nodes are too short.
- **Effect**: Vehicle also stops at wrong positions.

- The vehicle stops for the conflict at node **9**, position **a**, as it should.
- The vehicle stops for the conflict at node **6**, position **b**, and only pays attention to the edge between node **5** and **6**. The travel time at the edge between nodes **5** and **6** acts as a critical gap, if it is shorter than the meso critical gap entered for the conflict. The same applies for the subsequent nodes:
- The vehicle stops for the conflict at node **3**, position **c**, and only pays attention to the edge between node **3** and **2**. The travel time at the edge between nodes **3** and **2** acts as a critical gap, if it is shorter than the meso critical gap entered for the conflict.
- The vehicle stops for the conflict at node **1**, position **d**, and only pays attention to the edge between node **1** and **2**. The travel time at the edge between nodes **1** and **2** acts as a critical gap, if it is shorter than the meso critical gap entered for the conflict.

If the vehicle stops at a wrong position and the travel time at the edge leading into the meso network node is very short, the travel time acts as a critical gap. Vissim is then unable to model the conflicts in mesoscopic simulation realistically (as illustrated in the figure above).

If, for instance, no meso network node is defined for node **3** (at top of figure), Vissim does not recognize the conflict there and the conflict is ignored in mesoscopic simulation.

Solution: If for these types of intersections, with short edges between conflicts, only one meso network node is defined, Vissim is able to model conflicts realistically in mesoscopic simulation. With one meso network node only, the left-turning vehicle has only one stop position in all subsequent conflicts. The travel times at the incoming edges are long enough and the vehicle stops at the correct position. This is illustrated in the following figure and explained in the description given below it.



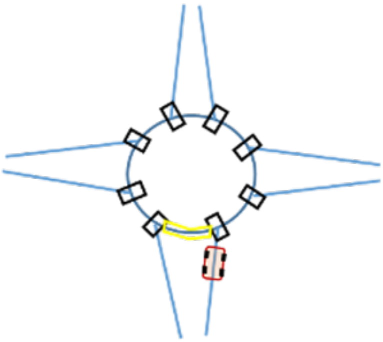
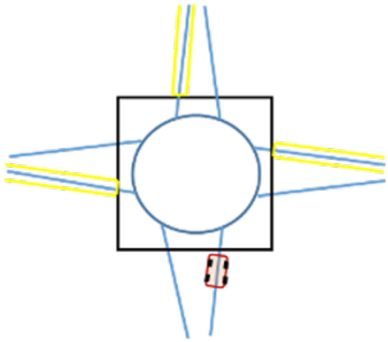
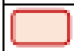
Correct modeling: The travel times at all edges leading into the node at conflict points are long enough. This ensures that the vehicle stops at the correct positions:

- With conflict **11** in the black dot, the vehicle is aware of the edge leading from node **1** into node **2**. If the travel time on this edge is longer than the meso critical gap for the conflict, the specified value is used as critical gap, e.g. 3.5 s.
- With conflicts **22** and **33** in the red dots, the vehicle is aware of the incoming edge between nodes **3** and **2**. If the travel time on this edge is longer than the meso critical gap for the conflict, the specified value is used as critical gap, e.g. 3.5 s.

8.6.2.2 Meso conflict relevant and non-relevant edges

This example shows a roundabout (right-hand traffic) for which multiple meso network nodes have been correctly positioned, in the figure on the left. In the figure on the right, only one meso network node has been positioned across the roundabout. The following two figures show the meso edges the vehicle is aware of when it stops at the meso network node:

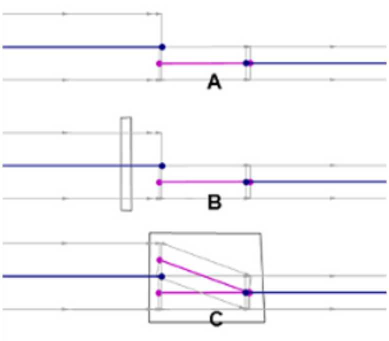
- **Correctly modeled**: The modeling in the figure on the left ensures that the vehicle is aware of the relevant meso edge (yellow between the two meso network nodes), leading directly into the correctly positioned meso network node at which the vehicle stops. The correct meso critical gap is used.
- **Incorrectly modeled**: The modeling in the figure on the right does not allow the vehicle to become aware of the relevant meso node. For conflicts in the meso network node, e.g. the entry of the vehicle into the roundabout, the vehicle is only aware of non-relevant meso edges (the three meso edges highlighted in yellow that lead into the meso network node from the left, top and right). The vehicle cannot become aware of the relevant node as shown in the figure on the left. Thus, it cannot take a correct meso critical gap into account. The vehicle stops at the meso network mode and gives priority to the vehicles coming from the right, top and left, as it is only aware of their meso nodes.

Correct: Vehicle is aware of relevant meso edge (yellow between the two bottom meso network nodes)	Incorrect: Vehicle is only aware of non-relevant meso nodes (yellow)
	
: Vehicle is coming from below and wants to turn right into roundabout	

8.6.2.3 Nodes in areas where the number of lanes changes

There are different ways to model areas in which the number of lanes changes. These impact dynamic assignment in mesoscopic simulation in different ways. This is illustrated in the following figure and explained in the table listed below it.

Connector connects a double-lane link with a single-lane link:

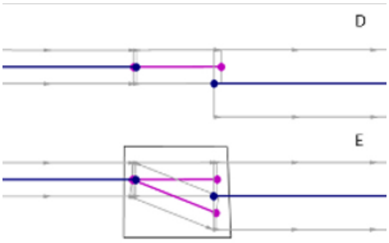


The vehicle may only change lane at the end of a meso node. This applies for meso nodes generated automatically by Vissim and for modeled meso network nodes (see "Mesoscopic node-edge model" on page 804).

Modeling	Situation	Mesoscopic simulation

A	- a connector - not a modeled meso network node	- Vissim automatically generates a meso node at the beginning of the connector. - On the double-lane link, vehicles may use the right lane only which is unrealistic.
B	- a connector - a modeled meso network node	- Vissim automatically generates a meso node at the beginning of a connector. In this case, you need not manually define the meso network node. This meso network node is defined manually, so that contrary to **A**, both lanes may be used and lane changes are possible. - On the double-lane link, vehicles may use both lanes. Vehicles may use the left lane up until the modeled meso network node. At the end of this meso network node, all vehicles must change from the left lane to the right lane.
C	- two connectors - a modeled meso network node	- On the double-lane link, vehicles may use both lanes. - For dynamic assignment when one of the connectors is closed then only one connector is available for the path search. However for mesoscopic simulation both connectors remain available. - Apply an edge closure to one of the edges for dynamic assignment. This way you can avoid parallel edges in dynamic assignment. Parallel edges multiply the number of possible paths significantly.

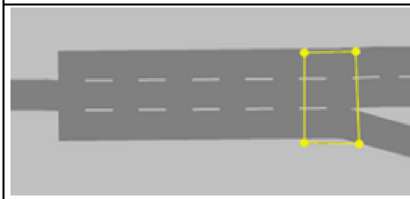
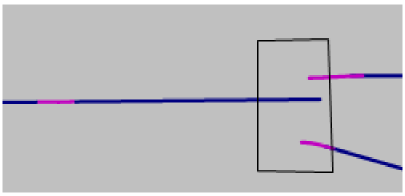
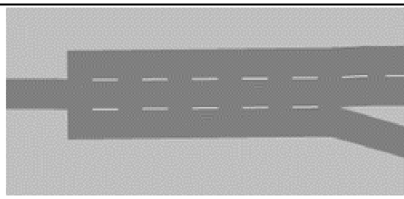
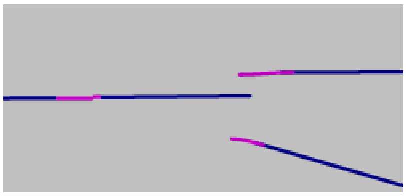
Connector connects a single-lane link with a double-lane link:



Modeling	Situation	Mesoscopic simulation
D	- a connector - not a modeled meso network node	- Vissim automatically generates a meso node at the end of the connector. - Vehicles can use both lanes of the double-lane link. To use the lane on the right, the vehicle must change lanes. Lane changes are penalized during lane selection. This is why the left lane is preferred.

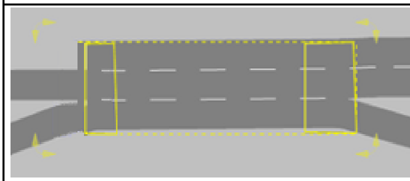
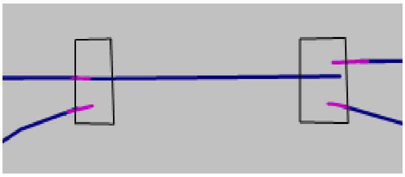
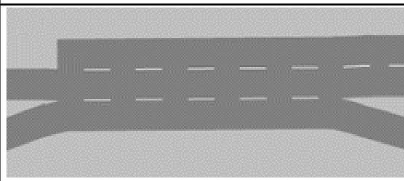
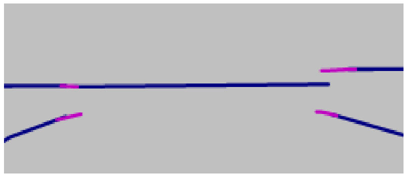
E	- two connectors - a modeled meso network node	- On the double-lane link, vehicles may use both lanes. As both lanes can be easily reached, no lane change is required and the vehicles are distributed evenly across the lanes, if both lanes are permitted for the vehicle route. - Apply an edge closure to one of the edges for dynamic assignment. This way you can avoid parallel edges in dynamic assignment. Parallel edges multiply the number of possible paths significantly.

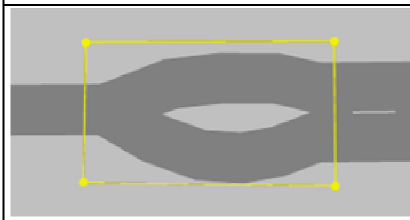
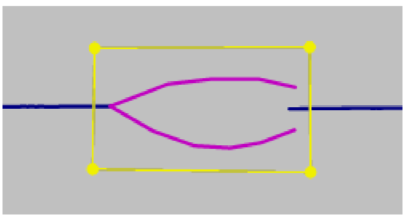
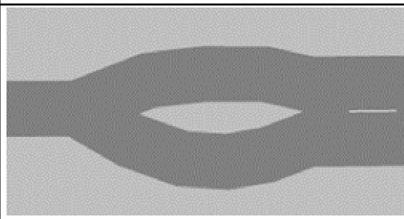
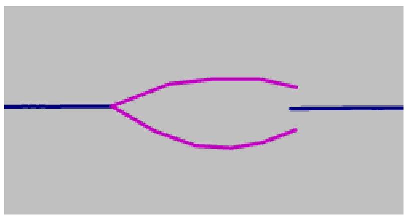
8.6.2.4 Modeling connectors in meso network nodes

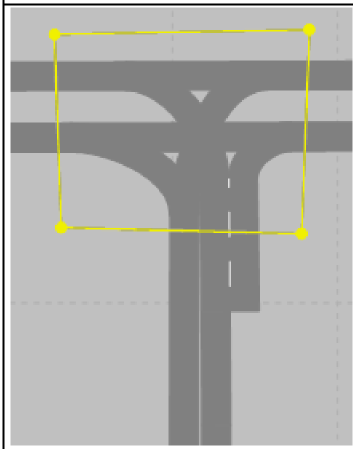
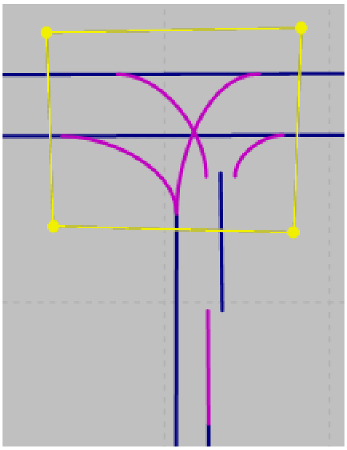
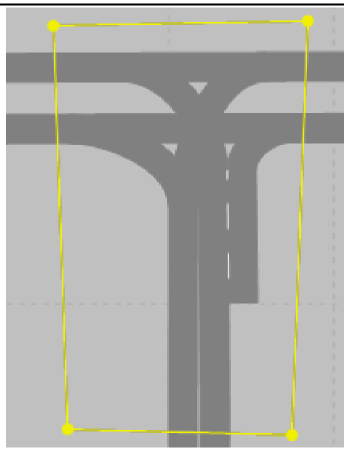
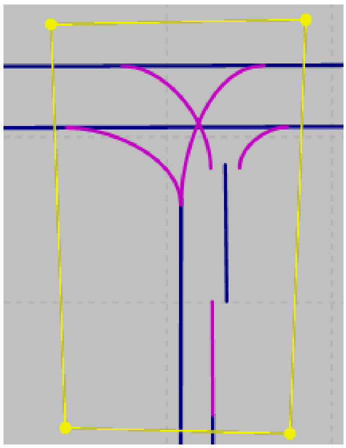
<table>
<tr><th>Rule</th><th colspan="2">Description</th></tr>
<tr><td>1</td><td colspan="2">Meso network nodes must be defined everywhere on a link where more than one connector begins or ends.</td></tr>
<tr><td rowspan="3"></td><td colspan="2">- A link leads into a node.
- Two connectors lead out of the node.
- The connectors do not have to lie entirely within the node.
- The connectors must begin within the node.</td></tr>
<tr><td>Correct</td><td>False</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

<table>
<tr><th>Rule</th><th colspan="2">Description</th></tr>
<tr><td>1</td><td colspan="2">Meso network nodes must be defined everywhere on a link where more than one connector begins or ends.</td></tr>
<tr><td rowspan="3"></td><td colspan="2">

The connectors do not have to lie entirely within the node.
Left meso node: Two connectors lead into node. The connectors must end within the node.
Right meso node: Two connectors lead out of node. The connectors must begin within the node.

</td></tr>
<tr><td>Correct</td><td>False</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

<table>
<tr><th>Rule</th><th colspan="2">Description</th></tr>
<tr><td>1</td><td colspan="2">Meso network nodes must be defined everywhere on a link where more than one connector begins or ends.</td></tr>
<tr><td rowspan="3"></td><td colspan="2">If the transition from a one-lane link to a two-lane link is modeled across two connectors, these must lie entirely within the node.</td></tr>
<tr><td>Correct</td><td>False</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

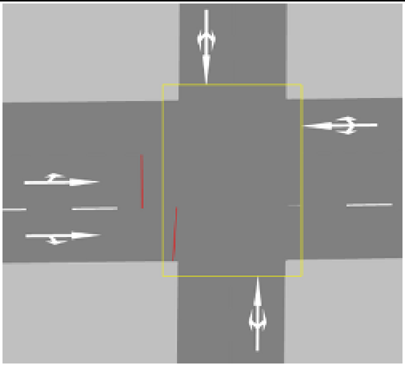
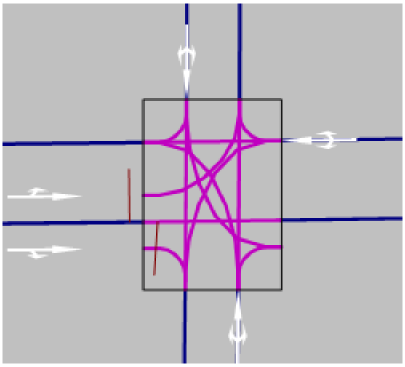
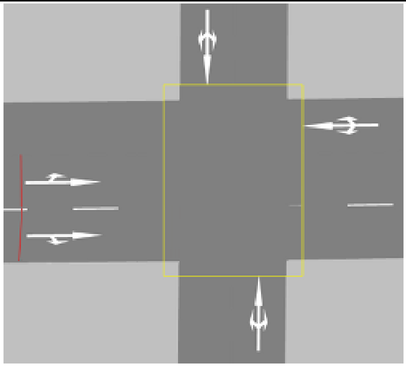
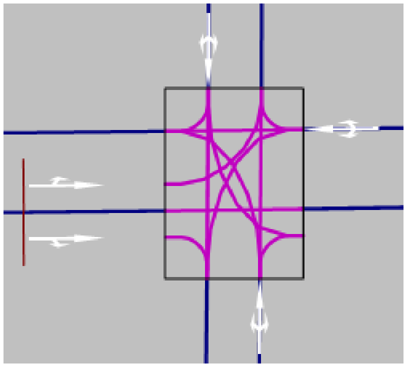
<table>
<tr><th>Rule</th><th colspan="2">Description</th></tr>
<tr><td>3</td><td colspan="2">On turn meso edges, the following properties must not change:
the number of lanes
the link behavior type
the meso speed, if the meso speed model Link-related is selected (see "Car following model for mesoscopic simulation" on page 803)</td></tr>
<tr><td rowspan="3"></td><td colspan="2">The number of lanes must be the same for each defining link contained within the meso network node.
However, the number of lanes of the inbound meso edge and the number of lanes of the outbound meso edge may differ.</td></tr>
<tr><td>Correct</td><td>False</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

8.6.2.5 Modeling a signalized intersection

<table>
<tr><th>Rule</th><th colspan="2">Description</th></tr>
<tr><td>2</td><td colspan="2">For each intersection, at least one meso network node must be defined. Depending on the node geometry, several meso network nodes may be defined.</td></tr>
<tr><td></td><td colspan="2">

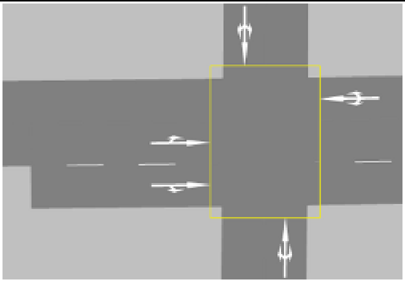
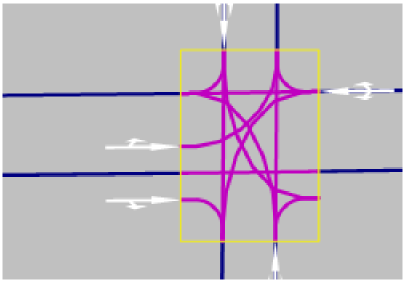
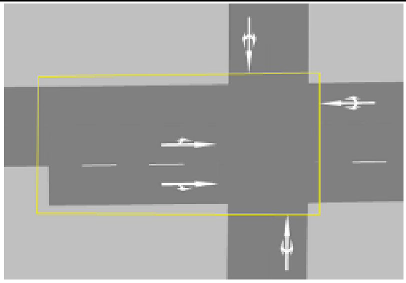
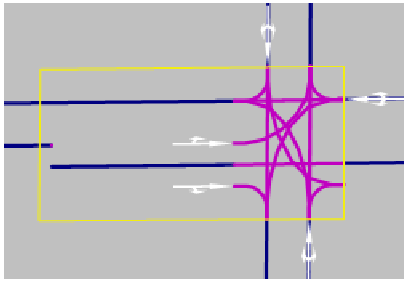
A signal head may be defined on links or connectors.
Position signal heads within a meso network node.
If a signal head is less than 5 m away from the border of the node, the software will not automatically generate another meso node. Vissim then assumes that the vehicle stops at the node border and the signal head belongs to the node.
If a signal head is more than 5 m away from the border of a node, Vissim automatically generates an additional meso node.
If meso nodes are positioned too close to each other, the edge between them might become so short that, in certain situations, Vissim cannot not model the driving behavior realistically. Vissim only accounts for the vehicles at the edge leading into the node, not any other nodes further downstream.

In the figure at the bottom right this means:
If the signal heads are 10 m from the meso network node of the intersection, Vissim automatically generates a meso node at the signal heads. The edge between the two nodes is then 10 m long. The travel time of a vehicle driving at 10 m/s on this edge is 1 s. This second acts as a critical gap for the vehicle approaching from the right and turning left, regardless of the actual meso critical gap defined, as the vehicle cannot tell whether, beyond the meso node, there is a vehicle approaching from the left that it must yield to. A critical gap of 1 s does not give the vehicle enough time to yield.
Solutions: a) Position the signal heads within the meso network node or b) reduce the distance between signal heads and meso network nodes to below 5 m or c) ensure that the length of the edge leading into the node is long enough to create a travel time on the edge that is longer than the meso critical gap of the turn conflict in the node.
</td></tr>
<tr><td></td><td>Correct: The signal heads are positioned within the meso network node or at a maximum of 5 m from it</td><td>Not recommended: The signal heads are positioned at a distance of more than 5 m from the meso network node</td></tr>
</table>

Rule	Description	
	 	 

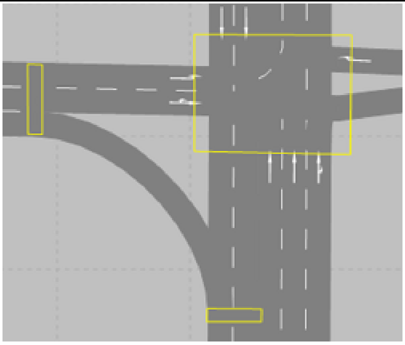
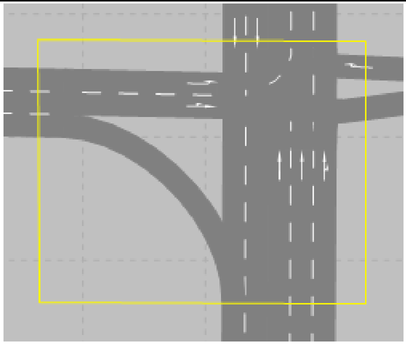
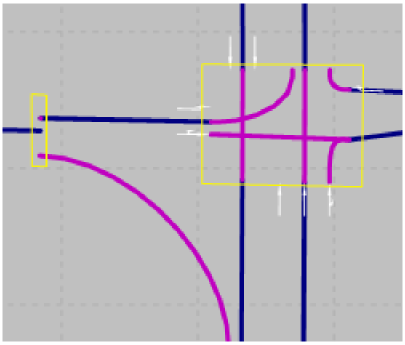
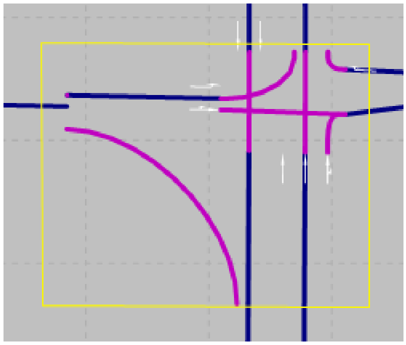
8.6.2.6 Modeling intersections with lane widening

Rule	Description
2	For each intersection, at least one meso network node must be defined. Depending on the node geometry, several meso network nodes may be defined.
	• Lane widening must not lie within the node. • When creating a meso graph, Vissim automatically generates a meso node of the type **Other** where the lane widening begins (see "Attributes of meso nodes" on page 829).

Rule	Description	
	Correct	False
	 	 

8.6.2.7 Modeling intersections with bypass and channelized turn

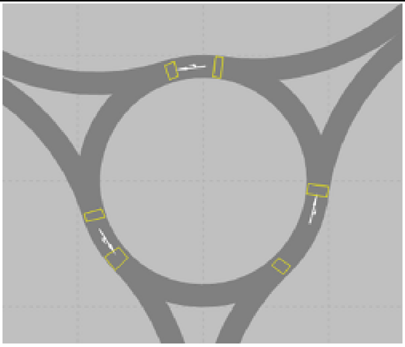
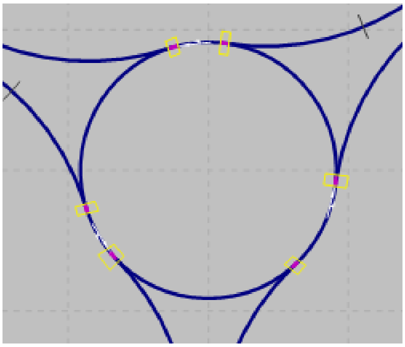
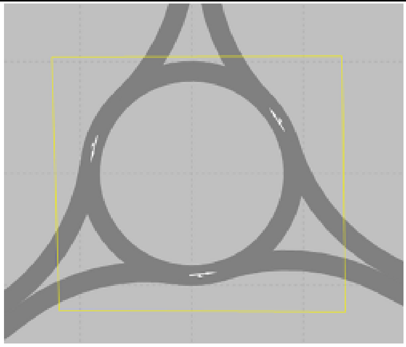
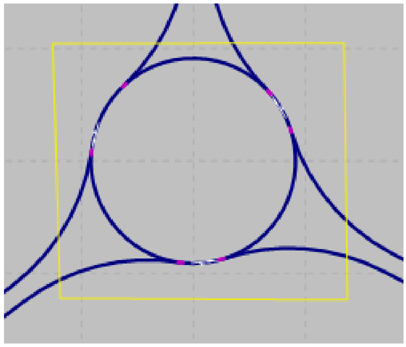
Rule	Description
2	For each intersection, at least one meso network node must be defined. Depending on the node geometry, several meso network nodes may be defined.
	➤ According to **Rule 1** meso network nodes must be placed at the branchings where the bypass begins and ends. These nodes must have the attribute **Use for mesoscopic simulation**. In the figure on right, the vehicle approaching from the left and turning to the right (downwards), already waits at the beginning of the right turn lane, at the large node that represents the entire intersection. This is not recommended. In the figure on the left, the vehicle waits at the end of the right turn lane, at the small meso network node modeled for this purpose. ➤ In addition, the intersection itself must lie within a meso network node.

Rule	Description	
	Correct	**Not recommended**
		
		

8.6.2.8 Modeling roundabouts

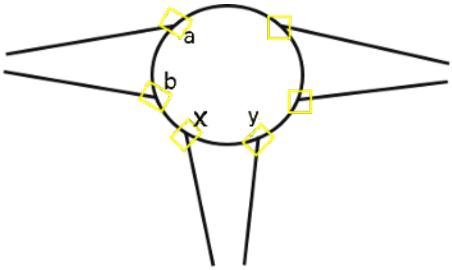
Note: The following tips refer to the modeling of simple roundabouts, e.g. those with a single lane, with no or only one bypass and few entries and exits. To model more complex roundabouts or roundabouts whose conflicts cannot be modeled correctly in mesoscopic simulation, define sections and perform a hybrid simulation (see "Hybrid simulation" on page 837).

Rule	Description
2	For each intersection, at least one meso network node must be defined. Depending on the node geometry, several meso network nodes may be defined.
	➤ Each branching and thus each entry and exit must lie within a meso network node. ➤ The connectors do not have to lie entirely within the node. ➤ If there is a bypass, meso network nodes must be placed at the branchings where the bypass begins and ends. These nodes must have the attribute **Use for mesoscopic simulation**.

Rule	Description	
	Correct	False
	 	 

Model meso network nodes for a roundabout depending on the distance between the exit and the next entry and according to **approach A** or **approach B**. The is illustrated in the figures and their descriptions below:

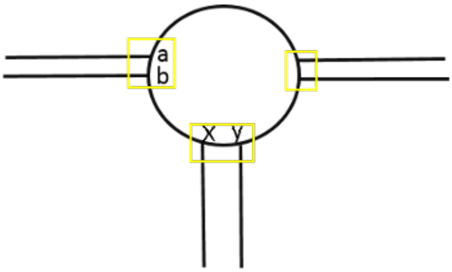
Description of approach A
If the distance between the exit and next entry downstream is large enough, define a meso network node for the exit and another one for the entry. This is the case in the following situations: - **Condition 1**: The travel time on the roundabout between exit **x** and the next entry downstream **y** is equal to or larger than the meso critical gap for the conflict in **y**: $t_{xy} > t_{CG(y)}$. If this condition is not met, but the following one is, you can still use approach A to model the roundabout: - **Condition 2** (for right-hand traffic): The travel time on the roundabout between exit **x** and the next entry downstream **y** is equal to or larger than the travel time on the lane between upstream entry **b** and the next downstream exit **x**: $t_{bx} < t_{xy}$. For the conflict in **y**, the critical gap is the travel time on the roundabout between exit **x** and the next entry downstream **y**.

Correct
 $t_{xy} > t_{CG(y)}$ or $t_{bx} < t_{xy}$

When you export a network from Visum and import it into Vissim via ANM import, Vissim automatically generates meso network nodes based on approach A. These nodes do not require any subsequent editing (see "Generated network objects from the ANM import" on page 372). The table lists different speeds to illustrate the minimum distance between exit **x** and the next downstream entry **y** with a critical cap of 3.5 s, in order for approach A to meet condition 1:

Veh speed on roundabout lane		**min. distance [m] x-y to meet condition 1**
m/s	**km/h**	
1	3.6	3.5
2	7.2	7.0
3	10.8	10.5
5	18	17.5
7	25.2	24.5
10	36	35.0
14	50.4	49

Description of approach B
If the distance between the exit and next entry downstream is not large enough, define a common meso network node for both the exit and entry. This is the case, when the two following situations happen at the same time: - The travel time on the roundabout lane between entry **b** and the next downstream exit **x** is larger than the travel time between exit **x** and the next exit downstream **y**: $t_{bx} > t_{xy}$ and - the travel time on the roundabout lane between exit **x** and the next entry downstream **y** is smaller than the meso critical gap for the conflict in **y**: $t_{xy} < t_{CG(y)}$

Correct
 $t_{bx} > t_{xy}$ and $t_{xy} < t_{CG(y)}$

The two following figures show wrong approaches to define meso network nodes: These approaches produce incorrect results when used to model conflicts in mesoscopic simulation:

Incorrect approach 1: The distance between the entry and the next exit downstream is not large enough. As a result, too many conflicts arise at each of the nodes:

- 4 meso turn edges:
  - from roundabout
  - from entry
  - into roundabout
  - into exit
- 6 meso turn conflicts

Solution: If the entry and the next exit downstream are very close to each other, use approach A, even if this results in short edges between the meso network nodes. In that case, at each of the two meso network nodes, there will be only one merging or branching conflict. At the branching conflict, the short edge leading out of the meso network node does not pose a problem.

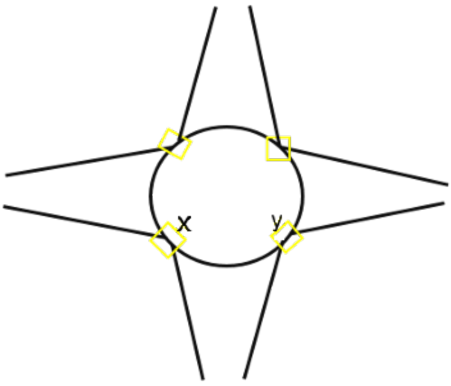
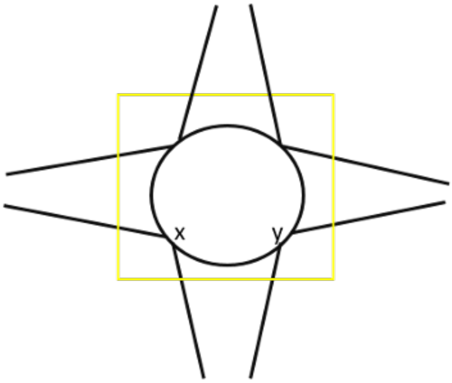
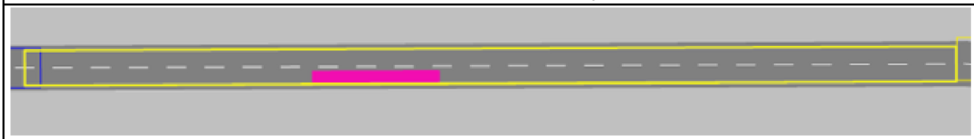
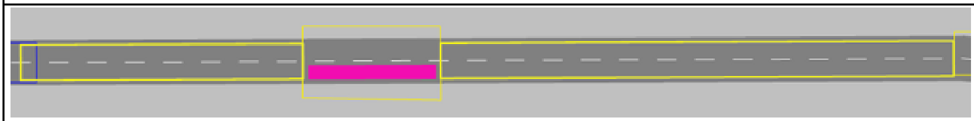
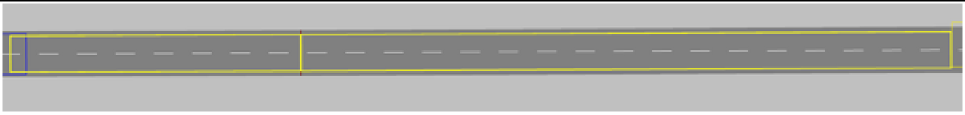


Figure below **Incorrect approach 2**: Only a single meso network node has been defined for all conflicts in the roundabout. Effect: Conflicts are not modeled realistically, vehicles stop at wrong positions and for conflicts, the time gap is based on non-relevant edges.


8.6.2.9 Modeling reduced speed areas on links

Description
- Reduced speed areas on links are only taken into account when the vehicle-based meso speed model is used on links. - In the top figure, the speed distribution specified for the reduced speed area has an impact on the entire meso node for vehicles heading from west (left) to east (right) on the right lane. - If you want the reduced speed area to have only a local impact, you need to insert a meso network node (see "Defining nodes" on page 708). The reduced speed area must lie entirely within the node. The meso edge within the node is a turn meso edge that is impacted by the speed distribution of the reduced speed area. - Desired speed decisions are treated the same way in mesoscopic simulation.



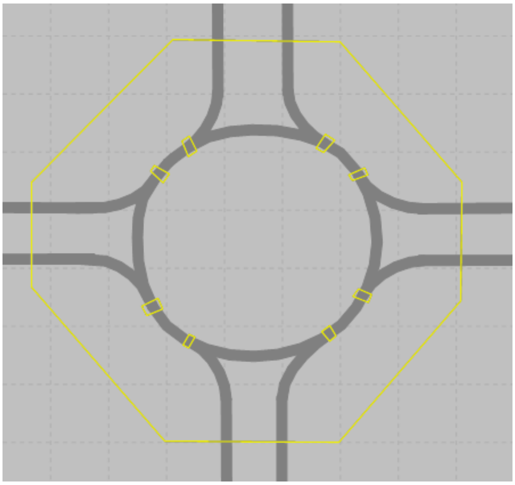
8.6.2.10 Modeling SCs on links

Description
• You do not have to create nodes for signal heads on links. • In this case, a meso node of the type **Other** is automatically generated on the link (see "Attributes of meso nodes" on page 829). When doing so, Vissim automatically generates two meso edges.


8.7 Defining meso network nodes

Please note the rules and examples for defining nodes and meso network nodes (see "Modeling nodes" on page 705), (see "Modeling meso network nodes" on page 809). In addition, consider the following limitations and information:

- Limit the extent of a meso network node to the necessary maximum extent. Conflict areas in particular do not have to lie entirely within the node.
- Ensure that there is a sufficient distance between the meso nodes. This includes the distance between automatically generated meso nodes that are not of the type A **Node** (see "Attributes of meso nodes" on page 829).
- Meso network nodes must not overlap.
- When modeling meso network nodes, please note that during dynamic assignment, path search and path selection are performed based on the generalized costs of edges and/or paths of dynamic assignment. When creating networks for mesoscopic simulation in Vissim, you therefore generally select the **UseForDynAssign** attribute (see "Defining nodes" on page 708).
- In networks imported via ANM import, for some intersections (e.g. roundabouts or intersections with lane widening) nodes are created with either the **UseForDynAssign** attribute or the **UseforMeso** attribute selected. Nodes with the **UseForDynAssign** attribute are exclusively used for calculations performed within dynamic assignment and the path file. These nodes must also not overlap. In principle, the attributes **UseForDynAssign** and **UseForMeso** may be selected independently from each other.
- You can use nodes of dynamic assignment to control the path selection based on edge closures (in the figure below, the outer node) and meso network nodes (in the figure below, the eight node in the roundabout), as described in the respective rules (see "Rules and examples for defining meso network nodes" on page 810):



1. Define the desired meso network nodes according to the definition of nodes (see "Modeling nodes" on page 705), (see "Defining nodes" on page 708).

   *The **Nodes** window opens.*

2. In the **Nodes** window, select **Use for mesoscopic simulation**.

   *The node then becomes a meso node. When creating a meso graph, Vissim automatically generates a **meso node** for each meso network node modeled and assigns it the attribute **Node** (see "Attributes of meso nodes" on page 829).*

8.8 Attributes of meso nodes

Vissim automatically generates a meso node when a meso graph is generated (see "Generating meso graphs" on page 837), (see "Mesoscopic node-edge model" on page 804). You can display these meso nodes and their attributes in the **Meso nodes** list.

When generating a meso graph, Vissim assigns each meso node in the **Type** attribute a value. This value is based on the network object type of the network object Vissim is generating the meso node for.

Example:

In the Network editor, you define a node that you need for mesoscopic simulation. For this node, you consequently select the **Use for mesoscopic simulation** attribute. The node then becomes a meso network node. You can show the meso network node, like all models nodes, in the **Nodes** list (see "Attributes of nodes" on page 709). When generating the meso graph, Vissim generates a meso node for the meso network node. Vissim then assigns the meso node the value **Node** in the **Type** attribute. The generated meso node is displayed in the **Meso nodes** list., but the modeled meso network is not.

- From the **Lists** menu, choose > **Network** > **Meso nodes**.

*The **Meso nodes** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Long name	Short name	Description
Number	**No**	Unique number
Type	**Type**	Vissim assigns the type, while generating the meso node, based on the network object type for which it is generating the meso node. The length of automatically generated meso nodes that are not of the type **Node** is 0.000 m.
Vissim can create the following types:		
➤ **PT stop lay-by entry**: Meso nodes at the beginning of connector that lead to the link with the PT stop lay-by		
➤ **PT stop lay-by exit**: Meso nodes at the beginning of connector that lead away from the link with the PT stop lay-by		
➤ **PT lines origin**: Meso node at the beginning of a PT line, leads to a meso edge of the type **PT input connector**.		
➤ **Parking lot**: Meso node within a parking lot of teh type **Zone connector**		
➤ **Vehicle input origin**: Meso node at the beginning of a link on which vehicles are deployed into the network, leads to a meso edge		
➤ **Origin zone**: Meso node for assigned origin zone in parking lot		
➤ **Routing decision**: Meso node of a dynamic routing decision		
➤ **Node**: Meso node located at a node of the network object type **Node**, for which the attribute **Use for mesoscopic simulation** is selected		
➤ **Other**: Other automatically generated meso nodes for signal heads on links or where link attributes change that are relevant for mesoscopic simulation (e.g. number of lanes)		
➤ **Transition meso-micro**: Meso node at the border of a section where transition from mesoscopic to microscopic simulation takes place		
➤ **Transition micro-meso**: Meso node at the border of a section where transition from microscopic to mesoscopic simulation takes place (end point of meso edge of the type **Micro-meso transition**)		
➤ **Public transport stop**: Meso node within a PT stop. When a PT vehicle is on a meso turn with a PT stop, no other vehicle may enter the same meso turn. A vehicle may, however, pass the stopping PT vehicle on the adjacent lane, if its use is permitted for the vehicle.		

Long name	Short name	Description
- **Transition micro-meso (virtual)**: Virtual meso node at the border of a section where transition from microscopic to mesoscopic simulation takes place (start point of meso edge of the type **Micro-meso transition**)		
- **Destination zone**: Meso node for assigned destination zone in parking lot		
- **Input**: Meso node where PT vehicles are deployed in the network		

Long name	Short name	Description
Nodes	**Nodes**	Number of respective node of the network object type **Node**
Parking Lot	**ParkLot**	Number of parking lot in which the meso turn lies
Public transport stop	**Public transport stop**	Number of public transport stop
Turn meso edges	**TurnMesoEdge**	Numbers of turn meso edges
PT lines (inputs)	**PTLinInput**	List of PT lines that begin at the meso node
Inbound meso edges	**InbMesoEdge**	Numbers of the meso edges that lead to a meso edge of the meso node
Outbound meso edges	**OutbMesoEdge**	Numbers of meso edges that exit the meso edge of the meso node
Dynamic vehicle routing decision	**VehRoutDecDyn**	Numbers of the dynamic vehicle routing decisions at meso edges of the type **Routing decision**

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Turn meso edges: These are meso edges of the type **Turn** (see "Attributes of meso edges" on page 832)
- Outbound meso edges: List of meso edges, leading out of the meso node (see "Attributes of meso edges" on page 832)
- Inbound meso edges: List of preceding meso edges that lead into the meso node (see "Attributes of meso edges" on page 832)
- Meso turn conflicts: (see "Attributes of meso turn conflicts" on page 835)
- Meso turns: Meant for entry of follow-up gap (see "Attributes of meso turns" on page 833)
- PT lines (inputs): List of PT lines that begin at the meso node (see "Attributes of PT lines" on page 520)

- Signal head: List of signal heads in meso node (see "Attributes of signal heads" on page 579)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

8.9 Attributes of meso edges

- From the **Lists** menu, choose > **Network** > **Meso edges**.

  *The **Meso edges** list opens.*

The list on the left may include the following attributes:

<table>
<tr><th>Long name</th><th>Short name</th><th>Description</th></tr>
<tr><td>Number</td><td>No</td><td>Unique meso edge number</td></tr>
<tr><td>From meso node</td><td>FromMesoNode</td><td>Number of meso node at which the meso edge begins</td></tr>
<tr><td>To meso node</td><td>ToMesoNode</td><td>Number of meso node at which the meso edge ends</td></tr>
<tr><td>Length</td><td>Len</td><td>Length of meso edge [m]</td></tr>
<tr><td>Meso lane</td><td>MesoLn</td><td>Meso-specific lane numbers</td></tr>
<tr><td>Type</td><td>Type</td><td>Vissim distinguishes between the following types of meso edges:</td></tr>
<tr><td colspan="3">• Micro-meso transition: Meso edge lies at the border of a section where transition from microscopic to mesoscopic simulation takes place. The length is 0.000 m.</td></tr>
<tr><td colspan="3">• Meso link: The meso edge lies between two meso nodes. This includes between two meso nodes of the type PT stop lay-by entry and PT stop lay-by exit.</td></tr>
<tr><td colspan="3">• Turn: The meso edge lies within a meso node.</td></tr>
<tr><td colspan="3">• Origin connector: Origin connector meso edge in parking lot. An origin connector meso edge does not have a preceding edge.</td></tr>
<tr><td colspan="3">• Destination connector: Destination connector meso edge in parking lot. A destination connector meso edge does not have a successive edge.</td></tr>
<tr><td colspan="3">• PT input connector: Meso edge at start section of public transport line</td></tr>
<tr><td colspan="3">• Vehicle input connector: Meso edge at vehicle input</td></tr>
</table>

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Outbound meso nodes: List of the successive meso nodes
- Inbound meso nodes: List of the preceding meso nodes
- Reduced speed areas (see "Attributes of reduced speed areas" on page 437)
- Meso lanes: Meso-specific lane numbers of meso edge
- Signal heads: Numbers of the signal heads at meso edge
- Link sequence: Numbers of links and connectors that lead via meso edge (see "Attributes of links" on page 409)
- Desired speed decisions (see "Attributes of desired speed decisions" on page 441)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

8.10 Attributes of meso turns

- From the **Lists** menu, choose > **Intersection control** > **Meso turns**.

  *The **Meso turns** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Long name	Short name	Description
Defining links	**DefLink**	Links and connectors of the meso turn. The defining links are specified depending on the number of links in the link sequence. If a link sequence contains only one link, it is the defining link. If a link sequence contains two links, the connector is the defining link. If a link sequence contains three or more links, all links and connectors included in their entity are defining links.
Nodes	**Nodes**	Number of node in which the meso turn lies
Meso edge	**MesoEdge**	Number of meso edge

Long name	Short name	Description
Meso node	**MesoNode**	Number of meso node in which the meso turn lies
From link	**FromLink**	Number of link or connector at which the meso turn begins
To link	**ToLink**	Number of link or connector the meso turn leads to
Link sequence	**LinkSeq**	Number of links or connectors via which the meso turn leads. Contrary to the defining links, this also includes the **From link** and **To link**.
Meso follow-up time	**MesoFollowUpGap**	The meso follow-up gap determines the maximum capacity (*3,600 s / meso follow-up gap*) of a subordinate flow within a node, as long as there is no traffic on a route that leads to conflict. The follow-up gap only becomes effective, if it is greater than the temporal distance between two successive vehicles that has been defined in the car following model. The meso follow-up gap is a link attribute (see "Attributes of links" on page 409). The meso follow-up gap only has an impact on simulation, if it is greater than the temporal distance specified in the car following model. Only then is the capacity reduced. When defining values for the follow-up gap, you can refer to established manuals such as HBS or HCM. Edit this attribute in the **Meso turns** list or in the coupled list **Nodes** - **Meso turns** (see "Attributes of nodes" on page 709). The values are saved to the defining links as **Meso follow-up gap**. The default value is 0.0 s: When you use the default value, only the car following model takes effect (see "Car following model for mesoscopic simulation" on page 803). Value range 0 - 1,000 s.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Defining links: List of the meso turn links and connectors that are significant for the behavior of vehicles in mesoscopic simulation (see "Attributes of links" on page 409)
- Link sequence: List of all links and connectors of the meso turn (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

8.11 Attributes of meso turn conflicts

The **Meso Turn Conflicts** list lists the conflicts relevant for mesoscopic simulation.

- From the **Lists** menu, choose > **Intersection control** > **Meso turn conflicts**.

  The ***Meso turn conflicts*** *list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Long name	Short name	Description
Nodes	**Nodes**	Number of network node in which the turn conflict lies
Meso node	**MesoNode**	Number of meso node in which the turn conflict lies
Turn meso edge 1	**TurnMesoEdge1**	Number of first turn meso edge
From link 1	**FromLink1**	Number of FromLink of first meso turn
To link 1	**ToLink1**	Number of ToLink of first meso turn
Turn meso edge 2	**TurnMesoEdge2**	Number of second turn meso edge
From link 2	**FromLink2**	Number of FromLink of second meso turn
To link 2	**ToLink2**	Number of ToLink of second meso turn
Conflict areas	**ConflictArea**	Numbers of respective conflict areas

Long name	Short name	Description
Status	**Status**	(see "Attributes of conflict areas" on page 565). When changes are made to the status, the value of the respective conflict areas is saved.
Meso critical gap	**MesoCriticGap**	**Meso critical gap**: The meso critical gap defines the temporal distance between two successive vehicles in the main flow that a vehicle in a subordinate flow needs to enter the node. The meso critical gap extends from the back edge of the preceding vehicle to the front edge of the tailing vehicle. The time required for the effective length of the preceding vehicle is not included. This allows you to also model realistic behavior when dealing with a large share of overlong vehicles. When defining values for the meso critical gap, you can refer to established manuals such as the HBS or HCM. In deviation from the definition used here, in the manuals, the meso critical gap is defined for between the front edges of successive vehicles. The meso critical gap is a conflict area attribute (see "Attributes of conflict areas" on page 565). Edit this attribute in the **Meso Turn Conflicts** list or in the coupled list **Nodes** - **Meso Turn Conflicts** (see "Attributes of nodes" on page 709). The values are saved with the respective conflict areas as the attribute meso critical gap (see "Attributes of conflict areas" on page 565). Default 3.5 s, value range 0 s to 1,000 s.

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Conflict areas: List of respective conflict areas (see "Attributes of conflict areas" on page 565)
- Link sequence 1: List of links of the first meso turn (see "Attributes of links" on page 409)
- Link sequence 2: List of links of the second meso turn (see "Attributes of links" on page 409)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

8.12 Generating meso graphs

A meso graph is automatically calculated when you start mesoscopic simulation. You may also create this graph interactively to show the data of meso-specific network objects in the attribute lists. When changes are made to the network that affect the meso graph, it is automatically deleted and the results are discarded. The results lists and attribute lists of meso-specific network objects then no longer list any data.

1. From the **Lists** menu, choose > **Network**. Then select the entry of your choice:

- Meso edges (see "Attributes of meso edges" on page 832)
- Meso nodes (see "Attributes of meso nodes" on page 829)
- Meso turns (see "Attributes of meso turns" on page 833)
- Meso turn conflicts (see "Attributes of meso turn conflicts" on page 835)

  If the meso graph has not been created yet, the list will display the following message: ***No <Name Meso network object> available. Please use the shortcut menu to create the meso graph****.*

2. Right-click in the list.
3. From the shortcut menu, choose **Create Meso Graph**.

Vissim calculates the meso graph and shows the attribute values in the attributes list. Attribute values are also available in the the lists of other network objects that are relevant for mesoscopic simulation.

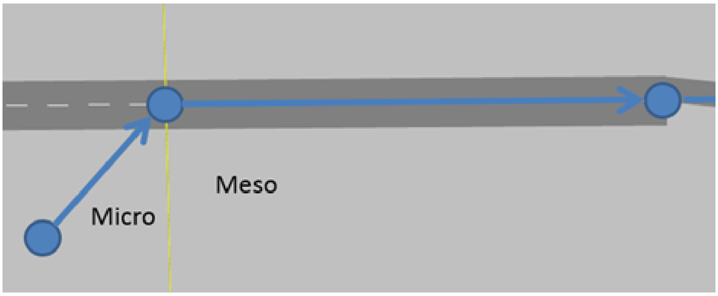
8.13 Hybrid simulation

During dynamic assignment, you can use mesoscopic simulation to simulate your entire network. At the same time, you may also use microscopic simulation to simulate one or several areas of your Vissim network. This combination allows you to create a Vissim network of the size required for your particular use case and perform a detailed analysis for the results of the relevant network parts based on microscopic simulation.

To define the areas you want to simulate microscopically, you define sections (see "Modeling sections" on page 677). You select these sections in the simulation parameters for mesoscopic simulation (see "Selecting sections for hybrid simulation" on page 838). Within the sections, vehicles are moved according to the algorithms of microscopic simulation. The settings of mesoscopic simulation do not apply for these sections. For vehicles traversing between the two areas, the mesoscopic simulation settings continue to apply. Accordingly, properties such as the desired speed are kept.

Please note the information on how to define sections (see "Modeling sections" on page 677). As the calculation basis for vehicle movement changes significantly at these transition areas, unrealistic vehicle behavior may occur, particularly if the edges are very short.

The following figure shows a schematic drawing of the transition area from micro- to mesoscopic simulation.



At the transition point, additional meso nodes and meso edges are created. For vehicles entering the meso area, the macroscopic speed specified for the first edge after transition is used. If this edge is very short, significant fluctuations occur that in turn impact vehicle inputs from the microscopic area. As a consequence, there might be unrealistically high wait times on the micro-meso transition edge, as vehicles are not able to leave it.

At the transition point from meso- to microscopic simulation, vehicles are introduced similarly to vehicle inputs. Here, too, you need to ensure that signal heads, desired speed decisions, reduced speed areas, etc. are not too close to the transition point, otherwise they might be ignored by the vehicles.

For a comprehensive analysis of the entire network, use the network performance and link evaluation (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085), (see "Showing data from links in lists" on page 1103). You can record data using node evaluation (see "Evaluating nodes" on page 1057).

8.14 Selecting sections for hybrid simulation

If during hybrid simulation of your Vissim network, you want to microscopically simulate parts of the network, you must define sections for these parts (see "Modeling sections" on page 677). The desired sections must then be selected for mesoscopic simulation.

1. From the **Simulation** menu, choose > **Parameters**.

   *The **Simulation parameters** window opens.*

2. Select the **Meso** tab.
3. Select the **Meso** simulation method.

   *In the **Sections for microscopic simulation** area, all **sections** are displayed. Vissim only performs microscopic simulation for the sections selected.*

4. Hold down the CTRL key and click the desired sections.

*When you start the simulation, the desired sections of the Vissim network are microscopically simulated. Outside of these sections, the Vissim network is mesoscopically simulated, but vehicles are not shown. For the selected sections, Vissim chooses the attribute **Meso - use as section for microsimulation** (**MesoUseForMicrosim**).*

8.15 Functional differences to microscopic simulation

The following functions are available in microscopic simulation. These functions cannot be used in mesoscopic simulation:

- Vehicle inputs and static routing
- Pedestrians
- Cyclists
- In dynamic assignment:
  - Parking lots of the type **Real parking spaces** as origin and destination of demand
  - Route guidance
  - Path selection type **Decide repeatedly**
- For signal control and intersection control:
  - Priority rules
  - Stop signs
  - Vehicle-actuated controls
- For public transport:
  - Partial PT routes
  - PT telegrams
  - PT vehicles only drive up to the last PT stop of your PT line and not to the end of the PT line, if the latter continues on a link.
  - Block control
- For evaluations:
  - Data collection points
  - Travel time measurements can be performed in sections selected for microscopic simulation (see "Selecting sections for hybrid simulation" on page 838)
  - Queue counters
  - Node evaluation

Should your use cases require any of the functions listed, simulate the relevant parts of the network microscopically (see "Hybrid simulation" on page 837).

9 Running a simulation

You must set simulation parameters before you can start the simulation or a test run (see "Defining simulation parameters" on page 840), (see "Selecting the number of simulation runs and starting simulation" on page 845). You can also start a simulation without a Network editor open.

The parameters for the vehicle simulation also apply to the pedestrian simulation. However, there are additional parameters available for pedestrian simulation (see "Pedestrian simulation" on page 860).

Via the COM Interface, you can also access network object attributes during the simulation (see "Using the COM Interface" on page 1189).

9.1 Selecting simulation method micro or meso

You need to choose between microscopic and mesoscopic simulation of your Vissim network.

1. From the **Simulation** menu, choose > **Parameters**.

   *The **Simulation Parameters** window opens.*

2. Select the **Meso** tab.

3. Select the desired simulation method:

   - **Micro**: When you start a simulation, the Vissim network is simulated microscopically. The **Sections for microscopic simulation** are not relevant for the **Micro** simulation method. They are only relevant for the **Meso** simulation method. Sections under **Sections for microscopic simulation** are displayed as deactivated.
   - **Meso**: When you start a simulation, the Vissim network is simulated mesoscopically (see "Using add-on module for mesoscopic simulation" on page 801). In this case, Vissim can simulate parts of your network microscopically, in a so-called hybrid simulation. These parts must lie within sections. You must select the sections you want Vissim to simulate microscopically (see "Selecting sections for hybrid simulation" on page 838).

9.2 Defining simulation parameters

Before you start your simulation, you can set general simulation parameters.

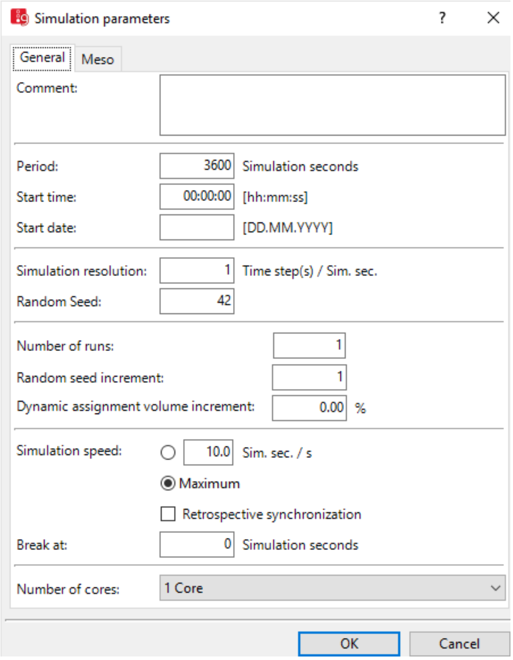
In doing so, you can enter the number of simulation runs you want to do. The following are typical use cases of simulation runs:

- Variation of the random seed for stochastic saving of results
- Performing iterations of a dynamic assignment
- Successive increase of total demand for dynamic assignment

The difference to a simple simulation is that depending on the use case, the corresponding parameter, e.g, the random seed, is automatically changed with each simulation run.

1. Make sure that the desired simulation method has been selected (see " Selecting simulation method micro or meso" on page 840).
2. From the **Simulation** menu, choose > **Parameters**.

   *The **Simulation parameters** window opens.*
3. Select the **General** tab.



4. Make the desired changes:

Element	Description
Comment	ID of simulation run. Is added at the beginning of output file. To save the comment to the network file **.inpx*, from the **File** menu, choose > **Save**.
Period	Simulation time in simulation seconds. You also need to account for lead times of signal controls.
Start time	The simulation start time is at simulation second 0 (see "Information in the status bar" on page 147).
Start date	For signal control procedures with a date-dependent logic, the start date is transferred to the controller DLL. Format: DD.MM.YYYY

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td colspan="2">Simulation resolution

Note:
The simulation resolution has an impact on the behavior of vehicles, pedestrians, and the way they interact. This is why simulations, using different simulation resolutions, produce different results.</td></tr>
<tr><td></td><td>Number of time steps per simulation second: specifies how often vehicles and pedestrian are move in a simulation second.

The position of vehicles is recalculated in a simulation second with each time step. The simulation resolution specifies the number of time steps.
The position of pedestrians is calculated 20 times per simulation second. This is also the case when the simulation resolution specifies less time steps. The simulation resolution then defines the following functions for pedestrians:

how often pedestrian movement is updated per simulation second
how often pedestrians can be reintroduced into the simulation per simulation second
how often pedestrians can make routing decisions per simulation second
how often evaluations may be performed per simulation second

Value range: integers from 1 to 20

Values < 5 lead to jerky movements. This is why this value range is less suitable for production of the final simulation results. As lower values accelerate the simulation, the use of lower values during setup of the network model can be helpful.
Values between 5 and 10 lead to a more realistic demonstration. This value range is suitable for the production of the final simulation results.
Values between 10 and 20 lead to smoother movements. This value range is suitable for high-quality simulation animations.
</td></tr>
<tr><td>Random Seed</td><td>This value initializes a random number generator. Two simulation runs using the same network file and random start number look the same. If you vary the random seed, the stochastic functions in Vissim are assigned a different value sequence and the traffic flow changes. This, e.g., allows you to simulate stochastic variations of vehicle arrivals in the network. This can lead to different simulation results. A comparison of these simulation results allows you to compare the effect of stochastic variations. For this purpose, Vissim calculates additional, meaningful values for various result attributes during its evaluations, e.g. minimum value, maximum value and mean.</td></tr>
</table>

Element	Description
	Notes: - For the model to converge during dynamic assignment, use the same random seed. Using different random seeds for dynamic assignment can cause a so-called seesaw effect. - Once the model has converged and you want to obtain different evaluation results, use different random seeds.
Number of runs	Number of simulation runs performed in a row. Logical value range: depends on use case 5 - 20. For dynamic assignment, more than 20 simulation runs may be necessary. Note: Before starting multiple simulation runs for dynamic assignment, select the attributes of your choice (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
Random seed increment	Difference between random seeds when you perform multiple simulation runs. This number is irrelevant for stochastic distribution. For the model to converge during dynamic assignment, enter 0. When you perform multiple simulation runs using different random seeds, the number of the respective simulation run is added to the name of the evaluation file **.ldp*.
Dynamic assignment volume incre- ment	For dynamic assignment only: Increases total demand of the origin-destination matrix with each simulation run defined (in the **Number of runs** box) by the value specified. The start value used is the parameter **Scale total volume to** of dynamic assignment (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771). For each iteration, the demand is automatically increased by this value until 100% of the total demand is reached. Once 100 % of the total demand is reached, any additional simulation runs (as specified in the **Number of runs** box) will be performed using 100 % of the total demand. Avoid using a start value larger than 100% of the total demand. If due to the **Number of runs** specified you do not perform enough simulation runs to reach 100 % of the total demand and you then save the Vissim network, the current value of the total demand is saved to the parameter **Scale total volume to** of dynamic assignment. The next time you open Vissim, this value will be used as the new start value.
Simulation speed	Corresponds to a time lapse factor: Indicates simulation seconds per real-time second Value 1.0: The simulation is run in real-time. Value 2.0: The simulation is run at double real-time speed.

Element	Description
	Maximum option: Select this option to run the simulation at the maximum speed. The simulation speed does not affect the simulation results. You can change the simulation speed during the simulation run. The desired simulation speed might not be reached, if you are using a large networks or a slower computer.
Retrospective synchronization	☑ Select this option if at the end of a time interval, you do not want Vissim to wait until the real-time set for this interval has elapsed. Instead, Vissim will wait until the real-time for all time intervals, since continuous simulation was last started, has elapsed. This allows VISSIM to make up for the time lost through slower, individual time intervals, e.g. due to external signal control or other external factors. With the help of external controller hardware, the synchronization function ensures that the time lost is regained. If you open a window during a continuous simulation run in Vissim, this can cause a delay. After you close the window, the delay is made up for through maximum simulation speed, until the simulation second divided by the simulation speed equals the total real-time elapsed. When the simulation speed of a continuous simulation run is reduced through retrospective synchronization, Vissim waits until the total real-time (since simulation start) equals the current simulation second divided by the simulation speed, before it executes the next time interval.
Break at	Simulation second after which the program automatically switches to **Simulation single step** mode. You can use this option to view the traffic conditions at a specific simulation time.
Number of cores	Number of processor cores used during simulation. The maximum number of cores used depends on your computer. Your setting remains selected when you start the next simulation run. Default: **Use all cores** The number of cores selected is saved to the network file **.inpx*. You cannot change this setting during the simulation run. Note: If at least one dynamic potential is used to simulate pedestrians, all available processor cores are used and not the number stated in this field.

5. Confirm with **OK**.

*The **Simulation Parameters** window closes Your settings are saved to the network file. The simulation runs are started with your current settings. The status bar shows the number of the current simulation run and the total number of simulation runs performed in brackets. Evaluation files are saved to the folder selected.*

The simulation parameters specified are taken into account for the next simulation or test run.

9.2.1 Special effect of simulation resolution on pedestrian simulation

From a mathematical point of view, this type of simulation basically represents a numerical integration of a system of coupled differential equations. Smaller time steps allow for solutions approaching the exact solution. A larger number of time steps, however, requires more computation. Vissim calculates pedestrians with a time step of 0.05 seconds, i.e. 20 times per second. This value only slightly diminishes the accuracy compared to the exact solution. By comparison, in many projects the impact of uncertain external factors is probably greater, e.g. the real number of pedestrians and their speed.

Vissim's program module for pedestrian movement internally communicates with other modules. These modules for instance generate pedestrians or remove them during simulation, calculate route choice or control the output of evaluations. The modules communicate at a **simulation resolution** specified in the simulation parameters. These other modules have less impact on pedestrian movement than the program module responsible, but still display some effect. For example, with a low simulation resolution, a pedestrian could skip a very narrow area on which a pedestrian routing decision lies. The routing decision would then have no effect on the pedestrian. The simulation resolution also has an impact on pedestrians that enter escalators: with each time step of simulation resolution, only one pedestrian may enter the escalator.

Suitable simulation resolution

- When still creating your model and not performing any evaluations yet, you can use a simulation resolution of 1 or 2 steps per second.
- Depending on your use case, simulation resolutions of 1 or 2 steps per second may change the simulation runs performed for testing purposes during the setup phase in a non-desired manner. You should then increase the simulation resolution. If you want to evaluate simulations at the end of a setup phase, use simulation resolutions of at least 5, better 10 or 20, steps per second.
- For AVI recordings that you can use for presentations of your simulation to external audiences, use a simulation resolution of 20 steps per second.

9.3 Selecting the number of simulation runs and starting simulation

Tip: To gain an impression of the stochastic distribution of results, run multiple simulations using different random seeds and compare the results (see "Defining simulation parameters" on page 840).

Notes:

- When using the 64-bit Vissim edition up until Vissim 10 with large networks and numerous vehicles, you might obtain different simulation results than with the 32-bit program version due to its different rounding behavior. From Vissim 11, Vissim is available as a 64-bit edition only.
- Opened lists might reduce the simulation speed. When you close opened lists, this may increase the simulation speed.

Set the simulation parameters before you start a simulation run (see "Defining simulation parameters" on page 840). You can then run a simulation in the **Simulation single step** or **Simulation continuous** mode.

1. From the **Simulation** menu, choose > **Continuous** or **Single Step**.

   Vissim initializes the simulation. Initialization might take a while, if you are using a large network. In this case, a window opens displaying the progress of initialization.

2. When this window is displayed, but you wish to cancel initialization, click the **Cancel** button.
3. If messages are shown during the simulation, follow the instructions and make the settings required.

You can use the **Simulation** toolbar to control simulation runs:

Symbol	Name	Description	Key
	Simulation continuous	Starts continuous simulation run or switches from **Simulation single step** mode to **Simulation continuous** mode.	F5
	Simulation single step	Starts simulation in **Simulation single step** mode or switches from **Simulation continuous** mode to **Simulation single step** mode or executes the next single step.	F6
	Stop sim-ulation	Stop started simulation run	Esc

Convergence may already be reached before the number of simulation runs is completed that has been defined in the simulation parameters, in the **Number of runs** box. In this case, you can select the **Behavior upon convergence:** (see "Attributes for achieving convergence" on page 782).

9.4 Showing simulation run data in lists

You can show data of the simulation runs in a results list.

- From the **Lists** menu, choose **Results** > **Simulation Runs**.

  *The **Simulation Runs** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Element	Description
No.	Consecutive number of simulation run
Timestamp	Date and time the simulation starts
Comment	Comment on simulation
SimPeriod	Simulation time in seconds specified in the simulation parameters
SimEnd	Time in seconds after the start at which the simulation was ended

Element	Description
Start Date	Start date specified in the simulation parameters
StartTime	Start time specified in the simulation parameters
RandSeed	Random seed specified in the simulation parameters
VissimVers	Program version installed

Tip: In the **Simulation runs** list, you can delete the simulation runs that are no longer required. This will accelerate loading the network file **.inpx*.

9.5 Displaying vehicles in the network in a list

During a simulation run, you can show vehicle data for each vehicle in the network in the **Vehicles In Network** list.

If vehicles are controlled via the COM interface or the driving simulator interface, the following applies:

- Vehicle attribute values may be based on external data, for example on values of the attributes **Headway**, **Leading target type**, **Leading target number**
- Vissim's car following vehicle model is not used. As the attributes **Following distance**, **Speed difference**, **Safety distance**, **Interaction state**, **Interaction target type** and **Interaction target number** are based on Vissim's car following model , their values are 0.

1. Start the simulation (see "Running a simulation" on page 840).
2. From the **Lists** menu, choose > **Results** > **Vehicles in Network**.

   *The **Vehicles In Network** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Element	Description
No	Number of the vehicle
VehType	**Vehicle type**: Vehicle type assigned to the vehicle
Lane	**Lane**: Number of lane on which the vehicle is located
Pos	**Position**: Distance covered on the link from the beginning of the link
Speed	**Speed**: Speed of the vehicle at the end of the time step
DesSpeed	**Desired Speed**: Desired speed distribution of the vehicle
Acceleration	**Acceleration** during the time step. - Positive value: Current speed of vehicle - Negative value: Current reduction of vehicle speed

Element	Description
LnChg	**Lane Change**: Direction in which the vehicle changes the lane: - **Links** - **Right** - **None**
DestLane	**Destination lane**: Number of lane to which vehicle changes
PTLine	**Public transport line**: Number of PT line of PT vehicle
PTDwellTmCur	**Public transport - dwell time (current)**: Total dwell time at the current PT stop, including slack time fraction

The vehicle has additional attributes that you can view in the **Vehicles In network** list and save as a vehicle log to a file or database (see "Saving vehicle record to a file or database" on page 1031). Among them are the following for example:

Attribute	Description
2D/3D model	3D model file of the vehicle
Current 3D state	Motion states of a pedestrian or a vehicle of the vehicle category **Pedestrian**, which can be stored in the 3D model file. If the model file contains three or 21 motion states, the **Current 3D state** is automatically changed during simulation depending on the speed.
Indicating	Current state of vehicle turn signal. Corresponds to current visualization during simulation: - **No**: Vehicle is not indicating - **Links** - **Right**
Following distance	Distance from the front edge to the front edge of the interaction vehicle in [m] before the time step
Number of stops	Number of stops (cumulated): All situations in which a vehicle comes to a standstill (speed = 0), except stops at PT stops and in parking lots.
Dwell time	Dwell time [s] at a stop sign or at a PT stop. For PT stops: Actual dwell time according to PT line-specific attributes (see "Calculating dwell time according to the advanced passenger model" on page 533).
Occupancy	Defines the number of persons or passengers in a vehicle. This value is output before and after boarding and alighting.

Attribute	Description
Emissions	For add-on module **API package** only: Results of emission calculation for selected file *EmissionModel.dll* (see "Activating emission calculation and emission model for a vehicle type" on page 274). Emission values are also displayed in: - **Vehicle Network Performance Evaluation** (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085) - Evaluation **Links per segment** (see "Showing data from links in lists" on page 1103) - **Vehicles in network** list (see "Displaying vehicles in the network in a list" on page 847)
Motion state - **Default**: Default state, if none of the following states are true. - **In queue**: The vehicle is stuck in a traffic jam. At least one queue counter must be defined in the network. - **Waiting for lane change**: Vehicle has been waiting more than 6s at the last position for lane change (emergency stop distance). - **Ignores priority rule**: Vehicle ignores priority rule to resolve a deadlock situation. - **In priority rule deadlock situation**: Vehicle is part of a deadlock situation caused by priority rules - **Wants to change lanes**: Due to the vehicle route, a lane change becomes necessary that the vehicle has not yet begun. - **Is changing lanes**: Vehicle is changing lanes. - **After lane change**: Lane change was performed in the last 6 s. - **Brakes to change lanes**: Vehicle brakes, as it need to change lanes. - **Brakes cooperatively**: Cooperative braking for an upcoming lane change maneuver of a vehicle, from an adjacent lane to its own lane. - **Sleep**: Vehicle is currently not paying attention. - **Ignores signal**: Vehicle has decided in the last 3 seconds of simulation to traverse a red signal head or a blocked section of a priority rule because its speed was too high to come to a stop in advance. - **Brakes heavily**: Vehicle brakes heavily (< -3.0 m/s²). - **Brakes moderately**: Vehicle brakes moderately (-3.0 m/s² to -1.0 m/s²).	
Next trip departure time	Departure time from parking lot (simulation second)
Next trip activity	Number of the activity
Next trip activity minimum duration	Minimum duration of activity
Origin parking lot	Number of the origin parking lot
Origin zone	Number of zone
Destination zone	Number of destination parking zone
Number	Number of the vehicle

Attribute	Description
Vehicle type	Number of vehicle type
Name	Name of vehicle type
Headway	Distance from the front edge to the front edge of the preceding vehicle before the time step
Distance traveled (total)	Total distance traveled so far
Time in network (total)	The total time which a vehicle is in the network. The value is saved only in the last second before the vehicle leaves the network. Also record the **Delay time** for the vehicle in this time step. For vehicles that are still in the network at the end of the simulation, you can determine the total time on the basis of their utilization time. The vehicle record contains the total time = 0.00, as long as the vehicle is still in the network.
Speed difference	Relative to the preceding vehicle in the time step (>0 = faster)
Interaction state	Short identifier for the state in the interaction procedure via which the acceleration or deceleration of the vehicle in the previous time step was determined (see "Driving states in the traffic flow model according to Wiedemann" on page 285). List of possible interaction states see below (see "Value of the Interaction state attribute" on page 852)
Cost (total)	Costs accrued so far
Power	Power [kW]
Length	Length
Weight	Weight [t]
Position (lateral)	Lateral position at the end of the time step. Value range 0 - 1: - 0: at the right lane edge - 0.5: middle of the lane - 1: at the left lane edge
Route number	Number of route
Routing decision no.	Number of routing decision
Start time	Network entry time [simulation second ss,f], where f (fraction) is a two-digit number
Start time	Start time as time of day [hh:mm:ss,f], where f (fraction) is a two-digit number
Simulation time (time of day)	Simulation time as time of day [hh:mm:ss,f], where f (fraction) is a two-digit number
Simulation second	Simulation time in seconds [ss,f], where f (fraction) is a two-digit number

Attribute	Description
In queue	- **+** = Vehicle in queue - **-** = Vehicle not in queue
Speed (theoretical)	Theoretical speed without hindrance
Delay time	Difference between optimal (ideal, theoretical) driving time
Leading target number	Number of the relevant preceding vehicle
Coordinates rear	The coordinates (x), (y) or (z) of the rear edge of the vehicle at the end of the time step
Coordinates front	The coordinates (x), (y) or (z) of the front edge of the vehicle at the end of the time step
Coordinate rear (x), (y), (z)	The coordinates (x), (y) or (z) of the rear edge of the vehicle at the end of the time step
Coordinate front (x), (y), (z)	The coordinates (x), (y), (z) of the front edge of the vehicle at the end of the time step
Desired speed	Desired speed
Safety distance	Safety distance during the time step
Destination parking lot	Number of the destination parking lot
Public transport - course number	Number of course
Public transport - dwell time (total)	Total of all stop dwell times
The following applies for the following PT attributes: - Before and after boarding and alighting, the values are zero. Values are only displayed, when the PT vehicle is stationary at the PT stop. - During boarding and alighting, the values are current for each time step, with the exception of the attributes **Public transport - waiting passengers** and **Public transport - waiting time (average)**. - For real passengers, the attribute values **Public transport - waiting passengers** and **Public transport - waiting time (average)** are evaluated when the vehicle stops at the waiting area. These values do not change while the PT vehicle is stationary.	
Public transport - alighting passengers	Number of alighting passengers at the current PT stop
Public transport - boarding passengers	Number of boarding passengers at the current PT stop

Attribute	Description
Public transport - passenger service time	Total passenger service time [s] at the current stop. - The **Public transport - passenger service time** is counted up during the dwell time. - Per default, the **Public transport - passenger service time** is one time step at the beginning of the dwell time. This ensures that at the end of boarding and alighting, the Public transport - passenger service time includes the entire **Public transport - dwell time (current)**.
Public transport stop	Number of current stop
Public transport - dwell time (current)	Expected remaining dwell time at current PT stop. Slack time fraction is taken into account.
Public transport - waiting time (average)	Average waiting time of boarding passengers at the current PT stop. For real pedestrians the waiting time is evaluated with the time step of arrival.
Public transport - lateness	Lateness in departing from the current PT stop (>0 = late)
Public transport - waiting passengers	Number of waiting passengers at the current PT stop. - For real pedestrians: The number is only evaluated with the time step of arrival and displayed during time steps in which the PT vehicle is stationary. - For calculated boarding and alighting passengers: The number calculated is displayed with the time step of PT vehicle arrival and is displayed in the time steps during which the PT vehicle is stationary.

Value of the Interaction state attribute

Status	Description
Free	Vehicle is not affected by any relevant preceding vehicle. It tries to drive at desired speed, free driving (see "Driving states in the traffic flow model according to Wiedemann" on page 285).
Follow	Vehicle tries to follow a leading vehicle at its speed (see "Driving states in the traffic flow model according to Wiedemann" on page 285).
Brake BX	Braking at the desired safety distance (before reaching the safety distance), approaching (see "Driving states in the traffic flow model according to Wiedemann" on page 285).
Brake AX	Braking at the desired safety distance (after reaching the safety distance) (see "Driving states in the traffic flow model according to Wiedemann" on page 285).

Status	Description
Close up	The vehicle slowly closes in the following cases: - There is a stationary vehicle in front of it - while it is pulling out of a parking space in reverse onto its original link and upstream there is a stationary vehicle or a vehicle approaching - until it reaches an obstacle, for example, a signal head, a stop sign, priority rule, conflict area.
Brake ZX	Target deceleration to an emergency stop distance for a lane change or a reduced speed area.
Brake LCH	Slight deceleration for a lane change in order to wait for the next upstream gap in the adjacent lane.
Brake cooperative	Cooperative braking to allow another vehicle to change lanes (setting via parameter **Maximum deceleration for cooperative braking**) (see "Editing the driving behavior parameter Lane change behavior" on page 300).
External	Acceleration/deceleration is controlled by an external driver model DLL.
Loss of attention	The parameter **Temporary lack of attention** is currently active, there is neither acceleration nor braking except for an emergency braking.
Pass	Acceleration/deceleration to reach a permitted speed depending on the lateral distance for passing another vehicle in the same lane or an adjacent lane.
Stop	The vehicle stops.

9.6 Showing pedestrians in the network in a list

During a simulation run, you can show pedestrian data for each pedestrian in the network in the **Pedestrians In Network** list.

1. Start the simulation (see "Running a simulation" on page 840).
2. From the **Lists** menu, choose > **Results** > **Pedestrians in Network**.

   *The **Pedestrians In Network** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list contains, amongst others, the following attributes:

Element	Description
No	Pedestrian number
PedType	**Pedestrian type**: Pedestrian type assigned to pedestrian
Length	Pedestrian length in 2D/3D model distribution
Width	Pedestrian width in 2D/3D model distribution
Height	Pedestrian size in 2D/3D model distribution
Level	Level on which the pedestrian moves

Element	Description
ConstrElNo	**Construction element number** of area, ramp or stairway on which the pedestrian moves
ConstrElType	Area, ramp or stairway on which the pedestrian moves
CoordCenter	**Center coordinates**: Coordinates of the center between the front and back of the pedestrian.
DesSpeed	**Desired speed** of pedestrian
Speed	Pedestrian's current velocity
StaRoutDecNo	Static routing decision number
StaRoutNo	Static route number
PTState	**Public transport state**: Is the pedestrian waiting, is he alighting from the PT vehicle or walking towards it: - None - Waiting - Approaching - Alighting
MotionState	**Motion state**: The current motion state of a pedestrian shows for how long the pedestrian does a certain activity:
	Approaching PT vehicle: The pedestrian is on the way from the waiting area to the PT vehicle.
	Alighting from PT vehicle: The pedestrian leaves the PT vehicle.
	Waiting for PT vehicle: The pedestrian waits for PT vehicle on a waiting area.
	Walking upwards on escalator: The pedestrian is walking upwards on an escalator.
	Walking downwards on escalator: The pedestrian is walking downwards on an escalator.
	Standing on escalator: The pedestrian is on an escalator and is not walking.
	Walking on moving walkway: The pedestrian is walking on a moving walkway.
	Standing on moving walkway: The pedestrian is standing on a moving walkway.
	Waiting at queue head: The pedestrian is first in a queue. The pedestrian is waiting in accordance with the dwell time distribution specified.
	Waiting in queue: The pedestrian is waiting in a queue and is not first in line.
	Walking upstairs: The pedestrian is on a stairway or ramp and is walking upwards. The pedestrian does not necessarily have to access another level.

Element	Description
	Walking downstairs: The pedestrian is on a stairway or ramp and is walking downwards.
	Approaching elevator: The pedestrian is walking from the waiting area to the elevator.
	Alighting from elevator: The pedestrian is exiting the elevator.
	Waiting for elevator: The pedestrian is waiting for an elevator.
	Riding elevator: The pedestrian is in an elevator and not exiting it.
	Waiting: The pedestrian is waiting on an area for which the attribute **Queuing** has not been selected.
	Walking on level: The pedestrian is walking on a level, e.g. on flat stairs or a ramp. None of these other motion states apply
IsInQueue	**Is in queue**: The pedestrian waits in the queue and can move up in it. The option is not selected: - if the pedestrian has not yet reached the end of the queue - if the next route location of the pedestrian is not a queue

The network object may have additional attributes, for example, the following. These can also be shown in the Attributes list:

Element	Description
ExperDens	**Experienced density** within the perception radius of a pedestrian: Density of pedestrians in the same area, measurement area or on the same ramp of a pedestrian. Based on the pedestrians that at the end of an evaluation interval are located within a radius around a pedestrian. In the network settings, you can specify the radius (see "Selecting network settings for pedestrian behavior" on page 204).
ExperVelVar	**Experienced velocity variance**: Vectorial speed differences of all pedestrians within the environment radiuses of their individual speed (see "Selecting network settings for pedestrian behavior" on page 204)
Coordinates rear	The coordinates (x), (y), (z) of the rear edge of the pedestrian at the end of the time step
Coordinates front	The coordinates (x), (y), (z) of the front edge of the pedestrian at the end of the time step
Coordinate rear (x), (y), (z)	The coordinates (x), (y) or (z) of the rear edge of the pedestrian at the end of the time step
Coordinate front (x), (y), (z)	The coordinates (x), (y) or (z) of the front edge of the pedestrian at the end of the time step
PosInQueue	**Position in queue**: Number of the position of the pedestrian in the queue. Front pedestrian =1, second pedestrian =2, etc.

9.7 Reading one or multiple simulation runs additionally

You can select a **.db* or **.sdf* file to which the result attributes of a simulation run are saved, or choose a directory to which multiple **.db* or **.sdf* files are saved and then import these files. This allows you to compare simulation runs that were originally generated on different computers or saved to different evaluation output directories. The files are copied to the evaluation output directory *.results* of the network currently loaded. If the evaluation output directory *.results* does not exist yet, Vissim will create the directory before it reads the file/s additionally.

9.7.1 Reading a simulation run additionally

You can select a **.db* ir **.sdf* file, to which result attributes of a simulation run are saved, and copy it to the evaluation output directory of the currently loaded network.

- File **.sdf* up to Vissim8: SQL Server Compact Edition
- File **.db* from Vissim9: SQLite database

1. From the **File** menu, choose > **Read Additionally** > **Simulation run (.sdf file, .db file)**.

   The ***Import Simulation Run*** *window opens. The file formats Simulation run results *.sdf; *.db are selected by default.*

2. Select the path to the directory in which the desired **.db* or **.sdf* file has been saved.
3. Select the desired file.
4. Click the **Open** button.

*The simulation run is saved to the directory ..\<Name of network file>.results of the currently loaded network. The next higher number available is assigned to the file name of the simulation run *.db or *.sdf.*

You can show simulation runs in the ***Simulation Runs*** *list* *(see "Showing simulation run data in lists" on page 846).*

9.7.2 Reading simulation runs additionally

You can select a directory to which multiple simulation runs have been saved in **.db* or **.sdf* files and then copy these files to the evaluation output directory of the currently loaded network.

1. From the **File** menu, choose > **Read Additionally** > **Simulation runs (entire folder)**.

   The ***Find folder*** *window opens.*

2. Select the path and desired directory to which the **.db* or **.sdf* files have been saved.
3. Confirm with **OK**.

*The simulation runs are saved to the directory ..\<Name of network file>.results of the currently loaded network. The next higher number available is assigned to the file name of the simulation runs *.db or *.sdf.*

*You can show simulation runs in the **Simulation Runs** list (see "Showing simulation run data in lists" on page 846).*

9.8 Checking the network

Errors in the network impact calculations and evaluations. Use the **Check network** command to check the Vissim network for consistency after editing it. The command **Check network** can identify errors in the Vissim network and allows you to fix some of the errors immediately. **Check network** finds inconsistencies, e.g. incorrectly entered attribute values. However, it cannot identify missing attribute values.

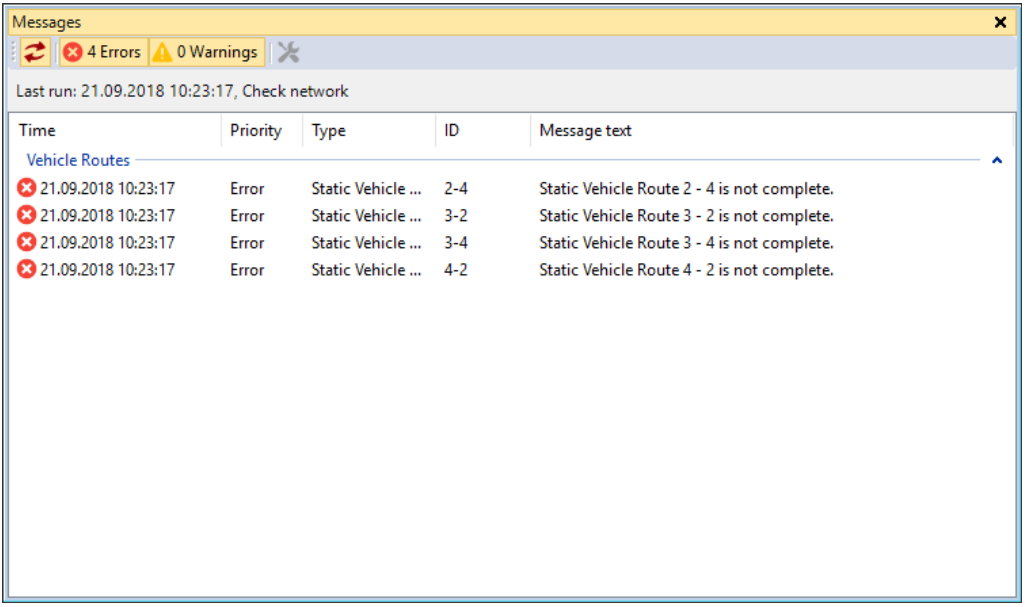
Check network also finds network objects with attribute values that Vissim cannot unambiguously assign to a vehicle type: A vehicle type can be assigned to more than one vehicle class. Several of these vehicle classes can be assigned to a network object, e.g. in the case of **Conflict areas**, for **Gaps By Vehicle Class**. If for different vehicle classes, different attribute values are selected, unambiguous assignment to a vehicle type that has been assigned multiple vehicle classes is no longer possible. This also applies for pedestrian types that are assigned to multiple pedestrian classes.

- When you start the simulation, Vissim automatically checks the Vissim for certain constraints that could prevent the simulation start and performs the **Check network** function, if it has been selected under User Preferences (see "Checking and selecting the network with simulation start" on page 155).
- When you call the **Check network** command without starting simulation, Vissim checks the Vissim network for consistency, but not for any constraints.

1. From the **Simulation** menu, select > **Check network**.

   *If the **Messages** window is open and contains entries, these are deleted. Vissim checks the network.*

   - If Vissim cannot identify any errors, a window is opened and a corresponding message is displayed.
   - Should **Check network** detect that the simulation cannot be started, the **Messages** window opens, showing the errors.
   - Should **Check network** detect that the simulation can be started, but that there are inconsistencies, the **Messages** window opens, showing warnings(see "Showing messages and warnings" on page 1178).



In the **Messages** window, Vissim groups errors by categories. These groups, for example, inlcude:

- Driving simulator
- Scripts
- Vehicle simulation
- Vehicle classes
- Pedestrian simulation
- Pedestrian classes
- PT pedestrian simulation
- Evaluations
- Presentation
- External driver model
- Dynamic assignment
- Vehicle Routes
- Mesoscopic simulation
- Scenario Management

The entries of a group can be edited or deleted together (see "Showing messages and warnings" on page 1178).

*Errors listed in the groups **Vehicle routes** and **2D/3D model segments** can be repaired interactively, after you have corrected their network objects. During an interactive repair, Vissim for instance reconnects interrupted vehicle routes, after you have corrected the corresponding links and connectors (see "Showing messages and warnings" on page 1178). Errors of other categories must be corrected manually.*

When for a 3D info sign, the reference object type and/or the reference object is deleted, ***Check network*** *displays a message informing you of the respective 3D info sign. You can then select a reference object type and/or reference object of your choice or delete the 3D information sign.*

You can correct individual entries or an entire group (see "Showing messages and warnings" on page 1178).

2. Use the messages displayed to correct your Vissim network.
3. If you correct errors in the groups **Vehicle routes** and **2D/3D model segments**, repair these objects afterwards using the Fix command (see "Showing messages and warnings" on page 1178).

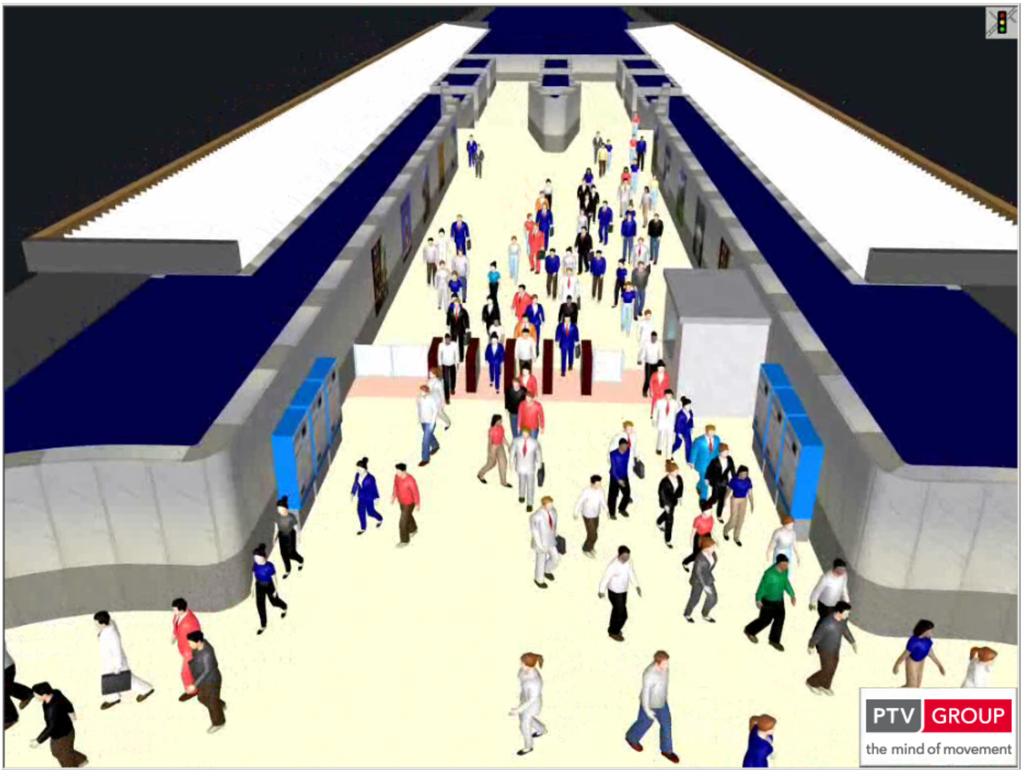
10 Pedestrian simulation

In addition to the simulation of vehicles by default, you can also use Vissim to perform simulations of pedestrians based on the Wiedemann model (see "Version-specific functions of pedestrian simulation" on page 861).

However, with the stand-alone product PTV Viswalk, you can simulate pedestrians based on Helbing but no vehicles. You can simulate vehicles and pedestrians only with Vissim and the add-on module PTV Viswalk. You can then choose whether to use the modeling approach of Helbing or Wiedemann.

10.1 Movement of pedestrians in the social force model

The movement of pedestrians in Vissim respectively Viswalk is based on the Social Force Model (Helbing and Molnár, 1995). The basic principle of the Social Force Model is to model the elementary impetus for motion of the pedestrians analogous to Newtonian mechanics. From the social, psychological, and physical forces results a total force, which eventually results in an entirely physical parameter Acceleration. These forces arise from the desire of the pedestrian to reach a goal, from the influence of other pedestrians and obstacles in his environment.



Prof. Dr. Dirk Helbing is a scientific advisor for PTV GROUP. Together with PTV GROUP he specially expanded the Social Force Model for the use in Viswalk. This simulation model was validated in three different ways:

Firstly, macroscopic parameters were calculated and adjusted to empirical data. Secondly it was assured that microscopic effects like lane formation (counterflow) and stripe formation (crossing) are reproduced. The resulting animation should be represented as realistically as possible.

The behavior of pedestrians can be divided into three hierarchical levels (Hoogendoorn et al. 2002):

- On the strategic level of minutes to hours, a pedestrian plans his route, generating a list of destinations.
- On the tactical level of seconds to minutes, a pedestrian chooses the route between the destinations. Thereby he takes the network into account.
- On the operational level of milliseconds to seconds, the pedestrian performs the actual movement. He avoids thereby oncoming pedestrians, navigates through a dense crowd, or simply continues the movement toward his destination.

The Social Force Model controls the operational level and parts of the tactical level. You define the settings of the strategic level.

10.2 Version-specific functions of pedestrian simulation

The Vissim license for vehicular traffic may optionally include the Viswalk add-on module for pedestrian simulation.

- If your Vissim license does not include the Viswalk add-on module, you still have access to the following functions:
  - You can import files containing elements for pedestrian modeling in **.inpx* format. They may also contain several levels and more than 30 pedestrians.
  - You can perform a simulation that also includes pedestrians. You can model up to 30 pedestrians in the network at the same time. If the imported network file does not contain pedestrian data, Vissim generates default data for pedestrian types, pedestrian classes and pedestrian compositions.
  - Pedestrian modeling can be performed using the Wiedemann or Helbing modeling approach. This is possible for up to 30 pedestrians in the network.
- If your Vissim license includes the Viswalk add-on module, the number of pedestrians is limited by the license size of Viswalk.
- You can choose whether to use the modeling approach of Helbing or Wiedemann with the Viswalk add-on module.
- When you start the Viswalk program, the following default data is generated:
  - Pedestrian types, for example **Man**,**Woman**, **Woman & Child**, **Wheelchair**
  - Pedestrian types, for example **Man, Woman**, **Wheelchair user**
  - Pedestrian compositions, e.g. **Pedestrians**

- The Viswalk product without vehicle simulation can only be used to simulate pedestrians.
- During the installation of Viswalk, a specific default *defaults.inpx* network file for the pedestrian simulation is installed which does not contain any vehicle-specific objects (see "Saving and importing a layout of the user interface" on page 146).
- If you have a maintenance agreement for Viswalk, you can contact PTV Vision Support (see "Service and support" on page 46).

10.3 Modeling examples and differences of the pedestrian models

The modeling examples show animations of various scenarios which are typical for pedestrian traffic. The various scenarios require different model data according to Wiedemann or Helbing.

10.3.1 Modeling examples: Quickest or shortest path?

On YouTube, you can find a very complex animation of various scenarios for modeling pedestrian traffic: http://www.youtube.com/watch?v=8SmRBTJ-jeU.

This animation demonstrates the principle of how simulated pedestrians in Vissim are made to walk along the path of estimated least remaining travel time in due consideration of other pedestrians and obstacles. The animation demonstrates as well the effect of the **Dynamic Potential** method. It compares pedestrians who select the quickest path with those who choose the shortest one (beginning at 01:42).

Much like vehicle drivers, pedestrians try to minimize their travel times to the destination. This desire can in some situations superimpose over all other aspects. Moreover, the walking direction for the quickest path cannot always be determined without problems.

Details of the method were published in an article in **Advances in Complex Systems**:

http://dx.doi.org/10.1142/S0219525911003281

Available at arXiv:

http://arxiv.org/abs/1107.2004

10.3.1.1 List of scenarios in the demo video

The following list shows at what time in the video which scenario begins.

Notes:

- The various scenarios run with different time lapse factors.
- The efficiency of the dynamic potential is always 100 %.
- In all situations, the pedestrians move with approximately the same speed.

mm:ss	Description
01:42	About 800 passengers alight from two trains arriving simultaneously at the station at the south entrance of Berlin's congress center (ICC). To create a model of a large group of pedestrians walking realistically and efficiently around a corner, mainly the Dynamic Potential method is used. With only a small group of pedestrians the trajectories of both the quickest and the shortest path would be almost identical, because both paths would have approximately the same course.
03:18	Here a large group of pedestrians has to take an almost complete U-turn in the course of their path. This is more difficult and therefore the difference between the two methods (left and right) is even more distinct.
04:48	In this scenario two large pedestrian groups meet as opposing flows. This is a situation where the use of the dynamic potential does not necessarily produce better results. However, it provides an alternative pedestrian behavior that becomes clear after a few seconds. The behavior on the left side is more realistic if the pedestrians assume that the counterflow will persist only for a short time, for example during the green phase at the pedestrian crossing, the behavior on the right side is more realistic, if the pedestrians assume that the constellation will persist longer, for example when visiting a public event.
06:18	If counterflow occurs at a 90° corner, the dynamic potential (right side) is able to better reproduce the fact, that the pedestrians move more efficiently in such situations and most of them are able to resolve the situation. However, with extremely high pedestrian traffic in reality it can also come to such a jam as visualized on the left.
07:48	Counterflow at a 180° turn (U-turn).
09:18	Some passengers (red) are urgently rushing for their train, some (green) have just alighted from a train and are heading towards exit whereas some others (blue) have arrived at the station before departure and now spend their waiting time standing or strolling around. (Easily recognizable, the group is quite large and behaves strangely; thereby the effect of the method can be better demonstrated.) The red and green pedestrians in the upper left video follow the shortest path. However, they are increasingly being blocked by the numerically growing blue group. The upper right video and the two scenarios below were simulated with the quickest path but with different values for parameter h. For details of parameter h please refer to the publication linked above. Note that in the two scenarios below the red and green pedestrian groups manage respectively to establish a separate walking direction or to form lanes spontaneously, whereas they fail to do so with parameter h = 0 in the example at top right.

mm:ss	Description
10:08	This is a theoretical model that does not even remotely occur in reality: However, it demonstrates very clearly and precisely the effect of the "quickest path" approach or alternatively of the dynamic potential.
10:48	So far all routing decisions were continuous. Thus, the pedestrian had always more path options to their destinations to choose from. This is the first example with discrete alternatives. The pedestrians have to choose if they want to use the left or the right corridor. The method of dynamic potential has not been developed for such situations. Other methods might be more helpful. In Vissim for example the partial routes are used. The Dynamic Potential method is however suited also in this case.
12:43	A grandstand: The interesting aspect of this example is that the grandstand for the pedestrians consists of a sequence of one-dimensional objects (links). Therefore the directions of the shortest and the quickest path can differ by 180 degrees. In this video it is very obvious when pedestrians prefer to take a detour to reduce the walk time.

10.3.2 Main differences between the Wiedemann and the Helbing approaches

When pedestrians are modeled as a vehicle type according to Wiedemann model, they do not move around freely but along user-defined links in the network. The spatial characteristics of their trajectories are thus formed by the input data for the model and they do not result from the simulation. Only the time at which a pedestrian crosses a link at a particular point is calculated and a result formed.

In Helbing's model the pedestrians can move freely in two spatial dimensions. Their trajectories are thus not defined in advance, but are calculated by the model. Therefore this approach for pedestrian simulation is more flexible, detailed and realistic.

However, there are situations, in which the essential elements of the dynamics are produced by the Wiedemann model. Examples are projects, where pedestrians have no role other than to cause interruptions to vehicular traffic at signalized junctions.

10.3.2.1 Options for pedestrian modeling in the Wiedemann model

The add-on module Viswalk is not required.

Levels of Interaction for pedestrians	Elements of the Vissim network
Pedestrians using pedestrian crossings in the road network	• Links • Pedestrians as a vehicle type
PT passengers	• Public transport stops • PT lines as vehicle type • Stop dwell time distribution or number of boarding volumes

10.3.2.2 Options for pedestrian modeling in the Helbing model

The add-on module Viswalk is necessary.

Levels of Interaction for pedestrians	Elements of the pedestrian module	Elements of the Vissim network
Only pedestrian flows, for example - Emergency situations - Airport, hotel etc.	- walkable construction elements (ramps/stairs and areas) - multilevel, if applicable - pedestrians as type/class of pedestrians - pedestrian compositions - area behavior types, if applicable - walking behavior parameters, if applicable - location distributions, if applicable	- None
pedestrians using pedestrian crossings in the road network	- pedestrians as type/class of pedestrians - pedestrian compositions - area behavior types, if applicable - walking behavior parameters, if applicable	- links as walkable areas - Signal control - Conflict areas - Detectors
Pedestrians as PT passengers in the network	- walkable areas, including platform edges and waiting areas, if applicable - multilevel, if applicable - pedestrians as type/class of pedestrians - pedestrian compositions - area behavior types, if applicable - walking behavior parameters, if applicable - location distributions, if applicable	- Public transport stops - PT lines as vehicle type with doors

10.4 Internal procedure of pedestrian simulation

For the simulation of pedestrians, several requirements must be met, for example, you need to define different base data. In areas where pedestrians are supposed to start, you insert pedestrian inputs and define routing decisions. Based on the routing decisions, you define routes for pedestrians that lead via areas, ramps and stairways to other routing decisions, where the routes then end. You may add intermediate points to areas, ramps and stairways.

10.4.1 Requirements for pedestrian simulation

- At least one pedestrian type has to be defined.
- At least one pedestrian composition has to be defined.
- At least one pedestrian input, yielding pedestrians, must be defined.

> Tip: As an alternative to pedestrian inputs, you can use a pedestrian OD matrix that contains demand data (see "Pedestrian OD matrices" on page 977). Based on the OD matrix, pedestrian inputs, routing decisions and routes for pedestrians are automatically generated.

- Pedestrian routing decisions must contain at least one route per pedestrian type that belongs to the pedestrian composition. The pedestrian composition must be defined in the pedestrian input. The pedestrian input must lie within the same area as the pedestrian routing decision.
- In each area that contains one or more pedestrian inputs, there must be at least one pedestrian routing decision.
- The area of the pedestrian input, all areas with intermediate points and the area of the route destination must be connected to each other via areas or ramps & stairways.

10.4.2 Inputs, routing decisions and routes guide pedestrians

When pedestrians are added to the network via pedestrian inputs, they are guided to their destination via routes obtained through routing decisions.

10.4.2.1 Using pedestrian inputs to add pedestrians to a network

You can define pedestrian inputs on pedestrian areas or have them automatically generated from an OD matrix. In doing so, you also generate routing decisions and routes to destinations (see "Pedestrian OD matrices" on page 977).

- If you define pedestrian inputs for pedestrian areas, you also need to define routing decisions for pedestrians.
- At least one route per pedestrian is required that is included in a time interval of the pedestrian input and belongs to the pedestrian composition of the input. This is automatically the case, when for the routing decision, you specify the attribute **AllPedTypes** (**All pedestrian types**). However, if you choose a pedestrian class that only includes some of the pedestrian types, a pedestrian type might be used at the pedestrian input, but no route will be defined for it. In this case, the simulation would be canceled and an error message displayed. A routing decision, for which the relative volumes of all routes are set to NULL for a time interval, is treated as if it did not exist.
- A routing decision assigns each newly generated pedestrian a route that has been defined for his pedestrian type or pedestrian class. If there is more than one route for a specific pedestrian type, a route is chosen randomly according to the relative volumes of indi-

vidual routes.

- Each route leads via a sequence of areas and ramps, which form the route locations.

10.4.2.2 Strategic routes guide pedestrians throughout the network

- A pedestrian always walks towards his next routing point.
- When a pedestrian reaches a route location which has been assigned a time distribution in the attribute **TmDistr** (**Time Distribution**), he waits for the period defined.
  - If this was not the pedestrian's last route location, he walks on to the next route location.
  - However, if this was the last route location of his route and if the area of the route location contains a routing decision with routes for this pedestrian type or class, he is assigned a new route that he will continue to follow.
  - If there is no route for the pedestrian, he is removed from the network.
  - If the area with the last route location contains a pedestrian input, the pedestrian is removed from the route, as it is assumed that routing decisions on areas that contain a pedestrian input are specifically meant for new pedestrians coming from this input.
  - If the last route location lies in an area that has been assigned the attribute **PTUsage** > **Waiting area**, the pedestrian remains in the network, if there is no routing decision for this area. He waits for a suitable public transport vehicle to board (see "Attributes of areas" on page 898) and (see "Modeling pedestrians as PT passengers" on page 984).
- There may be several "tactical" options for pedestrians to get from one route location to the next. For example, they can pass by obstacles, keeping left or right or use stairways and ramps to get to their destination. Moreover, pedestrians must be able to sidestep each other. Viswalk automatically performs the calculations and makes the decisions for such actions during the simulation. In multi-level scenarios, Viswalk internally computes a routing graph. The routing graph consists of the route locations you defined and additionally contains a route location for both ends of each stairway and ramp on the route. The routing graph thus includes relevant information on all routing variants that lead via the same level or via different levels that are connected via stairways or ramps. Based on the routing graph, Viswalk determines the shortest path between two route locations. The shortest path is used for the pedestrian's tactic movement. In Viswalk, pedestrians prefer routes without stairways or ramps, even if these are shorter. If instead you would like pedestrians to use stairways or ramps, place your route locations on the desired ramps.

10.4.2.3 Requirements for assigning routing decisions

The following requirements must be fulfilled for a pedestrian to be assigned a new route.

- A routing decision has been defined for this area.
- A routing decision becomes effective for its pedestrian type through class dependency.
- The pedestrian currently has no route. The might be due to the following:
  - The pedestrian was newly generated.

- In this time step, the pedestrian has reached the final route location of his current or previous route.
- The pedestrian has alighted from a public transport vehicle and is entering an area that has been assigned the attribute **PTUsage** > **Platform edge** (see "Attributes of areas" on page 898) and (see "Modeling pedestrians as PT passengers" on page 984)..

10.5 Parameters for pedestrian simulation

The parameters of the model can be categorized into the following groups:

- Parameters of the original model (by pedestrian type)
- Parameters of the model extensions for Vissim (by pedestrian type)
- Implementation-specific global parameters include all discretization parameters, which are necessary, for example, for models formulated using a continuous time. As analytical approaches to solve the differential equations are only possible in scenarios of limited size from every point of view, time needs to be discretized in some way to make a simulation on a computer possible.

10.5.1 Defining model parameters per pedestrian type according to the social force model

You can set parameters for each pedestrian type derived from the original model. In addition, you can set Vissim-specific parameters for each pedestrian type.

- **tau (τ)** (see "Defining walking behavior" on page 932)

  Tau represents the relaxation time or inertia that can be related to a response time, as it couples the difference between desired speed and desired direction v_0 with the current speed and direction v for acceleration $a : a = (v\_0 - v)/\tau$.

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\01 - Tau

---

- **lambda_mean (λ_mean)** (see "Defining walking behavior" on page 932)

  Lambda governs the amount of anisotropy of the forces from the fact that events and phenomena in the back of a pedestrian do not influence him (psychologically and socially) as much as if they were in his sight. Based on $lambda$ and the angle φ between the current direction of a pedestrian and the source of a force a, factor w is calculated for all social (e.g. non-physical) forces that suppress the force, if:

  $\varphi \neq 0$ and $\Lambda < 1 : w(\Lambda) = (\Lambda + (1 - \Lambda)(1 + \cos(\varphi))/2$

  Based on the above, $\varphi = 0$ then $w = 1$ and $\varphi = \Pi$ yields $w(\Lambda) = \Lambda$

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\06 - Lambda

- **A_soc_isotropic** and **B_soc_isotropic** (see "Defining walking behavior" on page 932)

  These two parameters and λ govern one of the two forces between pedestrians:

$$F = A\_soc\_isotropic\, w(\Lambda)\ \exp(-d/B\_soc\_isotropic)\, n$$

  with d as distance between the pedestrians (body surface to body surface) and n as unit vector, pointing from one to the other.

- **A_soc_mean**, **B_soc_mean** and **VD** (see "Defining walking behavior" on page 932)

  These parameters define strength (A) and the typical range (B) of the social force between two pedestrians. The social force between pedestrians is calculated according to the following formula, if the influencing pedestrian is in front of the one being influenced (180°) and exerts his influence from the front (+/- 90°), otherwise it is zero:

$$F = \ A \exp(-d/B)n$$

  Thereby the following applies:

  d, in the simplest case of $VD = 0$, is the distance between two pedestrians (body surface to body surface).

  n is the unity vector, pointing from the influencing to the influenced pedestrian.

Note: In addition, the relative velocities of the pedestrians are considered, if parameter $VD > 0$.

If parameter $VD > 0$, distance d is generalized and replaced by:

$$\widetilde{d_{01}} = \frac{1}{2}\sqrt{\left(d_{01} + d_{a,01}\right)^2 - \left|\vec{v}_{rel,01} VD\right|^2}$$

Where

- d_{01}: current distance between two pedestrians 0 and 1
- $d_{a,01}$: expected distance between two pedestrians on the basis of VD in seconds, if both pedestrians keep their speed:

$$d_{a,01} = \left|\left(\vec{x}_0 + \vec{v}_0 VD\right) - \left(\vec{x}_1 + \vec{v}_1 VD\right)\right| = \left|\overrightarrow{d_{01}} + \vec{v}_{rel,01} VD\right|$$

Apart from the last term below the root, the geometric mean between the current and expected distance is calculated and applied.

d points from the influencing to the influenced pedestrian, wit $|d| = d$. The force is calculated for the "influenced pedestrian".

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\09 - VD

- **noise** (see "Defining walking behavior" on page 932)

The greater this parameter value, the stronger the random force that is added to the systematically calculated forces if a pedestrian remains below his desired speed for a certain time.

Checking the noise value effect:

Have a group of pedestrians pass a narrow alleyway of approx. 70 cm width.

With noise = 0, so called pedestrian "arches" will form and remain stable. If the noise value lies within the range [0.8 to 1.4], one of the pedestrians will step back after a while and another one will pass through. Default 1.2

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

You can find further information in the following folder:

..\Examples Training\Pedestrians\Parameter Demonstration\10 - Noise

- **react_to_n** (see "Defining walking behavior" on page 932)

During calculation of the total force for a pedestrian, only the influence exerted by the n closest pedestrians is taken into account. Default 8.

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\03 - React to N

- **queue_order**: **degree of orderliness of a queue** and **queue_straightness**: **degree of straightness of a queue** (see "Selecting network settings for pedestrian behavior" on page 204), (see "Attributes of areas" on page 898)

These two parameters specify the shape of queues. Their value range is 0.0 - 1.0. The greater these parameter values, the more straight the queue will look.

- **side_preference** (see "Defining walking behavior" on page 932)

This parameter defines whether opposing pedestrian flows prefer using the right or the left side when passing each other:

-1: for preference of the right side

1: for preference of the left side

Default *0*: no preference, behavior as before: pedestrians do not shun each other

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\ 25 - Side Preference

10.5.2 Defining global model parameters

For each type of pedestrians, global model parameters can be set, apart from model parameters. Global model parameters are used as default values when you define pedestrian routes. When changing global model parameters, you do not change the parameters of the pedestrian routes already defined.

- **grid_size**: **Search neighborhood grid size** (see "Selecting network settings for pedestrian behavior" on page 204)

  With this parameter you can define the maximum distance at which pedestrians have an effect upon each other. Default 5 m.

  The pedestrians are stored in a grid with cells of size **grid_size x grid_size** square meters. A pedestrian in a cell interacts with pedestrians from the following 8 cells only:

  - the 4 adjoining cells
  - the 4 cells which hit a pedestrian's cell with one of their corners.

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\02 - Grid Size

- **routing_obstacle_dist**: **Default obstacle distance** (see "Selecting network settings for pedestrian behavior" on page 204)

  This parameter only has an effect on the calculation of the static potential. It is not considered in the calculation of the dynamic potential. The static potential is also called distance potential or distance look-up table. When calculating the value of the static potential the value of grid cells which are close to a wall increases, in addition to the value resulting from the distance from the exit. This ensures that the pedestrians choose a wide corridor rather than a narrow passage when they have both options on their way from A to B, and neither option is shorter than the other. Generally, the pedestrians maintain a certain distance to the walls. This parameter specifies the distance, up to which the nearby walls have a bearing on the distance potential. Default 0.50 m.

Tip: Your Vissim installation provides example data and a description for testing with these parameters.

..\Examples Training\Pedestrians\Parameter Demonstration\02 - Grid Size

- **cell_size**: **Cell size** (see "Attributes of pedestrian route locations" on page 958)

  This parameter defines the distances of control points to be set for the calculation of distances to a destination area. Default 0.15 m.

- **never_walk_back**: Option **Never walk backwards** (see "Selecting network settings for pedestrian behavior" on page 204)

  Use this parameter to define whether pedestrians should stop, if the direction of movement and the desired direction differ by more than 90° (1 = on, 0 = off):

  - No calculation, if the default value is 0.
  - With parameter value 1, the scalar product is derived from the desired direction and the calculated speed: With a negative value, the pedestrian will stop.

> Tip: In the following directory you will find sample files, which illustrate the mode of action of these parameters, as well as a description of the mode of action of the parameters in the modeling of the dynamic potential:
>
> *..\Examples Training\Pedestrians\Enhanced Routing\Dynamic Potential*

- **Dynamic_potential_g**: **g (general strength)**(see "Defining the Dynamic Potential for a static pedestrian route" on page 971)

  This parameter defines how in general the loss time for an occupied grid cell is estimated in relation to an unoccupied one. Default 1.5

- **Dynamic_potential_h**: **h (direction impact)**(see "Defining the Dynamic Potential for a static pedestrian route" on page 971)

  With this parameter you can define the influence of speed on the estimated travel time of a cell. Default 1.5

  - If parameter value is 0, there is no influence.
  - Parameter value of 1.0 means that the influence of a pedestrian with a typical desired speed can either double, when he goes in the opposite direction, or can drop to zero, when he is heading towards the destination.

- **Dynamic_potential_direction_change_clipping**: **Limit direction change** (see "Selecting network settings for pedestrian behavior" on page 204)

  Use this parameter to determine whether the angle between the fastest and the shortest route can increase at any speed, i.e. whether it can increase randomly from one time step to the next.

  - Default value of 1 specifies the allowable degree of deviation with the parameter **Direction change angle Dynamic_potential_ direction_change_p**.
  - If parameter value is 0, no test is made.

- **Dynamic_potential_direction_change_p**: **Direction change angle** (see "Selecting network settings for pedestrian behavior" on page 204)

  With this parameter you can define the maximum permitted angle, by which the angle between the quickest and the shortest path can increase from one time step to the next. If a larger deviation results for a position from the dynamic potential, the direction of the quickest path is determined from the direction of the quickest path from the last turn plus this angle. Default 4 degrees

This parameter value is only considered if for **Dynamic_potential_direction_change_ clipping** (**Limit direction change**), you have entered the value 1.

10.5.3 Using desired speed distributions for pedestrians

You can use desired speed distributions for pedestrians that are described in the specialized literature. You can allocate desired speed distributions for pedestrians to the following types of base data:

- The desired pedestrian types in pedestrian compositions (see "Attributes of pedestrian compositions" on page 931)
- The desired pedestrian classes in area behavior types (see "Defining area behavior types" on page 934)

The walking behavior of pedestrians and the simulation results are strongly affected by the various desired speed distributions. Upper and lower limits for pedestrian speed and intermediate points, which mark the changes in speed, are defined in the desired speed distributions for pedestrians.

Viswalk contains the following desired speed distributions for pedestrians that are described in the specialized literature (see "Attributes of desired speed distributions" on page 239):

No.	Name	Description
1020	**IMO-M <30**	Speed distribution for male pedestrians on ships, age class under 30 **International Maritime Organization**: Guidelines for evacuation analysis for new and existing passenger ships (as IMO below)
1021	**IMO-F <30**	Same as **IMO-M <30**, but for females
1022	**IMO-M 30-50**	Speed distribution for male pedestrians on ships, age class 30 to 50. **International Maritime Organization**
1023	**IMO-F >30-50**	Same as **IMO-M 30-50**, but for females
1024	**IMO-M >50**	Speed distribution for male pedestrians on ships, age class over 50. **International Maritime Organization**
1025	**IMO-F >50**	Same as **IMO-F >50**, but for females
1026	**IMO-M M1**	Speed distribution for male pedestrians on ships with reduced mobility, age class over 50. **International Maritime Organization**
1027	**IMO-F M1**	Same as **IMO-F >50**, but for females
1028	**IMO-M M2**	Speed distribution for male pedestrians on ships with severely reduced mobility, age class over 50. **International Maritime Organization**
1029	**IMO-F M2**	Same as **IMO-F >50**, but for females
1040	**Fruin 1**	Speed distribution according to **Fruin, J.J.**: In: Pedestrian Planning and Design, 1971, PEDESTRIAN PLANNING AND DESIGN
1041	**Fruin 2**	Speed distribution according to **Fruin, J.J.**: In: Pedestrian Planning and Design, 1971, PEDESTRIAN PLANNING AND DESIGN

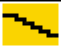
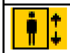
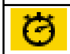
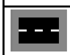
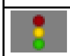
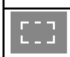
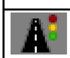
No.	Name	Description
1042	**Predt-Milinski**	Speed distribution for pedestrians in buildings. **Predtechenski, V.; Milinski, A.**: Planning for Foot Traffic Flow in Buildings. Amerino Publishing Co., New Delhi, 1978
1043	**Stairs Kretz 1**	Speed distribution based on measurements of the upward speed of pedestrians at the top end of long stairs. **Kretz, T.; Grünebohm, A.; Kessel, A.; Klüpfel, H.; Meyer-König, T. and Schreckenberg, M.**: Upstairs walking speed distributions on a long stairway. In: Safety Science 46(1) p. 72-78. 2008.
1044	**Stairs Kretz 2**	Speed distribution based on measurements of the upward speed of pedestrians at the top end of long stairs. **Kretz, T.; Grünebohm, A.; Kessel, A.; Klüpfel, H.; Meyer-König, T. and Schreckenberg, M.**: Upstairs walking speed distributions on a long stairway. In: Safety Science 46(1) p. 72-78. 2008.
1045	**Airport - S.B. Young**	Speed distribution for pedestrians in airports. **Young, S.B.**: journal article: Evaluation of Pedestrian Walking Speeds in Airport Terminals and full papers: Evaluation of Pedestrian Walking Speeds in Airport Terminals.
1046	**Airport - S.B. Young**	Speed distribution for pedestrians on moving walkways in airports. The speeds of pedestrians on the moving walkway are given relative to the speed of the walkway, so the speeds of some pedestrians are close to ZERO. Young and Viswalk define the speed distribution differently: - Young includes pedestrians who are standing on the moving walkway, in their speed distribution. - In Viswalk, you enter a value for the attribute **Walking percentage** (**WalkPerc**) of the relevant moving walkway. The difference is taken into consideration in Viswalk: In Viswalk, the speed difference 1009 of Young begins at 0.77 km/h. Therefore enter 74.5 % for the attribute **Walking percentage** (**WalkPerc**). **Young, S.B.**: journal article: Evaluation of Pedestrian Walking Speeds in Airport Terminals and Evaluation of Pedestrian Walking Speeds in Airport Terminals.

10.6 Network objects and base data for the simulation of pedestrians

The following network objects and base data are of particular importance for the modeling of pedestrians. In Vissim, using the network objects toolbar, you can hide network object types that are only relevant for vehicles and thus only show network object types that you need for pedestrian simulation. Viswalk does not display network object types that are only relevant for vehicles.

10.6.1 Displaying only network object types for pedestrians

1. Right-click in the Network Objects toolbar.
2. From the context menu, select the entry **Pedestrian Object Types Only**.

Symbol	Network object type
	Areas
	Obstacles
	Ramps & Stairs
	Elevators
	Pedestrian Inputs
	Pedestrian Routes
	Pedestrian Attribute Decisions
	Pedestrian Travel Times
	Pedestrians In Network
	Links
	Priority Rules
	Signal Heads
	Detectors
	Sections
	Backgrounds
	3D Traffic Signals
	Static 3D Models
	3D information signs

10.6.2 Base data

In addition to functions and distributions, the following base data is of particular importance:

- Pedestrian Types (see "Using pedestrian types" on page 876)
- Pedestrian Classes (see "Using pedestrian classes" on page 879)

- Walking Behaviors (see " Modeling area-based walking behavior" on page 932)
- Area Behavior Types (see "Defining area behavior types" on page 934)
- Display Types of areas and links (see "Defining display types" on page 320)
- Levels for multistory models (see "Defining levels" on page 922)

10.6.3 Base data in the Traffic menu

- Pedestrian Compositions (see "Modeling pedestrian compositions" on page 930)
- Pedestrian OD matrix (see "Pedestrian OD matrices" on page 977)

10.7 Using pedestrian types

You can combine pedestrians with similar properties in pedestrian types. You may also change pedestrian types during the simulation run, for example via the COM interface.

By default, the following pedestrian types are available:

- Man
- Woman
- Woman & Child
- Wheelchair User

10.7.1 Defining pedestrian types

1. Select from the menu **Base Data** > **Pedestrian Types**.

   The list of defined network objects for the network object type opens.

   By default, you can edit the list (see "Using lists" on page 93).

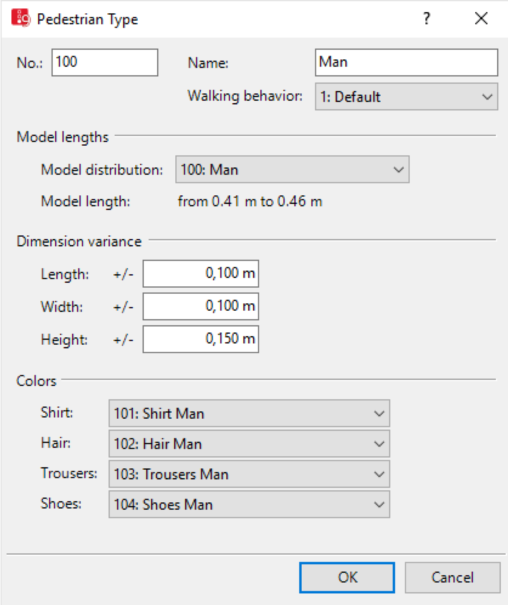
   You can define a new pedestrian type in the list.

2. Right-click in the list.

3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

   *The **Pedestrian Type** window opens.*



4. Edit the attributes (see "Attributes of pedestrian types" on page 877).
5. Confirm with **OK**.

*The attributes are saved in the **Pedestrian Types** list.*

10.7.2 Attributes of pedestrian types

The **Pedestrian Type** window opens when you define a new pedestrian type and have selected to have the Edit dialog opened automatically after object creation (see "Right-click behavior and action after creating an object" on page 152). Into the window, you enter attribute values for the network object. For already-defined network objects, you can call this window via the list of network objects of the network object type (see "Functions available in the shortcut menu of the row header" on page 104).

In the network objects list of the network object type, you can edit all attributes and attribute values of a network object (see "Opening lists" on page 95), (see "Selecting cells in lists" on page 106).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Element	Description
No.	Unique number of pedestrian type
Name	Name of pedestrian type

Element	Description
Walking behavior	**WalkBehav**: Name of walking behavior (see "Defining walking behavior" on page 932)
Model lengths	- **Model distribution** (**2D/3D model distribution**, **2D3DModVert**): Defines appearance, length, width and height of pedestrians of this pedestrian type (see " Using 2D/3D models" on page 219). - **Model length**: Shows length of 2D/3D model with minimum length and maximum length based on 2D/3D model distribution
Dimension variance	- **Length** (**Length variance**, **LngVar**): Scaling range for the object length (3D model length, for example, step length from the tip of the toe of the front foot to the heel of the rear foot) - **Width** (**Width varianceWidVar**): Scaling range for the object width (3D model width, for example, shoulder width) - **Height** (**Height variance**, **HgtVar**): Scaling range for the object height (3D model height, for example, only adults or adults and children)
Colors	Color distributions define the colors of the 3D display for the following attributes for pedestrians of the selected pedestrian type (see "Using color distributions" on page 262): - **Shirt**: **Shirt color distribution** (**ShirtColorDistr**) - **Hair**: **Hair color distribution** (**HairColorDistr**) - **Trousers**: **Trousers color distribution** (**TrousersColorDistr**) - **Shoes**: **Shoes color distribution** (**ShoesColorDistr**) This applies for all objects of a pedestrian type. The attribute **Use pedestrian type color** (**UsePedTypeColor**) must be selected for pedestrian classes.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Pedestrian Classes** (see "Defining pedestrian classes" on page 879).
3. Enter the desired data.

The data is allocated.

10.8 Using pedestrian classes

You can group pedestrian types and combine them into pedestrian classes. This can facilitate the modeling of certain scenarios.

Pedestrian classes are optional. A pedestrian type can belong to several pedestrian classes, but does not have to belong to a pedestrian class.

10.8.1 Defining pedestrian classes

You can define pedestrian classes and assign pedestrian types to the attributes.

1. Select from the menu **Base Data** > **Pedestrian Classes**.

   The list of defined network objects for the network object type opens.

   By default, you can edit the list (see "Using lists" on page 93).

   You can define a new pedestrian class in the list.

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Edit the attributes (see "Attributes of pedestrian classes" on page 879).

10.8.2 Attributes of pedestrian classes

In the **Pedestrian Classes** list, you can edit all attributes and attribute values of a pedestrian class (see "Opening lists" on page 95), (see "Selecting cells in lists" on page 106).

1. Select from the menu **Base Data** > **Pedestrian Classes**.

   The list of defined network objects for the network object type opens.

   By default, you can edit the list (see "Using lists" on page 93).

2. Click on the desired entry.

3. Make the desired changes:

Element	Description
No	Unique identification number of the pedestrian class
Name	Name of pedestrian class

Element	Description
Color	Default color of pedestrian class during simulation (see "Static colors of vehicles and pedestrians" on page 175). Is not used in the following cases: - When for a pedstrian class the attribute attribute **Use pedestrian type color** (**UsePedTypeColor**) is enabled. - When for the display of pedestrians in the network, from the Graphic Parameters menu, **DrawingMode** > **Use color scheme** is chosen, and for the **Color scheme configuration** attribute, a color scheme is specified that is to be used for classification.
PedTypes	**Pedestrian types**: list box with options for selecting the pedestrian types which you want to assign. Numbers and names of pedestrian types.
UsePedTypeColor	**Use pedestrian type color**: ☑ If this option is selected, the colors of the color distribution are used for shirt, hair, trousers, and shoes of the assigned pedestrian type. (see "Attributes of pedestrian types" on page 877). ☐ If this option is not selected, the **Color** attribute of the pedestrian class is used for all pedestrians whose pedestrian type is assigned to the pedestrian class. If a pedestrian type is assigned to two pedestrian classes, the color distribution is used for the shirt of pedestrians of the pedestrian type that is assigned to the pedestrian class with the lower number. The **Use pedestrian type color** attribute is not used, when for the display of pedestrians in the network, from the Graphic Parameters menu, **DrawingMode** > **Use color scheme** is chosen, and for the **Color scheme configuration** attribute, a color scheme is specified that is to be used for classification.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

10.9 Modeling construction elements

The following network objects are accessible construction elements:

- Areas
- Ramps, stairways, escalators, moving walkways

Obstacles are not accessible

10.9.1 Areas, Ramps & Stairs

An area is defined as circle, rectangle or polygon (see " Defining construction elements as circles" on page 894), (see "Defining construction elements as rectangles" on page 890), (see

" Defining construction elements as polygons" on page 893). Areas do not have a specific direction. You can import areas from AutoCAD.

Ramps and stairways are defined as rectangles. You can select different stairway shapes. The shape defines the number and direction of flights and the number of landings. The shape is displayed inside the rectangle.

“Ramp” can therefore be the generic term for construction elements, which connect areas of different levels: ramps, stairways, escalators, moving walkways. A stairway must start at one level (top) and end at another level (bottom).

Areas and ramps may optionally include additional information for pedestrians, e.g. routing decisions (see "Modeling routing decisions and routes for pedestrians" on page 939).

You can define public transport stop areas as waiting areas or platform edges (see "Modeling PT stops" on page 511), (see "Attributes of areas" on page 898).

Construction elements are automatically connected where they are directly adjacent or overlap. Where accessible elements overlap, pedestrians may walk from one element into the next one. Pedestrians do not require any connectors.

Before information on walkable areas is transferred to the pedestrian model, Vissim groups areas touching or overlapping each other into the largest possible walkable polygons. The original edges of these areas are not treated as obstacles. They are pedestrian accessible. So when you split an area during network editing, this does not affect pedestrian simulation in the network.

To model a complex area containing numerous corners, define several polygons in a row that overlap.

Note: Add pedestrian inputs to pedestrian areas as a source of pedestrian flows (see "Modeling pedestrian inputs" on page 936).

Whereas links defined with the attribute **Is pedestrian area** (option **Use as pedestrian area**) are accessible elements, on which you can place signal heads, detectors or conflict areas. They are meant for modeling the interaction of pedestrians with vehicular traffic or other pedestrian flows (see "Modeling links as pedestrian areas" on page 922).

10.9.1.1 Stairway shapes

For each stairway you can select the following shapes (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913):

- Straight
- Straight with landing
- Angle with quarter landing (90°)
- U with half landing (180°)
- U with 2 quarter landings (180°)

The wireframe shows the following stairway elements:

- The contour includes the length and width of the stairway as an enclosing rectangle. The contour thus covers all flights of stairs including the steps and landings, if the stairway has more than one or two landings.
- Parallel lines over the width of each flight of stairs mark the area of the steps.
- A triangle indicates the direction.

If the stairway has several flights of stairs, you can specify the length and width of each flight of stairs using the **Length Flight of Stairs<No>** and **Width Flight of Stairs <No>** attributes.

10.9.2 Escalators and moving walkways

Pedestrians may use escalators and moving sidewalks for automated transport. Escalators and moving walkways have a direction and are defined as rectangles.

10.9.3 Obstacles

Obstacles are not accessible. An obstacle has the same effect on pedestrian dynamics as if you were to model a hole in an otherwise accessible area. Obstacles are defined as circles, rectangles or polygons. You can import obstacles from AutoCAD (see "Importing walkable areas and obstacles from AutoCAD" on page 882).

If an obstacle intersect a ramp, a message is displayed when you start the simulation. The message shows the number of the obstacle, starting with the smallest number, that intersects a ramp and the number of the ramp. Click the **Continue** button to show the next message. Click the **Cancel** button to cancel the start of the simulation, e.g. in order to edit an obstacle.

10.9.4 Deleting construction elements

By default, you may delete construction elements in lists or network editors (see "Deleting network objects" on page 356).

10.9.5 Importing walkable areas and obstacles from AutoCAD

You can import AutoCAD data into Viswalk. Using AutoCAD data, you can create walkable areas and obstacles for pedestrians.

With the help of maps of the area, you create the geometry required for pedestrian flow simulation.

To complete an existing network or create a new one, import polylines in the data format **.dwg* and convert them.

Depending on the parameter settings you select for data import, all objects of an AutoCAD layer are converted into areas or obstacles.

- Objects of the type line or polyline are imported as long as they do not belong to AutoCAD blocks or AutoCAD groups.
- Lines and polylines that have a common point are connected to form a single polyline.

- Closed polylines are converted into polygons that you can import as obstacles or areas. Open polylines are not imported.
- Overlapping polygons and enclosed polygons are imported as overlapping construction elements. They are not interpreted as "holes" in areas or obstacles.
- In the **CAD Import - Configuration** window, you may select several attributes for the objects generated. For some attributes, default values are assigned.
- In Vissim, object names are derived from the concatenation of "Level <x>:" and the respective object name, where available. If there is no object name, a consecutive number is added.

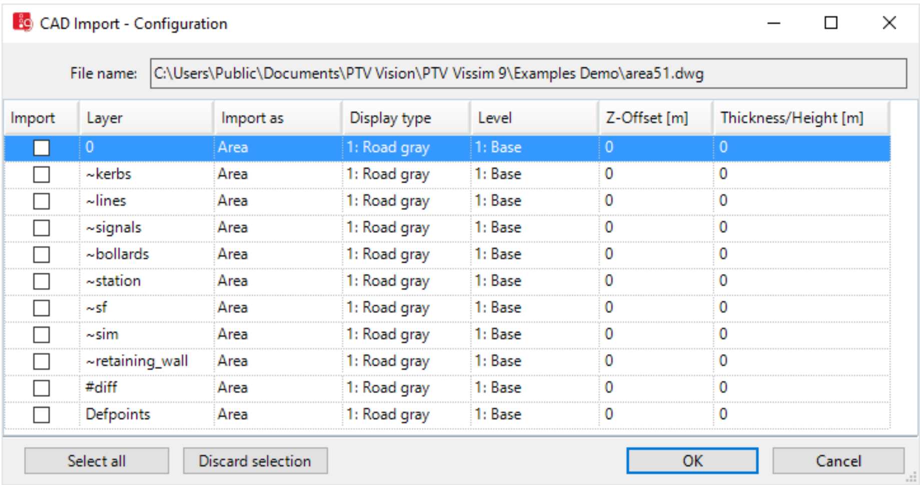
Notes: In the **.dwg* file, the x- and y-coordinates must be specified in meters.
During import of the **.dwg* file, z-coordinates are ignored.

---

1. From the **File** menu, choose > **Import** > **CAD for Pedestrian Areas**.

   *The **Import CAD File** window opens.*
2. Select the **.dwg* file of your choice.
3. Click the **Open** button.

   *The **CAD Import - Configuration** window opens. A level is displayed for each row.*



4. In the columns, select the attributes of your choice.

Column	Description
Import	☑ Select this option to import the respective level.
Layer	Name of the CAD level from the **.dwg* file

Column	Description
Import as	Network object type for the level: - **Obstacle**: Creates the level as an obstacle - **Area**: Creates the level as an area
Display type	Select the **Display type** Display types have to be defined (see "Defining display types" on page 320).
Level	Select **Level** from Vissim for the CAD levels from the **.dwg* file. The level has to be defined in Vissim (see "Defining levels" on page 922).
Z-Offset top/bottom	**Offset above**: Positive value for the distance between the ground and the top edge of the area or the obstacle. **Offset below**: Negative value for the distance between the ground and the bottom edge of the area or the obstacle.
Thickness/Height	- **Thickness**: for areas (see "Attributes of areas" on page 898) - **Height**: for obstacles (see "Attributes of obstacles" on page 910)
Select all	In the **Import** column, selects all layers from the **.dwg* file for import.
Deselect all	Deselects all selected options in the **Import** column.

Notes: The unit used in the window for all lengths corresponds to the unit of **length** for the smallest unit of length selected, for example [m]. Check this setting before import in the menu **Base data** > **Network settings** > **Units** tab.

5. Confirm with **OK**.

   *The **CAD Import - Configuration** window closes.*

The imported polygons are displayed as construction elements in the network editor and in lists. You can edit or delete the construction elements.

10.9.6 Importing Building Information Model files

Building Information Model data is used for building data modeling. The BIM principle is implemented via Industry Foundation Classes (IFC) (IFC - Industry Foundation Classes). IFC is an open data format used to describe building and construction models. IFC provides 3D geometry representation and additional data on other project elements such as doors, walls, ceilings or other building objects). Viswalk can import IFC-based building data for pedestrian simulation. Viswalk converts individual objects into construction elements. Viswalk supports version IFC2x3 (TC1) of the IFC standard (IFC2x Edition 3 Technical Corrigendum 1).

Relevant subcomponents

In the IFC format, all components of the building model are based on a hierarchical structure. For the import to Viswalk the following simplified hierarchy applies to the subcomponents:

IfcProject: the hierarchy's root node

- **IfcSite**: the base area, e.g. the building site. **IfcSite** may include a number of buildings.
  - **IfcBuilding**: a single building
    - **IfcBuildingStorey**: a single level of a building Stories are used to create the levels for the pedestrian areas.
    - **IfcSlab**: Ceiling or floor. An **IfcSlab** is imported as pedestrian area. An obstacle is created on the basis of a hole in the floor.
    - **IfcWall**: Wall. An obstacle is created on the basis of an **IfcWall**.
    - **IfcWindow**: Window. An obstacle is generated on the basis of an **IfcWindow**.
    - **IfcPlate**: A panoramic window or a non-load bearing wall. An obstacle is created on the basis of an **IfcPlate**.
    - **IfcStair**: Stairway
    - **IfcRamp**: Ramp

Further building model components (e.g. elevators or roofs) cannot be imported when using the IFC format.

Steps of BIM import

1. Convert your **.ifc* file to an **.inpx* file using the *IFC2INPX.exe* converter (see "Converting IFC files to INPX files" on page 885).
2. Import the **.inpx* file you created into Viswalk (see "Importing INPX files including building data" on page 890).

Steps in Viswalk

1. Use the network editor to check the location, height and display of the construction elements .
2. Use the network editor and/or **Level** list to check the level data.
3. Remodel your Viswalk, if required.

10.9.6.1 Converting IFC files to INPX files

If you want to use your Building Information Model data from an **.ifc* file in Viswalk, you will have to convert the **.ifc* file to an **.inpx* network file. You can then import the network file (see "Importing INPX files including building data" on page 890).

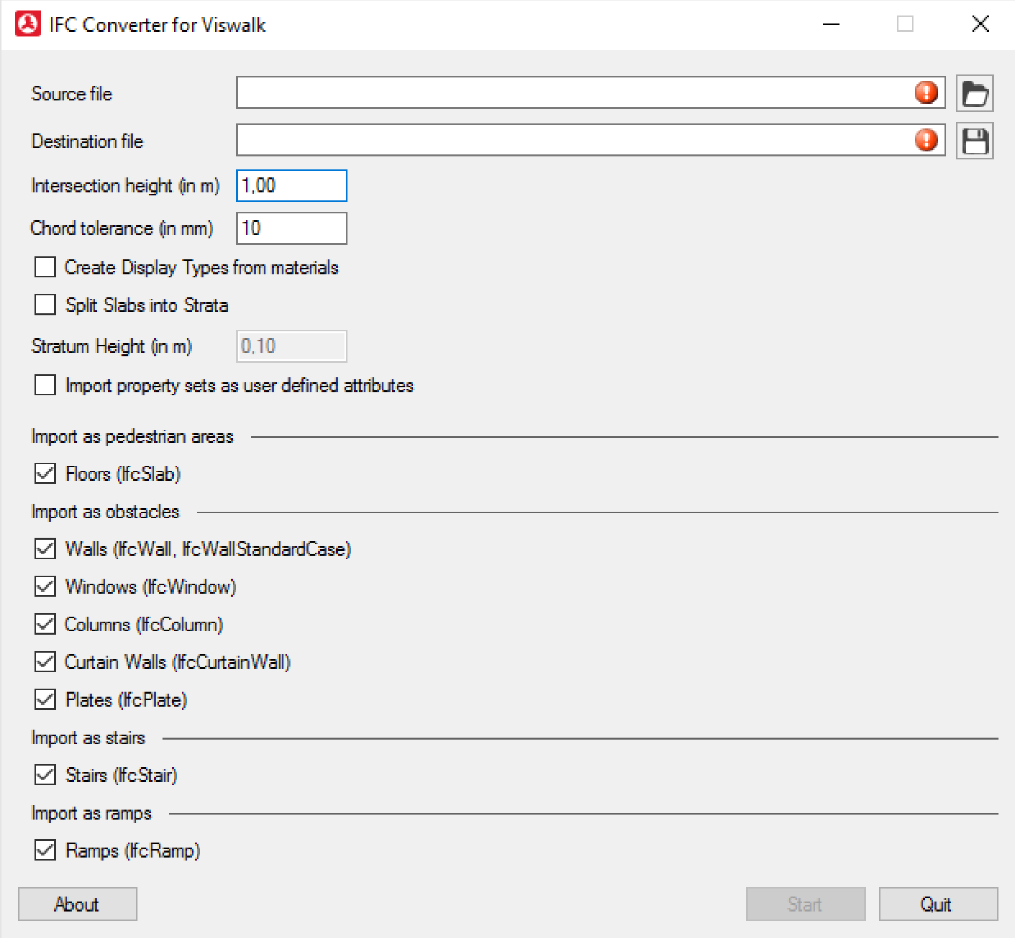
You may start the conversion process via Viswalk or a **.bat* batch file (see " Starting conversion in Viswalk" on page 885), (see "Starting conversion via a batch file" on page 887).

Starting conversion in Viswalk

The conversion process may take a few minutes, depending on the data volume. The conversion is run in the external *IFC2INPX_GUI.exe* application. It is not run in Viswalk. This allows you to continue to work in Viswalk during the conversion process. *IFC2INPX_GUI.exe* is stored to the *..\exe\IFC2INPX* directory of your Viswalk installation.

1. From the **File** menu, choose > **Import** > **BIM converter (**.ifc*)**.

   *The **IFC Converter** window opens.*



2. Make the desired changes:

Element	Description
Source file	**.ifc* file that is to be converted to an **.inpx* file.
Target file	**.inpx* file that is created by a conversion
Intersection height (in m)	The height above each imported level that runs in parallel to the base area in which Viswalk checks the building model's vertical subcomponents in terms of openings in order to identify window/door openings and obstacles. Default 1.00 m
Chord tolerance (in mm)	Viswalk uses polygons for modeling circular objects in IFC. The **chord tolerance** value specifies the maximum distance between the edges of the polygon and the actual circle before creating an additional corner point. Default 10.00 mm

Element	Description
Creating display types using materials	Creating a display type using each material imported.
Stratifying slabs	Modeling sloped ceilings or floors based on the BIM by stratifying the resultant pedestrian areas. Multiple pedestrian areas with different Z-coordinates are created. The attribute **Stratum height (in m)** defines the vertical distance.
Stratum height (in m)	Indicates height of the terrace. For floors above the base area. Default 0.10 m
Imported property sets as user-defined attributes	Creating user-defined attributes based on the IFC property sets.
Import as pedestrian areas	**Floors (IfcSlab)**: Selecting floors to create construction elements based on the network object type **pedestrian area**.
Import as obstacles	Selecting the following subcomponents to create construction elements based on the network object type **Obstacles**: - **Walls (IfcWall, IfcWallStandardCase)**: - **Windows (IfcWindow)**: - **Columns (IfcColumn)**: - **Curtain Walls (IfcCurtainWall)**: - **Plates (IfcPlate)**:
Import as stairs	**Stairways (IfcStair)**: Selecting stairs to create construction elements based on the network object type **Stairs**.
Import as ramps	**Ramps (IfcRamp)**: Selecting ramps to create construction elements based on the network object type **ramps**.
About	Information on *IFC2INPX*, the software used and contact details
Start	Starting conversion to the **.inpx* file

3. Import the network file (see "Importing INPX files including building data" on page 890).

Starting conversion via a batch file

You must create a batch file for the conversion of an **.ifc* file to an **.inpx* network file. Here you will have to enter the conversion parameters.

The conversion may take a few minutes, depending on the data volume. The conversion runs on the external *IFC2INPX.exe* application. *IFC2INPX.exe* is stored to the *..\exe\IFC2INPX* directory of your Viswalk installation.

1. Create a blank **.txt* file in a directory of your choice.
2. Name the **.txt* file according to your requirements.
3. Rename **.txt* file with a **.bat* file extension.

*In the *.bat file, the conversion parameters have to follow the following scheme:*

```
ifc2inpx [inputfile[outputfile]] [/IntersectionHeight=h] [/Bogentoleranz=c] [/IfcEle-
ments=e] [/ImportPropertySets] [/DefaultStairWidth=w] [/ImportMaterials] [/SplitSlab-
sIntoStrata] [/StratumHeight=s]
```

Parameter	Description
inputfile	Input file: Path to the **.ifc* file to be converted.
outputfile	Output file: Path to the **.inpx* file which the network created by conversion should be saved to.
IntersectionHeight	Intersection height (see " Starting conversion in Viswalk" on page 885): At intersection height, Ifc elements are intersected above each imported level (in parallel to the base area) in order to find out whether network file elements need to be stored as obstacles. Without this parameter the default value is 1.00 m.
h	Intersection height in meters, decimals > 0
ChordTolerance	Chord tolerance (see " Starting conversion in Viswalk" on page 885): The **chord tolerance** value specifies the maximum distance between the edges of the polygon and the actual circle before generating an additional corner point in the polygon. Without this parameter the default value is 10.00 mm.
c	Chord tolerance in millimeters, decimals from 0 to 1,000
IfcElements	List of Ifc elements to be converted during import. All elements enabled for conversion will be converted by default.
e	List of Ifc elements to be converted during import (separated by commas) such as Slabs, Walls, Windows, Columns, CurtainWalls, Plates, Stairs, Ramps
ImportPropertySets	Create user-defined attributes based on the IFC property sets. Without this parameter the property sets will not be imported.
DefaultStairWidth	Default value for the width of stairways, if the width cannot be specified on the basis of the geometry. Without this parameter the default value is 2.0 meters.
w	Default value for the width of stairways in meters, decimals > 0
ImportMaterials	Creating a display type that belongs to an imported element, using each imported material Material is imported by default.
StratifySlabs	Stratifying slabs: Modeling sloped ceilings and floors by stratifying the resultant pedestrian areas. Multiple pedestrian areas with different Z-coordinates are created. Without this parameter, slabs will not be stratified.
StratumHeight	If the **StratifySlabs** parameter is set: Defines the stratum height at which slabs are intersected. Without this parameter the default value is 0.1 meter.
s	Stratum height in meters, decimals > 0

4. Enter the conversion parameters of your choice in the **.bat* file.
5. Save the **.bat* file.

   *You can run the *.bat file at the time of your choice and thus start conversion of an *.ifc file to an *.inpx network file:*
6. Double-click the **.bat* file.

*The batch process initiates the conversion and stores an *.inpx network file.*

An example of two project conversions

Two projects are converted to two **.inpx* files:

```
ifc2inpx c:\projects\project1.ifc c:\projects\Viswalk\project1.inpx
ifc2inpx c:\projects\project2.ifc c:\projects\Viswalk\project2.inpx
```

An example of a project conversion with different intersection heights

A project with different intersection heights is converted to six **.inpx* files. Default values will be used, if parameters are not specified.

```
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-01.inpx /In-
tersectionHeight: 0.1
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-05.inpx /In-
tersectionHeight: 0.5
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-10.inpx /In-
tersectionHeight: 1.0
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-15.inpx /In-
tersectionHeight: 1.5
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-20.inpx /In-
tersectionHeight: 2.0
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-25.inpx /In-
tersectionHeight: 2.5
```

Example of a project conversion using the fcElements parameter

If six projects mentioned in the example above contain areas and ramps, you may use the **fcElements** as follows:

```
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-01.inpx /In-
tersectionHeight: 0.1
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-05.inpx /In-
tersectionHeight: 0.5 /IfcElements: Walls,Windows,Columns,CurtainWalls,Plates
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-10.inpx /In-
tersectionHeight: 1.0 /IfcElements: Walls,Windows,Columns,CurtainWalls,Plates
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-15.inpx /In-
tersectionHeight: 1.5 /IfcElements: Walls,Windows,Columns,CurtainWalls,Plates
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-20.inpx /In-
tersectionHeight: 2.0 /IfcElements: Walls,Windows,Columns,CurtainWalls,Plates
ifc2inpx c:\projects\projectX.ifc c:\projects\Viswalk\projectX-ISH-25.inpx /In-
tersectionHeight: 2.5 /IfcElements: Walls,Windows,Columns,CurtainWalls,Plates
```

In this case, areas and ramps are only included in the first **.inpx* file you created. Using the **Read Additionally** feature in Viswalk allows you to combine this data with the conversion results you get from other **.inpx* files.

Show list of possible parameters

- Enter `ifc2inpx` without further parameters in the Windows command line prompt.

The list of possible parameters is displayed.

10.9.6.2 Importing INPX files including building data

Once you have converted an **.ifc* file to an **.inpx* network file, you can import it into Viswalk.

1. From the **File** menu, choose > **Read Additionally** > **Network**.
2. The window **Select file to read additionally** opens.
3. Select the **.inpx* network file you converted.
4. Click on **Open**.

   Network objects such as levels and construction elements are created. The following colors are assigned to the elements:

   - Gray: Walkable pedestrian area
   - Green: Walls
   - Orange: Holes in accessible pedestrian areas
   - Red: Elements with ambiguous IFC data or files that do not comply with the openBIM standard.
5. Use the network editor to check and correct the red elements
6. and check the location, height and display of the other construction elements.
7. Go to the network editor and/or **Level** list to check the level data.
8. Correct and remodel your Viswalk network, if required.

10.9.7 Defining construction elements as rectangles

You can define all types of construction elements as rectangles:

Icon	Network object type of the construction element
	Areas You may also define areas as polygons or circles (see " Defining construction elements as polygons" on page 893), (see " Defining construction elements as circles" on page 894)
	Obstacles You may also define obstacles as polygons or circles.
	Ramps & Stairs, **escalators** and **moving walkways** For technical reasons, a 10 cm wide obstacle is automatically added on both sides of a ramp. This reduces the accessible area to the surrounding areas.

Note: If two levels should be connected via a ramp, stairway, moving walkway or escalator, both levels must be defined (see "Defining levels" on page 922).

Tips: Alternatively, you can define construction elements in the Network Editor via the context menu > **Add New <Name Construction element type>** when a network object type is selected in the network object toolbar.

You can drag the rectangle to the desired size in the Network editor.

You can also define the rectangle for an area, ramp or stairway by entering numerical values for the length, width and angle. This is not possible for obstacles.

10.9.7.1 Defining a construction element by dragging it with the mouse

1. Click on the network object toolbar on the desired network object type of the construction element **Areas**, **Obstacles** or **Ramps & Stairs**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click again **Areas**, **Obstacles** or **Ramps & Stairs**.
3. Select **Rectangle**.

   When you insert a ramp, stairway, a moving walkway or an escalator, drag the network object towards the construction element that the pedestrian should use.

4. Using the mouse pointer, point to the desired position of the first corner point of the construction element.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

5. Press the CTRL key, hold down the right mouse button and drag the mouse to the desired end point of the link.
6. Release the keys.

   The length of the construction element is set.

7. Drag the mouse pointer sideways to the desired width.
8. Double click.

The construction element is shown in color in the Network Editor. In ramps and stairways, escalators and moving walkways, the tip of a triangle shows in the direction towards which pedestrians can use the construction element. The ***<Name Construction element type>*** *window opens.*

9. Edit the attributes of the inserted construction element type:
   - Attribute of areas (see "Attributes of areas" on page 898)
   - Attribute of obstacles (see "Attributes of obstacles" on page 910)
   - Attribute of ramps, stairways, moving walkways and escalators (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913)
10. Confirm with **OK**.

The attributes are saved in the list of the construction element type. All construction elements are numbered in ascending order by default, independent of the individual type of construction element.

10.9.7.2 Defining an area, ramp or stairway by entering length, width and angle

1. On the network object toolbar, click the desired network object type of the construction element **Areas** or **Ramps & Stairs**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button ▾ *is shown.*
2. Again click **Areas** or **Ramps & Stairs**.
3. Select **Rectangle**.
4. Using the mouse pointer, point to the desired position of the first corner point of the construction element.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

5. Press the CTRL key and hold down the right mouse button.

   A transparent window opens showing the current numerical values of the length, width and angle. You can overwrite the numerical values without clicking in the window:
6. Keep the CTRL key pressed and type in a numerical value for the length.
7. Confirm with ENTER.
8. Continue to keep the CTRL key pressed and type in a numerical value for the width.

9. Confirm with ENTER.
10. Continue to keep the CTRL key pressed and type in a numerical value for the angle.
11. Confirm with ENTER.
12. Release the keys.

    The length, width and angle of the construction element are defined. The construction element is shown in color in the Network editor. In ramps and stairways, escalators and moving walkways, the tip of a triangle shows in the direction towards which pedestrians can use the construction element. The ***<Name Construction element type>*** *window opens.*

13. Edit the attributes of the inserted construction element type:
    - Attribute of areas (see "Attributes of areas" on page 898)
    - Attribute of obstacles (see "Attributes of obstacles" on page 910)
    - Attribute of ramps, stairways, moving walkways and escalators (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913)
14. Confirm with **OK**.

The attributes are saved in the list of the construction element type. All construction elements are numbered in ascending order by default, independent of the individual type of construction element.

10.9.8 Defining construction elements as polygons

You can define areas and obstacles as polygons, circles,or rectangles. All other types of construction elements can be defined as rectangles only (see "Defining construction elements as rectangles" on page 890) , (see " Defining construction elements as circles" on page 894).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. Click on the network object toolbar on the desired network object type of the construction element **Areas** or **Obstacles**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click again **Areas** or **Obstacles**.
3. Select **Polygon**.

4. Using the mouse pointer, point to the desired position of the first corner point of the construction element.
5. Press the CTRL key and right-click on the desired starting point of construction element and keep the mouse button pressed.
6. In order to insert multiple polygon points, repeat the next two steps.
7. Drag the mouse pointer in the desired direction.
8. Right-click the next desired point.
9. Double-click, when you no longer wish to insert additional polygon points.

   The last polygon point is connected with the first polygon point. The polygon is closed.

   The construction element is shown in color in the Network Editor. The ***<Name Construction element type>*** *window opens.*

10. Edit the attributes of the inserted construction element type:
    - Attribute of areas (see "Attributes of areas" on page 898)
    - Attribute of obstacles (see "Attributes of obstacles" on page 910)

The attributes are saved in the list of the construction element type. All construction elements are numbered in ascending order by default, independent of the individual type of construction element.

11. Confirm with **OK**.

Tips:

- You can define areas and obstacles, in which you import CAD data from files in file format **.dwg* (see "Importing walkable areas and obstacles from AutoCAD" on page 882). Thereby, you can complete an available network for pedestrians or generate a new network.
- Alternatively, you can define construction elements in the Network Editor via the context menu > **Add New <Name Construction element type>** when a network object type is selected in the network object toolbar.

10.9.9 Defining construction elements as circles

You can define areas and obstacles as circles, polygons or rectangles. All other types of construction elements can be defined as rectangles only (see "Defining construction elements as rectangles" on page 890), (see " Defining construction elements as polygons" on page 893).

A circular construction element is inserted by default with a contour containing 48 points. You can use these points to enlarge or reduce the construction element, but you cannot change the circular shape. If you want to change the circular shape, insert another point into the contour. This way, the **circle** is turned into a **polygon**, so that you may move or delete any point in the contour.

You can drag the circle to the desired size in the Network editor or enter a numerical value for the radius.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

Tips:

- You can define areas and obstacles by importing CAD data from files in the file format *.*dwg* (see "Importing walkable areas and obstacles from AutoCAD" on page 882). This way you can complete an existing pedestrian network or generate a new network.
- Alternatively, you can define construction elements in the Network Editor via the context menu > **Add New <Name Construction element type>** when a network object type is selected in the network object toolbar.

10.9.9.1 Defining a construction element by dragging it with the mouse

1. On the network object toolbar, click the desired network object type of the construction element **Areas** or **Obstacles**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click again **Areas** or **Obstacles**.
3. Select **Circle**.
4. Using the mouse pointer, point to the desired position of the construction element.
5. Press the CTRL key and right-click on the desired starting point of the construction element and keep the mouse button pressed.

   A transparent window opens and displays the current radius and the default 48 points of the contour.

6. Drag the mouse pointer in the desired direction.
7. Once the construction element has reached the desired size, release the mouse button and the CTRL key.

   *The **<Name Construction element type>** window opens. The construction element is shown in color in the Network Editor.*

- Area: gray, contour: yellow with 48 points
- Obstacle: red, contour: yellow with 48 points

8. Edit the attributes of the inserted construction element type:

   - Attribute of areas (see "Attributes of areas" on page 898)
   - Attribute of obstacles (see "Attributes of obstacles" on page 910)

The attributes are saved in the list of the construction element type. All construction elements are numbered in sequence by default, independent of the individual type of construction element.

9. Confirm with **OK**.

10.9.9.2 Defining construction elements by entering a radius

Entering a value for the radius allows you to define the circle in precisely the desired or required size.

1. On the network object toolbar, click the desired network object type of the construction element **Areas** or **Obstacles**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button  *is shown.*

2. Click again **Areas** or **Obstacles**.
3. Select **Circle**.
4. Using the mouse pointer, point to the desired position of the construction element.
5. Press the CTRL key and right-click on the desired starting point of the construction element and keep the mouse button pressed.

   A transparent window opens and displays the current radius and the default 48 points of the contour. You can overwrite the value for the radius without clicking in the window:

6. Keep the CTRL key pressed and type in a numerical value.
7. Release the mouse button and the CTRL key.

   *The **<Name Construction element type>** window opens. The construction element is shown in color in the Network Editor.*

   - Area: gray, contour: yellow with 48 points
   - Obstacle: red, contour: yellow with 48 points

8. Edit the attributes of the inserted construction element type:

   - Attribute of areas (see "Attributes of areas" on page 898)
   - Attribute of obstacles (see "Attributes of obstacles" on page 910)

The attributes are saved in the list of the construction element type. All construction elements are numbered in sequence by default, independent of the individual type of construction element.

9. Confirm with **OK**.

10.9.10 Editing construction elements in the Network Editor

Unlike other network objects, you can edit the polygons and rectangles of construction elements with additional functions. The level, on which the construction element is located, may not be blocked (see "Defining levels" on page 922).

10.9.10.1 Inserting another point on the border line of a polygon

1. Click on the network object toolbar on the network object type of the construction element **Areas** or **Obstacles**.
2. In the Network Editor, right click on the desired position on the border of the area or the obstacle.
3. From the shortcut menu, choose **Add Point**.

The point is inserted. You can move the point in order to model the shape of the polygon.

10.9.10.2 Moving the polygon point

1. Click on the network object toolbar on the network object type of the construction element **Areas** or **Obstacles**.
2. Click in the Network Editor on the polygon point, hold down the mouse button and drag the mouse pointer to the desired position.
3. Release the mouse button.

10.9.10.3 Deleting the polygon point

1. Click on the network object toolbar on the network object type of the construction element **Areas** or **Obstacles**.
2. In the Network Editor, click on the polygon point, hold down the mouse button and move the polygon point to an adjacent polygon point of the construction element.
3. Release the mouse button.

10.9.10.4 Rotating the construction element

You can rotate construction elements like other network objects (see "Rotating network objects" on page 354)

10.9.10.5 Enlarging or reducing rectangular construction elements

1. Click on the network object toolbar on the network object type of the construction element **Areas** or **Obstacles**.
2. Click in the Network Editor on a corner of the area, obstacle, ramp or stairway; hold down the mouse button and move the mouse pointer to the desired position.
3. Release the mouse button.

If the area contains colored dots for pedestrian inputs, pedestrian routes, or pedestrian travel times that would lie outside the area if you changed its shape, Vissim automatically moves them back into the area.

10.9.10.6 Enlarging or reducing circular areas or obstacles

1. Click on the network object toolbar on the network object type of the construction element **Areas** or **Obstacles**.
2. In the Network Editor, enlarge the view of the construction element until you can identify the individual spline points in the contour of the construction element.
3. In the Network Editor, click one of the spline points, hold down the mouse button and drag the mouse pointer to the desired position.
4. Release the mouse button.

If the area contains colored dots for pedestrian inputs, pedestrian routes, or pedestrian travel times that would lie outside the area if you changed its shape, Vissim automatically moves them back into the area.

10.9.10.7 Moving one side of the construction element

You may move any side between two corner points of a rectangle or polygon, both vertically and horizontally. Adjacent sides are thus extended or shortened.

1. Point the mouse pointer to the side of your choice.

   The mouse pointer appears as double arrow.

2. Hold the mouse button down and drag the mouse horizontally or vertically to move the side of the construction element to the position of your choice.
3. Release the mouse button.

10.9.11 Attributes of areas

The **Pedestrian Area** window opens when you insert an area, if the automatic opening of the Edit dialog after object creation is selected (see "Right-click behavior and action after creating an object" on page 152).

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

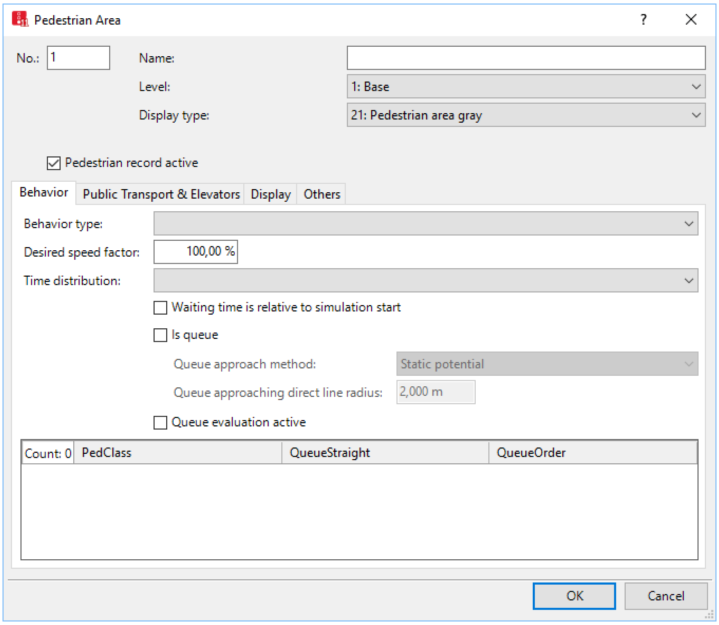
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



A network object may have the following attributes: These can be shown in the attributes list.

Element	Description
No.	Unique number of the area
Name	Name of area
Level	Level if the area for the modeling of multistory buildings is to be on one level (see "Defining levels" on page 922)
Display type	Display type for the colored display of the area (see "Defining display types" on page 320)

The options can be used, for example, for modeling tunnels and underpasses in 2D because the option selected during the simulation is considered only if it corresponds to the relevant selected pedestrian attribute **Show individual pedestrians** or **Show classified values**.

Element	Description
Pedestrian record	**Pedestrian record active** (**PedRecAct**): ☑ If this option is selected, the pedestrians in this area are taken into consideration in the pedestrian record (see "Saving pedestrian record to a file or database" on page 1053).

1. Select the desired tab.

2. Make the desired changes:

10.9.11.1 Behavior tab

Element	Description
Behavior type	Optionally, select an area behavior type (**AreaBehavType**). Is used to model occasional changes to the speed or other parameter (see " Modeling area-based walking behavior" on page 932). If no area behavior type is selected or if an area behavior type has not been assigned a walking behavior, the walking behavior assigned to the pedestrian type is used.
Desired speed factor	Factor for changing the desired speed of all pedestrians within the area, standard value 100 %, value range 10 % to 300 %. With the desired speed factor you can reduce the desired speed in the respective area, if the pedestrians move slower compared to their original desired speed, for example to take into account traveling on rough terrain or crossing a road very carefully. With the desired speed factor you can increase the desired speed in the respective area, if the pedestrians move faster compared to their original desired speed, for example when they cross a road very quickly. If areas overlap, Vissim applies the following rules to determine the area whose desired speed factor is used: - The area with the higher vertical position is preferred. This position results from the z coordinate of the level and the z offset of the area. - The area whose default value has been changed to 100% of the desired speed factor is preferred. - The smaller area is perferred. - The area with the lower number is preferred. If an area behavior type is selected for the area, its desired speed is multiplied by the desired speed factor.
Time distribution	**TmDistr**: Optionally, select a time distribution, which is allocated to pedestrians who enter the area according to their strategic routes. Time distribution on input areas have no effect on pedestrians which are used on this area of the network. For pedestrian areas with **PT usage** of the type **Platform edge** or **Waiting area**, you can define a minimum dwell time via the allocation of a time distribution. PT vehicles depart once the minimum dwell time runs out. The PT vehicles depart once all alighting passengers have exited. This also applies for a minimum dwell time = 0. Optionally you can activate the option **Late boarding possible** for each line stop (see "Calculating dwell time with PTV Viswalk" on page 534).

Element	Description
Waiting time is relative to the start of simulation	**WaitTimeIsRelToStartSim**: If a time distribution is selected for the area and the pedestrian route has a route location within the area, the pedestrian will be allocated a waiting period from the time distribution as soon as he/she enters the area. The waiting period can refer to the following points in time: • ☑ If this option is selected, the waiting time refers to the simulation start. The pedestrian continues on his/her route as soon as the simulation second corresponds to the time of the time distribution. If the simulation second has already exceeded the time of the time distribution when the pedestrian enters the area, the pedestrian immediately continues on his/her pedestrian route. • ☐ If this option is not selected, the waiting time refers to time when the pedestrian enters the area. This option cannot be selected, if the **Queuing** attribute is selected.
Is queue	**IsQueue**: Attribute for the waiting behavior on pedestrian areas which are the destination or intermediate point of a route. ☑ If this option is selected, queuing pedestrians build a queue during their dwell time. If no time distribution is allocated, the positioning only occurs for a few seconds. The positioning is effected according to the orientation vector during generation of the area. In wireframe view, the vector is displayed as a dashed line with an arrowhead. Define the desired queue behavior in the list on the **Area Behavior** tab for the queue and assign the desired pedestrian class. This option cannot be selected if the **Waiting time is relative to the start of simulation** attribute is selected.
Queue approaching method	Method used to calculate how pedestrians approach the end of a queue. • **Direct line**: Pedestrians approach the end of the queue in a direct line. This method requires only minimal computing time. However, pedestrians may be permanently blocked by obstacles. • **Static potential**: Pedestrians approach the end of the queue up to the distance defined in the attribute **Queue approaching direct line radius** based on a static potential. This allows pedestrians to circumvent the obstacles, taking the shortest path possible. Within this radius, they walk the remaining distance to the end of the queue in a direct line. The static potential is calculated based on the position of the pedestrian standing at the end of the queue. Viswalk checks this position with each time step. If the position changes, the static potential is re-calculated. This method allows pedestrians to pass through obstacles. In queues that Viswalk automatically generates next to PT vehicles and elevators, pedestrians always approach the end of the queue in direct line.

Element	Description
Queue approach direct line radius	(**QueueApproachingDirectLineRadius**): Radius around the current end of the queue within which the method **Direct line** is used. Outside this radius, the selected queue approaching method is used. Default 2.0 m
Queue evaluation active	**Queue Evaluation Active**: ☑ If this option and the option **Is queue** are selected, you may output result attributes for the queue (see "Evaluating pedestrian density and speed based on areas" on page 1034).

Behavior within the queue

By default, pedestrians move to the position of the person in front them when they leave their position. For each queue selected for the queue approaching method **Static potential**, Viswalk calculates an additional potential as soon as the simulation is started. This potential shows at the beginning of the queue. It becomes effective each time you update the queue. The distance to the person in front of each pedestrian in the queue is calculated. If the distance is greater than 2 meters, pedestrians do not move up in the queue in direct line, but in walk mode according to the new potential. This mode ends as soon as the pedestrians are closer than 2 meters to the person in front of them. Then pedestrians move up in the queue in direct line again. This procedure allows for a more realistic moving up in line of the pedestrians, particularly when queues are angled, for example in front of a desk at the airport, where shut-off systems are used to create a zigzag queue.

Define queue behavior and assign pedestrian class

1. In the **Area Behavior** tab, activate the ☑ **Queuing** option.
2. Right-click in the list.
3. From the shortcut menu, choose **Add**.

   A new line for the pedestrian class with the lowest number is added.
4. Make the desired changes:

Element	Description
PedClass	Select the **pedestrian class** for which the queue order and/or the queue straightness needs to be defined

QueueStraight	The **Queue straightness** defines the shape of the queue, with values between 0.0 and 1.0. The greater the value, the more straight the queue will look: - 0.0: snake shaped queue - 1.0: straight queue If no waiting behavior is defined for this waiting area, the attribute applies to all elevators of the elevator group, to all floors and all pedestrian classes. Default values are defined in the network settings (see "Selecting network settings for pedestrian behavior" on page 204). They are used if this attribute is not defined for the elevator group. Using the coupled list **Queuing behavior**, you can limit the shape of the queue to the desired pedestrian classes, and for the selected pedestrian classes, define the queue shape. The **Is queue** attribute of the area must be selected.
QueueOrder	The **Queue order** defines the orderliness of the queue formed by pedestrians, with values between 0.0 and 1.0. The higher this value, the more orderly pedestrians line up one behind the other: - 0.0: Pedestrians are standing together in groups - 1.0: Queue of pedestrians lined up one behind the other If no waiting behavior is defined for this waiting area, the attribute applies to all elevators of the elevator group, to all floors and all pedestrian classes. Default values are defined in the network settings (see "Selecting network settings for pedestrian behavior" on page 204). They are used if this attribute is not defined for the elevator group. Using the coupled list **Queuing behavior**, you can limit the orderliness of the queue to the desired pedestrian classes, and for the selected pedestrian classes, define the orderliness. The **Is queue** attribute of the area must be selected.

10.9.11.2 PT & Elevators tab

PT section

Element	Description
Usage	Usage of area in PT: - **None**: Area not used for PT - **Waiting area**: Location, in which the pedestrians wait, in order to board at the allocated public transport stop in the desired PT transport line. When the area is allocated at least one public transport stop, select for the pedestrians an arbitrary point in the waiting area where they are waiting for the next PT vehicle. Default color for the waiting area: blue. - **Platform edge**: Location, to which pedestrians go, when they alight from their PT line to the allocated public transport stop. Alighting passengers will always use the nearest platform edge. Afterwards, they follow the routing decision, which is placed on this area. If there is no routing decision defined for this area, the pedestrians are removed from the network. For a platform edge, a width of at least 2 m is required. An area of the type Polygon, which serves as a platform edge, can be generated automatically with a public transport stop (see "Generating platform edges" on page 517). You can select the side and thus specify on which side of the PT stop boarding is allowed (see "Attributes of PT stops" on page 513). A platform edge can be allocated to multiple PT stops. Default color for the platform edge: pink. As soon as you allocate public transport usage with the attribute **Waiting area** or **Platform edge** and a public transport stop with the attribute **for public transport stop(s)** to the area, Vissim checks whether boarding volumes are defined for this stop. If no boarding volumes are defined, Vissim enters default data in the **Public transport stop** window on the **Boarding volumes** tab.
for PT stop(s)	Allocating one or more public transport stops to the area. The area must be a **Waiting area** or a **Platform edge**. As soon as you select a PT stop and confirm the entries with **OK**, Vissim changes the following data of the PT stops affected (see "Attributes of PT stops" on page 513): - For the **Areas** attribute, the number of areas with the selected attribute **Public transport usage** is entered. - For the **Pedestrians as passengers** attribute, the option is selected. - In the **Boarding Passengers** tab, a line with data is created.

Element	Description
Boarding location	As a standard, for pedestrian areas with **Public transport usage** for the **Boarding location** the option **Nearest door** is used. Alternatively, for the door selection when boarding, you can allocate a location distribution after the queuing boarding passengers have distributed for the boarding at the doors of the PT vehicle (see "Using location distributions for boarding and alighting passengers in PT" on page 249).

Elevators section

Element	Description
Waiting area for elevator group	**ElevGrp**: Elevator groups whose elevators take the area into account. When a pedestrian enters an area that includes a route location of his route, an elevator is requested depending on the following route location.

PT & Elevators section

Element	Description
Waiting behavior	**PTWaitBehav**: Waiting behavior of pedestrians in public transport waiting area and elevators: - **Wait at fixed location**: Prevents pedestrians from waiting outside the waiting area. Each pedestrian attempts to reach his randomly determined waiting position in the waiting area before he stops to wait for the next public transport vehicle or the next elevator. If, in the case of a boarding passenger, a relevant PT vehicle has already arrived at the public transport stop or the next elevator has already arrived, this pedestrian is no longer routed to a randomly determined position at the waiting area. Instead, the pedestrian tries to go directly to the PT vehicle or elevator as soon as he reaches the waiting area, provided that the PT vehicle or elevator has an additional dwell time of at least two more seconds.

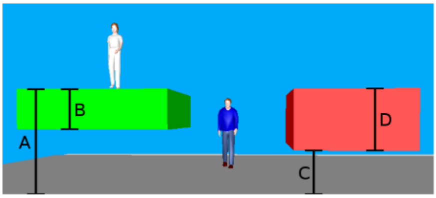
Element	Description
Waiting behavior	- **Wait if held up**: If pedestrians whose destination is a waiting area are closer than 5 m to the waiting area and who are walking more slowly than 0.4 m/s for 3 time steps, Vissim consider these pedestrians as "arrived at the waiting area". In this case, the pedestrians remain standing and waiting, even if they are not yet at the waiting area. This can cause unrealistic jams in the simulation because queuing pedestrians create a permanent obstacle. In order to avoid this, you can select the **Wait at fixed location** attribute or define a sufficiently large boarding capacity for each waiting area. From a larger waiting area, you can build multiple small waiting areas to reach a better area distribution for the waiting pedestrians. Alternatively, you can define an intermediate destination in a large waiting area, in which the pedestrians can be distributed in the waiting area. In addition, for destinations on waiting areas, use the dynamic potential (see "Dynamic potential" on page 968). This causes pedestrians who are already queuing to be more aware of the incoming pedestrians and to choose their paths respectively. The dynamic potential can extend the calculation time.
Waiting position approach method	Method according to which pedestrians in a waiting area move towards a waiting position - **Direct line**: Pedestrians approach the waiting position in a waiting area in a direct line. This method requires only minimal computing time. However, pedestrians may be permanently blocked by obstacles. - **Static potential**: Pedestrians approach a waiting position in a waiting area based on a static potential. The potential is calculated as soon as pedestrians reach the waiting area. Then they move along the potential gradient to the waiting position, up to a certain distance that is calculated based on the **cell size**. This allows pedestrians to circumvent the obstacles, taking the shortest path possible. If the distance of the pedestrians to the waiting position is less than the diagonal of the cell size, they move to the waiting position in a direct line. If they are pushed back from this distance, they start moving again based on the static potential. If an elevator door opens or a PuT vehicle stops, pedestrians will move directly towards them. **Cell size:** Edge length of a grid mesh used for the calculation of distances to the destination area with the static or dynamic potential (see "Defining global model parameters" on page 871). Default 0.15 m. **Obstacle distance**: Distance up to which the nearby walls have an impact on the distance potential (see "Defining global model parameters" on page 871). Default 0.5 m.

Note: If, at a public transport stop at least one pedestrian area is defined as a **Waiting area** or **Platform edge**, Vissim assumes that the boarding/alighting at this public transport stop is calculated during the simulation and that the dwell time should not only be determined numerically regarding the number of boarding/alighting passengers.

The difference becomes visible in the window **PT Line Stop**: The content of the window changes as soon as a pedestrian area is defined as a **Waiting area** or **Platform edge** (see "Editing a PT line stop" on page 526).

10.9.11.3 Display tab

3D section

Element	Description
z-offset	**Z-offset:**: Vertical offset > 0.000 along the z axis up to the specified edge for the 3D display of the area. z = 0 is the floor on which the pedestrians walk (**A** in the figure below) - If **Thickness > offset**, the floor rises in 3D from below 0 - If **Thickness < offset**, the floor seems to float in 3D
Thickness	Thickness of area for the 3D display (B in the figure below). Not relevant for the simulation. Thickness > 0 for an area reduces the clearance displayed below the area in the 3D view because the thickness of the ceiling is not considered when calculating the length of the opening or the ramp foot from the clearance height.  Area left (green): A = offset top, B = thickness Obstacle right (red): C = offset bottom, D = height

Visibility section

Element	Description
Individual pedestrians	(**ShowIndivPeds**): This option is considered if the pedestrian attribute **Show individual pedestrians** is selected. - ☑ Select this option to show individual pedestrians in the area during the simulation. If for a coordinate there is contradictory data on multiple areas, the area setting with the smallest surface area is applied. - ☐ If this option is not selected, aggregated data is shown in the simulation.

Element	Description
Show classified values	**Classified values ShowClsfValues**: This option is considered if the pedestrian attribute **Show classified values** is selected. Then the global LOS scheme for this area can be ignored for the area-based LOS display (see "Using LOS schemes for showing aggregated pedestrian values" on page 186). ▪ ☑ If the option is selected, classified values are displayed in the area during the simulation (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182) ▪ ☐ If the option is not selected, no classified values are displayed in the area during the simulation.
Label	☐ If this option is not selected, the label for the area is not displayed, even if label for all areas is selected.

10.9.11.4 Others tab

Usage in pedestrian OD matrix

Element	Description
Always use as origin area	**Always use as origin area**: Origin areas are considered to be origins (rows) in the pedestrian OD matrix (see "Selecting origins and destinations in the Pedestrian OD Matrix" on page 978). ☑ If the option is selected, the area is displayed as the **origin** in the **Pedestrian OD matrix**. ☐ If the option is not selected, the area is included in the matrix as an **origin** if at least one pedestrian input and at most one static routing decision are specified for it.
Always use as destination area	**Always use as destination area**: Destination areas are taken into account as destinations (columns) in the pedestrian OD matrix (see "Selecting origins and destinations in the Pedestrian OD Matrix" on page 978). ☑ If this option is selected, the area in the **Pedestrian OD matrix** is shown as a **destination**. ☐ If this option is not selected, the area in the **Pedestrian OD matrix** is included as a **destination** if a static route from an origin area ends on it.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Pedestrians: Pedestrians within the area. The evaluation may reduce simulation speed.
- Pedestrian routing decision (static) (see "Attributes of static routing decisions for pedestrian routes" on page 960)
- Static pedestrian route locations (see "Attributes of pedestrian route locations" on page 958)
- Static pedestrian route destinations (see "Attributes of pedestrian route locations" on page 958)
- Pedestrian routing decision (partial) (see "Attributes of pedestrian partial routing decisions" on page 962)
- Partial pedestrian route locations (see "Attributes of pedestrian route locations" on page 958)
- Partial pedestrian route destinations (see "Attributes of pedestrian route locations" on page 958)
- Pedestrian inputs (see "Attributes of pedestrian inputs" on page 938)
- Public transport stops (see "Attributes of PT stops" on page 513)
- Points: edit coordinates of the corners
- Queuing behavior: The **Public transport waiting behavior** is described further above.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.9.12 Attributes of obstacles

The **Obstacles** window opens when you insert an obstacle, if automatic opening of the Edit dialog after object creation is selected (see "Right-click behavior and action after creating an object" on page 152).

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

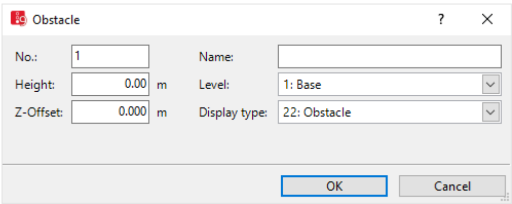
The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

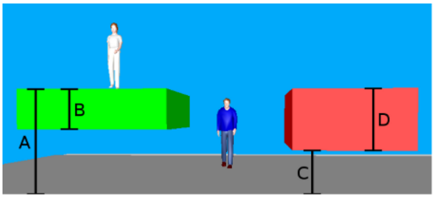
The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).



1. Make the desired changes:

Element	Description
No.	Unique number of the obstacle
Name	Name of the obstacle
Level	Level, if when modeling multistory buildings, you want to place the obstacle on a level (see "Defining levels" on page 922)
z-Offset	**zOffsetB**:Offset > 0.000 along the z-axis up to the edge specified for 3D display of the obstacle (**C** in the figure below)

Element	Description
Height	Height of obstacle for 3D display(**D** in the figure below)  Area left (green): **A** = offset top, **B** = thickness Obstacle right (red): **C** = offset bottom, **D** = height
Display type	Display type for colored display of the obstacle (see "Defining display types" on page 320)

The network object may have additional attributes, for example, the following. These can also be shown in the Attributes list:

Long name	Short name	Description
Created by	**CreatedBy**	Indicates whether the obstacle is user-defined or has been generated by Vissim. - **User**: The obstacle was defined by a Vissim user (see "Defining construction elements as rectangles" on page 890), (see " Defining construction elements as polygons" on page 893) - **3D traffic signal**: The obstacle is the mast of a 3D traffic signal. This obstacle was defined by Vissim during a simulation run (see "Attributes of 3D signal heads" on page 587).

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

2. On the list toolbar, in the **Relations** list, click > **Points**.
3. Edit the coordinates of the vertices.

The data is allocated.

10.9.13 Attributes of ramps and stairs, moving walkways and escalators

The window **Ramps & Stairs** opens when a ramp, stairway, escalator or moving walkway is inserted, if automatic opening of the Edit dialog after object creation is selected (see "Right-click behavior and action after creating an object" on page 152).

Into the window, you enter attribute values for the network object. For network objects which have already been defined, you can call the window using the following functions:

- In the list of network objects of the network object type, double-click the row with the desired network object.
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Edit**.

The network object may have additional attributes. In the network objects list of the network object type, you can show all attributes and attribute values. You can open the list via the following functions:

- In the network object toolbar, right-click the desired network object type. Then from the shortcut menu, choose **Show List** (see "Context menu in the network object toolbar" on page 64).
- In the Network editor, select the network object of your choice. Then, from its shortcut menu, choose **Show In List** (see "Selecting network objects in the Network editor and showing them in a list" on page 359).
- From the **Lists** menu, in the desired category, choose the network object type.

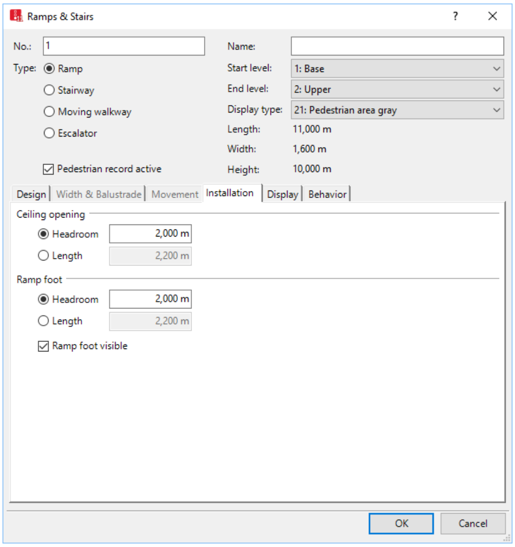
In the network objects list of the network object type, you can edit attributes and attribute values of a network object (see "Selecting cells in lists" on page 106), (see "Using lists" on page 93).

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). On the Lists toolbar, in the **Relations** box, you can show and edit the coupled list with the attributes of the desired relation on the right (see below **Showing and editing dependent objects as relation**) and (see "Using coupled lists" on page 119).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

If the pedestrian should be navigated by the dynamic potential instead of the static potential when using ramps or stairways, you can select the Use dynamic potential option. (see "Dynamic potential" on page 968).

10.9.13 Attributes of ramps and stairs, moving walkways and escalators



1. Make the desired changes:

Element	Description
No.	Unique identification of the construction element (ramp, stairway, moving walkway or escalator)
Type	Select the desired construction element type. The functions that are irrelevant for the selected type will be enabled and displayed in gray on the tabs **Installation**, **Design**, **Width & Balustrade**, **Movement**, **Display** and **Behavior**. ➤ **Ramp**: Define the attributes of the ramp in the **Installation** tab. ➤ **Moving walkway**: Define the attributes of the moving walkway in the tabs **Installation**, **Design**, **Width & Balustrade** and **Movement**. ➤ **Stairway**: Two levels must be defined. Define the attributes of the stairway in the tabs **Installation** and **Design**. ➤ **Escalator**: Two levels must be defined. Define the attributes of the escalator in the tabs **Installation**, **Design**, **Width & Balustrade** and **Movement**. You can model the length, headroom, ceiling opening and other attributes of the construction element (see "Modeling length, headroom and ceiling opening" on page 921). You can define these attributes in the tabs **Installation**, **Design**, **Width & Balustrade** and **Movement**.
Name	Name of the construction element

Element	Description
Start level	(**StartLvl**): Here you can start defining the construction element. A ramp, stairway, moving walkway or escalator can be located within a level, or the two levels **Level (start)** and **Level (end)** can be linked together. You can end the definition of a level when the construction element for the modeling of multistory buildings begins on one level and ends on another (see "Defining levels" on page 922).
End level	**EndLvl** : Here you can end the definition of the construction element.
Display type	Display type for color display of the construction element (see "Defining display types" on page 320)
Length	**Length2D**: Length of the construction element for the definition of the construction element in the Network Editor **Length3D** accounts for the z-Offset of the link.
Width	Width of the construction element
Height	Height of the construction element from the height of the level and the Offset.

2. Select the desired tab.

3. Make the desired changes:

10.9.13.1 Installation tab

Area over and under the construction element.

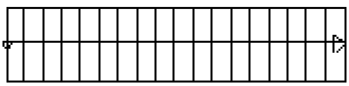
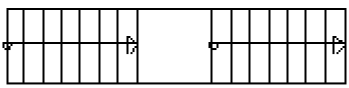
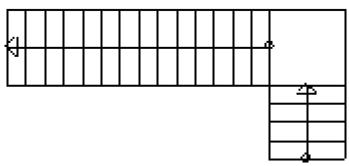
Element	Description
Ceiling opening	➤ **Headroom** (**OpenHeadr**): head clearance ➤ **Length** (**OpenLen**): length of the ceiling opening
ramp foot	➤ **Headroom**: up to the height of the headroom ➤ **Length** (**FootLen**) Length of the ramp foot
Show solid obstacle	**FootVisible**: ☑ If this option is selected, the ramp foot is shown in 3D mode (filled). ☐ If this option is not selected, the ramp foot is shown in 3D mode (not filled).

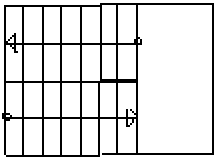
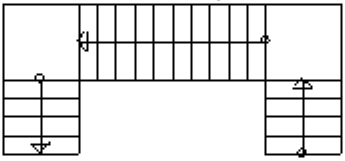
10.9.13.2 Design tab

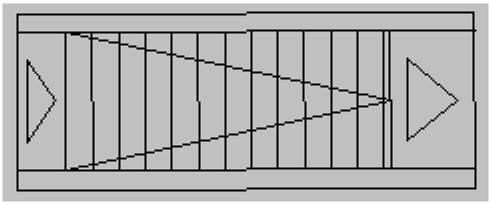
You can edit the attributes on the tab if **Stairway**, **Escalator** or **Moving walkway** is selected as attribute **Type** of the construction element.

For a stairway, the start level and end level must not be identical.

Element	Description
Define stairway by	- **Total steps** (**TreadsDefBy**): Number of steps defined, height of the steps - **Rise**: Height of steps defined, length of the steps - **Going**: Length of steps defined, height of the steps

Element	Description
Geometry	- Go to the **Shape** list box and select one of the following stairway shapes. - The **length** and **width** of the construction element are displayed in the network editor. If the construction element is a stairway, the length and width are displayed for each of the maximum three flights of stairs, **Flight 1**, **Flight 2**, **Flight 3**. For the maximum three flights, only the boxes **Length** and **Width** are activated for entering values that are useful for the geometry of the selected shape of the staircase. The other boxes are grayed out.
Shapes listed in the **Shape**list box:	
Straight	The top and bottom of the stairway are on the same line. The stairway consists of one flight of stairs and has no landing: 
Straight with landing	The top and bottom of the stairway are on the same line. It consists of two flights of stairs. The flights of stairs are connected by a flat landing: 
Angle with quarter landing (90°)	The stairway consists of two flights of stairs. They are connected by a flat landing. The bottom of the second flight of stairs is at an angle of 90° to the first flight of stairs: 

Element	Description
U with half landing (180°)	The stairway consists of two flights of stairs. They are connected by a flat landing. The bottom of the second flight of stairs is at an angle of 180° to the first flight of stairs: 
U with 2 quarter landings (180°)	The stairway consists of three flights of stairs. They connect two flat landings. The bottom of the third flight of stairs is at an angle of 180° to the first flight of stairs: 

Element	Description
Landing platforms	Horizontal, flat, immovable area before and after the construction element: - **Entry Length** (**EntryLandLen**): in front of the construction element - **Exit Length** (**ExitLandLen**): after the construction element - **Display type** (**LandingDisplType**): Select display types for the landing platforms (see "Defining display types" on page 320) - **Horiz. Runout area** (**EntryRunLen**), (**ExitRunLen**), Horizontal runout area: Length from **Entry** and **Exit**, so that the steps continue horizontally with any slope in order to avoid accidents in the transition to and from the walking areas. In the wireframe view, triangles mark the landing platforms before and after the construction element in the direction of movement:  The **Horiz. Runout area** is not displayed.

10.9.13.3 Width & Balustrade tab

Display and measurement of the handrail and the balustrade.

Element	Description
Usable width	Width (**UsableWid**), pedestrians can walk on. Base for the socket width. Value range 600 to 1,200 mm. Alternatively, a warning opens at the start of the simulation.
Handrail **Balustrade** **Socket**	Width and display type of handrail (**HandrWid**), (**HandrDisplType**), balustrade (**BalustrWid**), (**BalustrDisplType**), and socket (**SocketWid**), (**SocketDisplType**). The socket width must be larger than the width of the balustrade and the handrail. *Socket width = (Total width - usable width) / 2*
Show balustrade (3D)	**ShowBalustr**: ☑ If this option is selected, the balustrade is shown in 3D mode. The balustrade does not move.

10.9.13.4 Movement tab

The movement of the construction element has an influence on the result of the simulation.

Element	Description
Treads	➤ **Direction of travel** (**MovDir**): Movement direction, in which the moving walkway or escalator is moving forward. **Forward** = Direction in the definition. ➤ **Speed** (**OpSpeed**): of the moving walkway or escalator. Default 0.50 m/s
Pedestrians - moving	➤ **Classes**: Moving pedestrians of a pedestrian class. Pedestrians of other pedestrian classes are not moving. **All pedestrian types** also include pedestrian types, which are not allocated a pedestrian class. ➤ **Walking percentage** (**WalkPerc**): Share of the pedestrians who are moving on a moving walkway or escalator
Pedestrians - standing	**Standing location**: Side in the direction of travel, on which the pedestrians are located

10.9.13.5 Display tab

Element	Description
z-offset (start)	**Start z-offset** (**StartZOffset**): Offset > 0.000 of the **Level (start)** along the Z axis to the given edge for the 3D graphics mode of the construction element. This is the floor on which the pedestrians walk (A in the figure below) ▶ If **Thickness > offset**, the floor rises in 3D from below 0 ▶ If **Thickness < offset**, the floor seems to float in 3D
z-offset (end)	**End z-offset** (**EndZOffset**): Offset > 0.000 of the **Level (end)** along the Z axis to the given edge for the 3D graphics mode of the construction element. ▶ If **Thickness > offset**, the floor rises in 3D from below 0 ▶ If **Thickness < offset**, the floor seems to float in 3D For a **Stairway** or **Escalator**, the **Z-offset (start)** and **Z-offset (end)** must be different. The value of the z-coordinates are calculated from the height of the respective **Level** and the corresponding **Offset**.
Thickness	Thickness of the construction element for 3D representation Not relevant for the simulation. The thickness > 0 for the construction element reduces the clearance shown in 3D under the construction element because the thickness of the construction element is not considered when the length of the opening or the ramp foot of the clearance is calculated.

10.9.13.6 Behavior tab

The following rule applies to the **area behavior types**: If no area behavior type is selected or no walking behavior is allocated to the area behavior type, the walking behavior allocated to the pedestrian type will be used.

You can optionally select one of the following **area behavior types**:

Element	Description
Flat	Optionally, select an area behavior type (**AreaBehavType**). Is used to model occasional changes to speed or other parameters on a flat construction element, e.g. a moving walkway (see " Modeling area-based walking behavior" on page 932)
Downwards	Optionally, select an area behavior type (**AreaBehavType**). Is used to model occasional changes to speed or other parameters on a ramp, stairway or escalator going downwards. It can be used to model the walking behavior of elderly people or wheelchair users (see " Modeling area-based walking behavior" on page 932).
Upwards	Optionally, select an area behavior type (**AreaBehavType**). Is used to model occasional changes to speed or other parameters on a ramp, stairway or escalator going upwards. It can be used to model the walking behavior of elderly people or wheelchair users (see " Modeling area-based walking behavior" on page 932).

Element	Description
Desired speed factor	Factor for changing the desired speed of all pedestrians on the construction element, standard value 100 %, value range 10 % to 300 %. The desired speed factor allows you to reduce the desired speed on the construction element when pedestrians move slower compared to their original desired speed, for example when the speed on a staircase is only 50% of the speed on a horizontal surface. The desired speed factor allows you to increase the desired speed on the construction element when pedestrians move faster compared their original desired speed, for example when they are moving down a ramp with a gradient. In addition to the reduced speed caused by the uphill slope, the desired speed factor will have an impact on the speed on the ramp.

Element	Description
Cell size	Edge length of a grid mesh which is used for the calculation of distances to the destination area with the static or dynamic potential (see "Defining global model parameters" on page 871). Default value 0.15 m. Avoid values **>** object radius.
Obstacle distance	Distance up to which the nearby walls have a bearing on the distance potential (see "Defining global model parameters" on page 871). Default 0.5 m.
Dynamic potential	**UseDynPot**: ☑ If this option is selected, routing takes place along the path with the lowest estimated travel time that pedestrians can take within a single level. The option enables the input boxes for the parameters of the dynamic potential (see "Dynamic potential" on page 968), (see "Defining the Dynamic Potential for a static pedestrian route" on page 971).
➤ **Impact** ➤ **Calculation interval** ➤ **g (basic force)** ➤ **h (direction impact)**	(see "Dynamic potential attributes" on page 972)

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Pedestrians: Pedestrians within the area. The evaluation may reduce simulation speed.
- Walking - Pedestrian Classes
- Points: edit coordinates of the corners

The attributes are described further above.

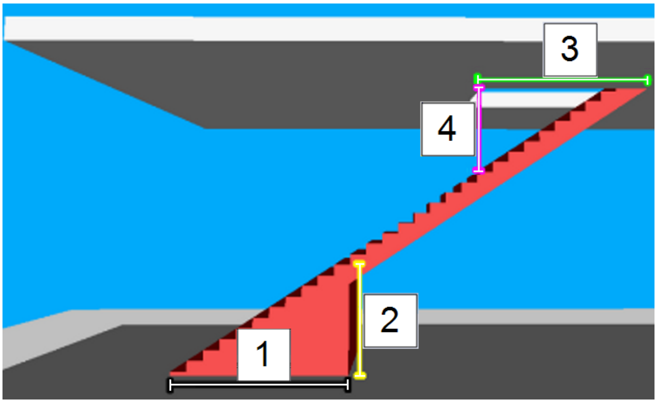
2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.9.14 Modeling length, headroom and ceiling opening

You can model the length, head room and ceiling opening for ramps, stairways, moving walkways and escalators. You can define these attributes in the tabs **Installation**, **Design**, **Width & Balustrade** and **Movement** (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913).



The length, headroom and ceiling opening are marked by lines in the following figure:

- Length (**1**) and head clearance (**2**) on the ramp foot of the stairway are used for the calculation of available space on the floors.
- The ceiling opening at the top of the stairway is determined either by the **length** attribute (**3**) or the **headroom** attribute (**4**).
- The length, headroom and ceiling opening do not affect the pedestrian flows which use the element.

- For the lower floor, the size of the area built is produced from the dimensions of the obstacle.

Note: Define sufficient head clearance. Otherwise the heads of the pedestrians "overrun" the underside of the element.

The thickness > 0 for the construction element, which connects two levels with each other, reduces the height shown in 3D under the construction element because the thickness of the ceiling or the ramp is not considered when the length of the opening or the ramp foot of the clearance is calculated.

10.9.15 Defining levels

You may define multiple levels, e.g. for multistory buildings or bridge structures for links. For levels, you can define links, backgrounds, static 3D models, 3D signal heads and construction elements.

By default, Vissim already contains a level with the **Height** attribute = 0.0.

1. From the **Base Data** menu, choose **Levels**.

   *The **Levels** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Make the desired changes:

Element	Description
No	Unique number of the level
Name	Name of the level
zCoord	**z-coordinate:** z value of layer in meters

*The level is displayed in the **Levels** list and in the Levels toolbar.*

Tip: Using the Levels toolbar, you can show and hide levels and activate or deactivate their selectability (see "Using the Level toolbar" on page 65).

10.10 Modeling links as pedestrian areas

You can define links as pedestrian areas and thus model the interaction between vehicles and pedestrians. Pedestrians become part of the network via pedestrian inputs that you position in areas (see "Modeling pedestrian inputs" on page 936). These areas must overlap the links that are used as pedestrian area so that pedestrians can use these links.

In addition, you can define additional construction elements for pedestrian flows, e.g. areas, ramps or obstacles (see "Modeling construction elements" on page 880).

Using the default settings, you can edit and delete pedestrian areas. When you delete a pedestrian area, the pedestrian link generated for the opposite direction is also automatically deleted.

For pedestrians on links that serve as a pedestrian area, the same mechanisms are used as for vehicles: At a junction where a vehicle path and a pedestrian path intersect, a Vissim link forms the basis for the automatically generated pedestrian area.

In contrast to walkable pedestrian areas, on pedestrian links, pedestrian flows can be controlled via link-based Vissim network objects.

10.10.1 Differences between road traffic and pedestrian flows

For links used as pedestrian areas the following applies:

- They are not directional.
- They cannot be split.
- They cannot contain any intermediate points.
- They have no pedestrian inputs.
- They cannot be the start or end of a connector.

Using the default settings, you can define links as pedestrian areas and edit them (see "Modeling links for vehicles and pedestrians" on page 406).

10.10.2 Differences between walkable construction elements and link-based pedestrian areas

The differences between walkable construction elements (e.g. areas, ramps) and links used as pedestrian areas are as follows:

For links used as pedestrian areas the following applies:

- They cannot be edited as an area.
- They cannot be a ramp.
- The links cannot be the start, end or intermediate point of a pedestrian route.
- A pedestrian area can be positioned on a pedestrian link. There you can define start, destination and intermediate points of pedestrian routes.

10.10.3 Modeling obstacles on links

An obstacle cannot only be added to an area, but also to a link that serves as a pedestrian area. The option **Use as pedestrian area** (Attribute **Is pedestrian area** (**IsPedArea**)) must be selected for this link (see "Attributes of links" on page 409). The steps for adding an obstacle as a polygon or rectangle to a link correspond to the steps for adding an obstacle to an area (see " Defining construction elements as polygons" on page 893), (see "Defining construction elements as rectangles" on page 890).

10.10.4 Network objects for pedestrian links

You can define the following Vissim network objects for a pedestrian link and model the interaction between vehicles and pedestrians or pedestrian flows:

- Conflict areas with road traffic (see "Defining the right of way at conflict areas" on page 564), (see "Modeling conflict areas for pedestrians" on page 926)
- Signal Heads (see "Defining signal groups in the SC editor" on page 639), (see "Modeling signal controls for pedestrians" on page 925)
- Detectors (see "Defining detectors" on page 594), (see "Modeling detectors for ped-estrians" on page 929)
- Priority Rules (see "Defining priority rules" on page 547), (see "Modeling priority rules for pedestrians" on page 929)

10.10.5 Defining pedestrian links

You can define existing links as pedestrian links or create new pedestrian links. On the pedestrian links, you can position signal heads, detectors, priority rules or conflict areas for pedestrians. You cannot insert intermediate points in pedestrian links.

For the pedestrian link, you may enable the Consider Vehicles in Dynamic Potential of Pedestrians option. Pedestrians using the pedestrian link will thus avoid collision with vehicles in a conflict areaVissim, which is located between the vehicle link and the pedestrian link.

1. To insert a new pedestrian link, define the link (see "Defining links" on page 407).
2. After you have added the new link, or when you want to define an existing link as a pedestrian link, display the link attributes in the **Link** window or the **Links** list (see "Attributes of links" on page 409).
3. Open the **Link** window and click the **Pedestrian Area** tab.
4. For the link, select the **IsPedArea** attribute (**Is pedestrian area**).

   Vissim generates the following pedestrian links with identical coordinates. Both pedestrian links overlap. Then the pedestrian link can be used in both directions. Vehicles cannot use a pedestrian link.

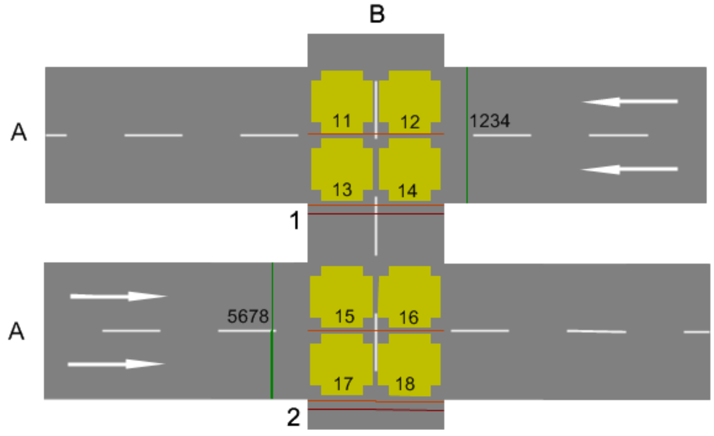
   - Pedestrian link no. 1, with the direction you dragged the link open in the Network editor.
   - Pedestrian link no. 2, with the opposite direction.
5. If pedestrians using the link should avoid collision with vehicles in conflict areas, enable the **Consider vehicles in dynamic potential** option.
6. For the **Level** attribute, select the level of your choice (see "Attributes of links" on page 409), (see "Defining levels" on page 922).
7. If desired, edit the other attributes (see "Attributes of links" on page 409).

8. Confirm with **OK**.

10.10.6 Modeling interaction between vehicles and pedestrians

Model the interaction of vehicles and pedestrians or pedestrian flows with the following network objects:

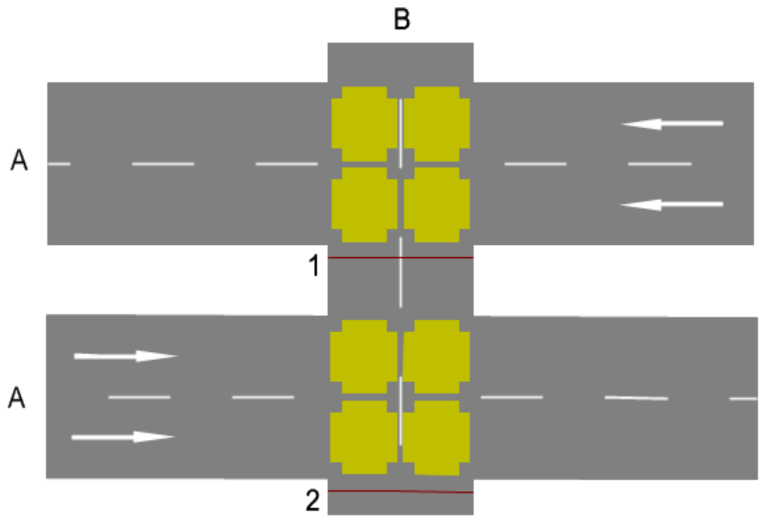
- **Signal control** : in the figure, the red bar (**1**), (**2**) (see "Modeling signal controls for pedestrians" on page 925)
- **Conflict areas**: in the figure, the eight yellow areas (see "Modeling conflict areas for pedestrians" on page 926)
- **Detectors**: in the figure, the four orange lines (see "Modeling detectors for pedestrians" on page 929)
- **Priority rules** (see "Modeling priority rules for pedestrians" on page 929): (**11**) to (**18**) stop lines, (**1234**) and (**5678**) conflict markers
- Links for vehicles (**A**)
- Links for pedestrians (**B**)



10.10.7 Modeling signal controls for pedestrians

1. Ensure that the following network objects are defined:
   - Links for pedestrians in which the attribute **Use as pedestrian area** is selected (see "Defining pedestrian links" on page 924). (**B**) in the figure.
   - Signal control with signal groups on the links (see "Defining signal groups in the SC editor" on page 639). (**1**), (**2**) in the figure.
2. Select the desired **pedestrian classes** in the attribute **PedClasses** of the signal heads which lie in the pedestrian link (see "Attributes of signal heads" on page 579). A pedestrian observes a signal head only if it belongs to a pedestrian type whose pedestrian class is selected in the attribute of the signal head.

The figure shows a modeling example:



The signal head works exclusively in the direction of the pedestrian link at the time when the signal head is inserted on the pedestrian link. The signal head does not work for the opposite direction. The direction of the pedestrian link is indicated by yellow arrows on the edge of the pedestrian link. These are arrows are displayed when you mark the pedestrian link.

- When the signal head is inserted in the direction of the pedestrian link, it is either open or closed for pedestrians walking in this direction. This depends on the dynamic state of the assigned signal group:
  - Red and amber: The signal head is closed. Pedestrians stop at the signal head until it no longer shows red or amber.
  - Other cases: The signal head is open, pedestrians can continue walking on the pedestrian link.
- Pedestrians can always pass the signal head when walking In the opposite direction of the pedestrian link, regardless of its condition. To supply these pedestrians with data via signal heads, reverse the direction of the pedestrian link and insert another signal head. You can then switch back to the original direction of the pedestrian link. This means there are two signal heads on the pedestrian link that have an impact on different directions.

3. To reverse the direction of the link, click the link and hold down the TAB key.

The dynamic state (for example red or green) of the signal head is linked to the dynamic state of the respective signal group to which the signal head belongs.

10.10.8 Modeling conflict areas for pedestrians

1. Ensure that the following network objects are defined:

   - Links for pedestrians in which the attribute **Use as pedestrian area** is selected (see "Defining pedestrian links" on page 924): **B** in the figure.

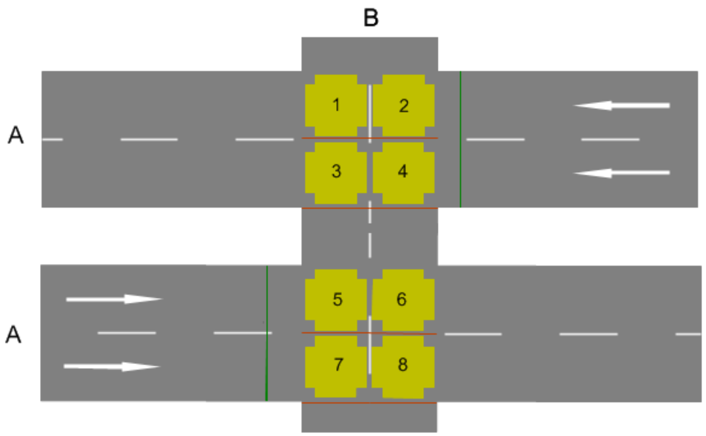
- Links for vehicles in which the attribute **Use as pedestrian area** is deactivated (see "Attributes of links" on page 409): **A1**, **A2** in the figure.

2. On the Network objects toolbar, click **Conflict Areas**.

   Passive conflict areas are marked yellow by default.

3. Repeat the following steps until the desired priority rule is shown.
4. Right-click the desired conflict area.
5. In the context menu, select the entry **Change conflict area status**.

The figure shows a modeling example:



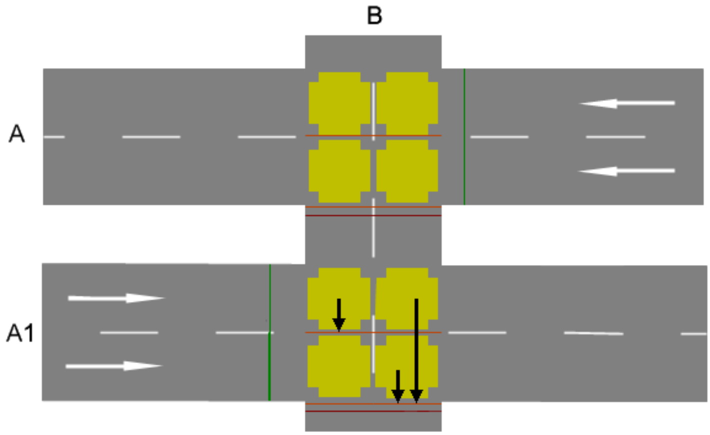
Vissim creates passive conflict areas in the two opposite directions of pedestrian links. You may change the priority rules for the conflict areas (see "Using conflict areas" on page 560). Conflict areas on a link for pedestrians always have the same priority rule: pedestrians have the right of way or must give the right of way. Based on the current priority rule and the current speed as well as desired speed, pedestrians and vehicles decide when they can pass a conflict area.

A conflict area has a stop line distance. The stop line distance describes the upstream position on the link at which the pedestrians or vehicles have to wait if they have to grant the right of way. At a default value of NULL, the stop line position in the direction of travel is immediately before the start of the conflict area. You can position the stop line upstream from the regular stop line of the conflict area.

Note: If the two links intersect at an acute angle, the distance for crossing the road is greater. This also increases the time required by pedestrians to cross the link. Since there is no geometric analysis of the relative position of the two links related by the conflict area, a pedestrian may not have yet left the lane when he reaches the next vehicle.

10.10.8.1 Case 1: Pedestrians yield to vehicles

Pedestrians who want to cross the vehicle link (**A1**) and enter the conflict area on the left lanes of (**A1**) in the figure above have to consider the minimum speed that is calculated on the stop lines where the arrows end.

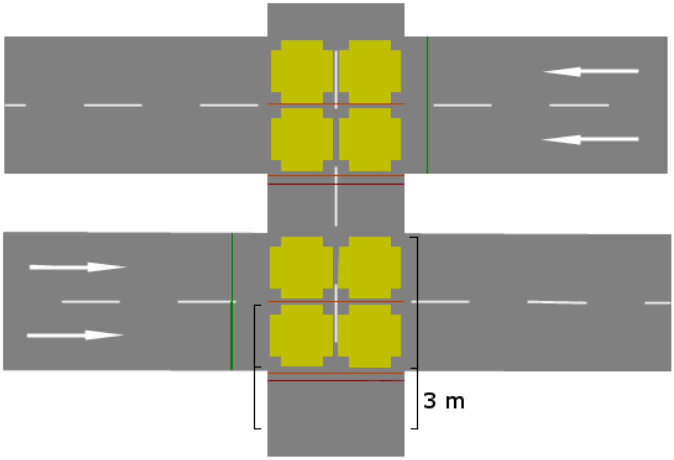


Minimum speeds are calculated dynamically at stop lines: pedestrians using minimum speed or a higher speed can pass the stop line, but other pedestrians have to wait before the stop line.

Pedestrians enter a conflict area with a vehicle link only if they can cross the entire link (in stages) so that they never are on a lane on which a vehicle is just passing through the conflict area. The desired speed of the pedestrian and the current speed of the vehicle are assumed here. Vehicles that are not yet in the network are not taken into account here. Vehicles that are more than 75 m times the number of lanes of the link from the conflict area are also not taken into account. This condition can no longer be fulfilled after entering the conflict area if, for example, a vehicle accelerates or enters the network at a relatively short distance. In this case, the pedestrian waits before the first lane of the vehicle link which he probably cannot cross completely before the arrival of the vehicle.

10.10.8.2 Case 2: Vehicles have to wait

On the pedestrian link, the area 3 m upstream from the conflict area to the start of the conflict area is used as a "detector area" for the pedestrians approaching the conflict area (see "Modeling detectors for pedestrians" on page 929). Pedestrians within the conflict area are always taken into account:



If no pedestrian is expected to enter the conflict area, the gaps between pedestrians or groups of pedestrians are transferred to the approaching vehicles. Thus they can respond accordingly.

10.10.9 Modeling detectors for pedestrians

Detector areas are used to model push buttons for pedestrian signals.

Detectors are also used as SC detectors for pedestrians. In this case, the detector must be placed on the pedestrian link with the signal head (see "Activation tab for pedestrians" on page 597).

10.10.10 Modeling priority rules for pedestrians

You can define priority rules of the vehicular traffic accordingly for conflicting pedestrian streams or for the interaction between vehicular and pedestrian traffic.

Note: Model the standard priority rules for conflicting traffic flows which are not controlled by signals by means of conflict areas. Only use priority rules, if conflict areas do not produce the desired results and if you have sufficient experience in modeling with priority rules.

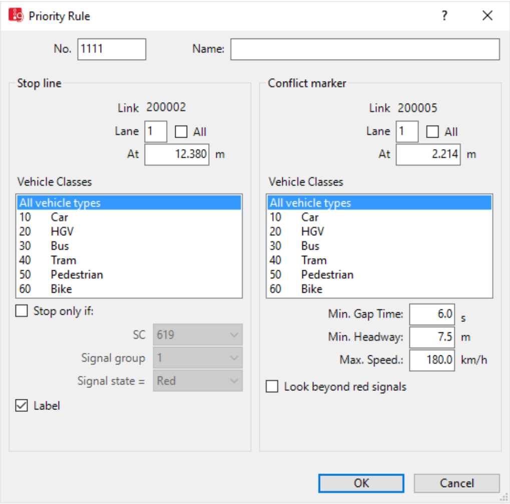
Priority rules for pedestrians must lie on a pedestrian link (see "Modeling links as pedestrian areas" on page 922). Pedestrians may be both the conflict marker and the stop line. With priority rules for pedestrians, the following combinations are possible:

- Pedestrians x Vehicles
- Vehicles x Pedestrians
- Pedestrians x Pedestrians

For these combinations and depending on the link attribute **Use as a pedestrian area**, you can select the vehicle classes and pedestrian classes for the conflict marker and the stop line.

1. Make sure that the attribute **Use as a pedestrian area** is selected in the attributes of the desired links for pedestrians (see "Defining pedestrian links" on page 924).
2. Make sure that the attribute **Use as a pedestrian area** is deactivated in the attributes of the desired links for pedestrians (see "Attributes of links" on page 409).
3. Add the first priority rule (see "Defining priority rules" on page 547).
4. Edit the attributes of the priority rule (see "Attributes of priority rules" on page 549).

   *The **Priority Rule** window opens. The vehicle classes or pedestrian classes are shown in the areas **Stop line** and **Conflict marker** depending on the link attribute **Use as a pedestrian area**.*



5. Select the desired entries.

6. Confirm with **OK**.

10.11 Modeling pedestrian compositions

For pedestrian flows you define the pedestrian compositions. Pedestrian compositions consist of pedestrian types (see "Defining pedestrian compositions" on page 931), (see "Using pedestrian types" on page 876). The definition is comparable to the definition of vehicle compositions (see "Modeling vehicle compositions" on page 452). Assign pedestrian compositions to pedestrian inputs.

Do not define vehicle compositions for pedestrian flows because they are bound to links and follow the traffic flow model by Wiedemann (see "Driving states in the traffic flow model according to Wiedemann" on page 285).

10.11.1 Defining pedestrian compositions

You can define new pedestrian compositions, assign the desired pedestrian types and enter attribute values.

1. Choose **Pedestrian Compositions** from the **Traffic** menu.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Enter the desired attribute values in the right hand list (see "Attributes of pedestrian compositions" on page 931).

   In the next steps, assign the desired pedestrian types to the chosen pedestrian compositions. For each pedestrian type you must add a row to the right hand list and you can define the attribute values.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

4. In the list on right, on the toolbar, click the **Add** button .
5. Select the desired entry.
6. Enter the desired attribute values in the right-hand list (see "Attributes of pedestrian compositions" on page 931).

The data is allocated.

10.11.2 Attributes of pedestrian compositions

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Tip: To open coupled lists for a network object type, from the **Lists** menu, choose > **<Name network object type>**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Unique number of the vehicle composition
Name	Designation of the vehicle composition
RelFlow	Relative volumes: Taken from the right list

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Pedestrian composition relative flows**

Element	Description
PedType	In the list box, select **pedestrian type**
RelFlow	**Relative flow**: Share of this pedestrian type in the composition. The sum of the relative volumes must not necessarily yield 1.0. As in the route shares, the sum of all relative volumes always forms the basis of 100%; on the basis of Vissim the absolute shares are calculated.
DesSpeedDistr	Reference to the **Desired speed distribution** for the pedestrian type

10.12 Modeling area-based walking behavior

In their attributes, you can assign the following objects an area-based walking behavior:

- For elevators: area-based behavior in an elevator car and area-based behavior that sets in when the elevator car of defined elevator groups is exited (see "Attributes of elevator groups" on page 996)
- Areas (see "Attributes of areas" on page 898)
- Ramps & Stairs (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913)

A walking behavior type consists of (see "Defining walking behavior" on page 932):

- desired speed that you assign with the pedestrian compositions
- parameters for the model of pedestrian dynamics, which you assign to the walking behavior and that are assigned to the pedestrian type

An area behavior type consists of (see "Defining area behavior types" on page 934):

- one or more types of walking behavior
- the assigned pedestrian class per walking behavior
- the time interval
- the desired speed distribution

10.12.1 Defining walking behavior

You can define different walking behaviors and assign them to an area behavior type and to the pedestrians of a pedestrian type. Walking behavior is comparable to driving behavior (see "Defining driving behavior parameter sets" on page 282).

To link the walking behavior to a construction element, assign the area behavior type selected for it to the desired walking behavior. You select the area behavior type via the **Behavior type** attribute of the construction element. In area behavior type, you can define a special walking behavior for each pedestrian class (see "Defining area behavior types" on page 934). You can also assign several walking behaviors to one area behavior type.

If no area behavior type is selected for an area, ramp or stairs or if an area behavior type has not been assigned a walking behavior, the walking behavior assigned to the pedestrian type is used.

The following default walking behaviors can be predefined:

Element	Description
Elevator (in cab)	Walking behavior of pedestrians in an elevator cab The **default** walking behavior differs from the **Noise** parameter = 0.3 (walking behavior **default** = 1.2). This means the pedestrians in the elevator cab are not moving much.
Elevator (alighting)	Walking behavior of pedestrians alighting from an elevator car to continue their route in the area in front of the elevator. The following parameters differ from the **default** walking behavior: - **Tau** = 0.2 (walking behavior **default** = 0.4) - **ASocIso** = 2.04 (walking behavior **default** = 2.72) - **BSocIso** = 0.3 (walking behavior **default** = 0.2) These values allow passengers alighting from the elevator cab to easily pass other passengers.

1. From the **Base Data** menu, choose > **Walking Behaviors**.

   *The **Walking Behaviors** list opens. The columns in the list also contain model parameters per pedestrian type according to the social force model (see "Defining model parameters per pedestrian type according to the social force model" on page 868).*

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

2. Right-click in the list.

3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

4. Enter the desired data.

Element	Description
No	Unique number of walking behavior
Name	Name of walking behavior

The following columns contain model parameter values (see "Defining model parameters per pedestrian type according to the social force model" on page 868):	
Tau	Relaxation time or inertia that can be related to a reaction time Default value 0.4 s **Tau** will be set to 0.8 s for pedestrians that wait in front of a red signal head. This allows for realistic passing of approaching pedestrians and calm waiting behavior of pedestrians in stationary state. As soon as the signal head turns green, **Tau** is reset to the default value.
ReactToN	**React to n**: During calculation of the total force for a pedestrian, considers only the influence exerted by the *n* closest pedestrians.
ASocIso	**A social (isotropic)** and **B social (isotropic)** govern the force between pedestrians.
BSocIso	**B social (isotropic)** and **A social (isotropic)** govern the force between pedestrians.
Lambda	**Lambda** governs the degree of anisotropy of the forces.
ASocMean	**A social (mean)** governs the strength *(A)* of the social force between two pedestrians.
BSocMean	**B social (mean)** governs the range *(B)* of the social force between two pedestrians.
VD	Parameter VD in seconds
Noise	The greater this value, the stronger the random force that is added to the systematically calculated forces if a pedestrian remains below his desired speed for a certain time. **Noise** is set to 0 for pedestrians waiting in front of a red signal head. This allows for realistic passing of approaching pedestrians and calm waiting behavior of pedestrians in stationary state. As soon as the signal head turns green, **Noise** is reset to the default value.
Side preference	Specifies whether opposing pedestrian flows prefer using the right or the left side when passing each other.

10.12.2 Defining area behavior types

You can define, edit and delete area behavior types. These allow you to group properties that influence the walking behavior of pedestrian classes on areas, ramps and stairs. Assign the desired time interval, pedestrian class, their desired speed and the walking behavior to an area behavior type via area behavior type elements. You allocate the desired area behavior type in the attributes of the area, ramp or stairs (see "Attributes of areas" on page 898), (see " Attributes of ramps and stairs, moving walkways and escalators" on page 913).

1. From the **Base Data** menu, choose > **Area Behavior Types**.

   *The **Area Behavior Types** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

In the Attributes list, you can define new objects and edit attribute values.

The objects of this object type may have relations to other objects. This is why the attributes list is shown as part of a coupled list (on the left). In the **Relations** list box, to the right of the list on the left, you can show the coupled list with the attributes of the relation of your choice (see "Using coupled lists" on page 119).

2. Right-click in the list.

3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

4. Enter the desired data.

Element	Description
No.	Unique number of the area behavior type
Name	Name of area behavior type

Allocating area behavior type elements via relation

The attribute and attribute values of this base data type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.
2. On the list toolbar, in the **Relations** list, click **Area behavior type elements**.

   The list on the right contains attributes and attribute values of the area behavior type elements.

3. Right-click on the row header in the right-hand list.

4. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

   By default, you can edit the list (see "Using lists" on page 93).

5. Enter the desired data.

Element	Description
AreaBehaviorType	The area behavior type selected in the list on the left
PedClass	Pedestrian classes (see "Using pedestrian classes" on page 879)
TimeInt	Time interval (see "Using time distributions" on page 246)
DesSpeedDistr	Desired speed distribution (see "Using desired speed distributions" on page 237)
WalkBehav	Walking behavior (see "Defining walking behavior" on page 932)

The data is allocated.

10.13 Modeling pedestrian demand and routing of pedestrians

You can model the pedestrian demand by defining the pedestrian inputs for the areas, inserting routing decisions in the walkable areas and defining routes to the destination areas.

As this can be very time-consuming in big networks with many pedestrian inputs and route destinations, you can define pedestrian demand in an origin-destination matrix with less detail, based on flows between origin and destination areas.

You can combine the two approaches in your network model.

10.13.1 Modeling pedestrian inputs

You can define pedestrian inputs for pedestrian areas. Select the attributes to enter the pedestrian inputs and select a pedestrian composition. During the simulation period,Vissim generates single pedestrians for pedestrian inputs at a random point of time based on pedestrian compositions and the volumes entered.

You can create time intervals for pedestrian inputs and thus determine the volume for each time interval in a pedestrian input. You enter the volume for a pedestrian input as pedestrians per hour. Here you can select whether the exact number of pedestrians is entered or Vissim selects the number stochastically based on the volume entered. You must always enter the number of pedestrians per hour, even if the time interval is shorter or is 3.5 hours, for example.

The time when the pedestrian enters an area in the Vissim network is defined by Vissim stochastically: An average time gap between two pedestrians results from the hourly volume. This average time gap is used as an average value of a negative exponential distribution. Vissim obtains the time gaps from this distribution which relates to a Poisson distribution. In real life scenarios, the entry time may be subject to greater variability than in Vissim on the basis of the Poisson distribution.

10.13.1.1 Defining pedestrian inputs

Pedestrian inputs are positioned on areas. The pedestrians in the pedestrian input are not generated at this point, but are generated randomly at different positions in the area. This way, you can use the shape and size of this area to determine whether Vissim should create pedestrians for selected spots or large areas.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, select **Pedestrian Inputs**.
2. Hold down the CTRL key and right-click on the desired position of the pedestrian input in the pedestrian area.

   *The pedestrian input is defined. Per default, the pedestrian area displays a black circle with a blue outline. The pedestrian area is displayed in green. The **Pedestrian Inputs** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152).*

   A new row with default data is inserted.

   *The default value for the **Volume** (**Volume**) is 0. The default value for the **Pedestrian composition** (**PedComp**) is **Pedestrian**. Both default values are valid for all time intervals defined for pedestrian inputs.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

3. Enter the desired attribute values in the left-hand list (see "Attributes of pedestrian inputs" on page 938).

   In the next steps, you assign the desired network objects to the selected pedestrian input.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Area: Area attributes
- Pedestrian volumes by time interval: If you have entered a volume or selected a pedestrian composition and then select **Pedestrian volumes by time interval**, the attributes of the pedestrian volumes for the first time interval are displayed.

4. On the list toolbar, in the **Relations** list, click the desired entry.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

5. Enter the desired attribute values in the right-hand list (see "Attributes of pedestrian inputs" on page 938).
6. If you wish to define more pedestrian volumes for the selected pedestrian input, enter more time intervals in the right-hand list.

The data is allocated.

10.13.1.2 Attributes of pedestrian inputs

The **Pedestrian Inputs** list opens when you insert the network object, if automatic opening of a list after object creation is enabled (see "Right-click behavior and action after creating an object" on page 152).

1. From the **Lists** menu, choose **Pedestrian traffic** > **Inputs**.

   The ***Pedestrian inputs*** *list opens.*

   By default, you can edit the list *(see "Using lists" on page 93)**.*

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Tip: To open coupled lists for a network object type, from the **Lists** menu, choose > **<Name network object type>**.

The list on the left may include the following attributes:

Column	Description
No	Unique number of the pedestrian input
Name	Name of the pedestrian input
Area	Name of the area to which the pedestrian input was added
Volume(0)	**Volume**: number of pedestrians per hour and not per time interval
Pt	Coordinates of the circle of the pedestrian input
ShowLabel	Show label: ☐ If this option is not selected, the label for the pedestrian input is not displayed, even if the label for all pedestrian inputs is selected.

2. If you want to change the time interval for a pedestrian input, right-click the desired entry in the **Pedestrian Inputs** list.
3. In the context menu, select **Edit Time Intervals**.

   The ***Time Intervals*** *list opens* *(see "Defining time intervals for a network object type" on page 326)**.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

Showing and editing dependent objects as relation

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

Pedestrian Volumes By Time Interval list

2. Enter the desired values.

Column	Description
Cont	**Continued**: Adopts the volume of the previous interval, if several time intervals for pedestrian inputs have been defined. The first time interval is therefore always deactivated. - ☐ The option is not selected: The cells are white and are only valid for this interval. - ☑ The option is selected: The cells are gray and are valid for the period of the combined intervals. Only the last cell, for which the **Cont** attribute is not selected, can be edited. When this option is selected, a change in volume is adopted for all the following cells.
TimeInt	**Time interval**: Start and end of the interval in simulation seconds (see "Defining time intervals for a network object type" on page 326).
PedComp	**Pedestrian composition** (see "Defining pedestrian compositions" on page 931)
Volume	**Volume**: number of pedestrians per hour and not per time interval
VolType	**Volume type**: - **Stochastic**: Stochastic variations of the number of pedestrians may occur. The cells are white. - **Exact**: Exactly the specified number of pedestrians are generated and used. The cells are yellow.

3. If you want to allocate additional volumes to other time intervals for the selected pedestrian input, right-click on the **Pedestrian Volumes By Time Interval** list.
4. From the shortcut menu, choose **Add**.

   The next defined time interval will be added.
5. Enter the desired values.

10.13.2 Modeling routing decisions and routes for pedestrians

The definition and editing of pedestrian routing decisions, static pedestrian routes and partial pedestrian routes is mostly similar to that for vehicles (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459).

Static pedestrian routes and partial pedestrian routes are edited in the same way. Therefore, descriptions relating to pedestrian routing decisions and pedestrian routes will always refer to both types, unless explicit distinction is made between static pedestrian routes and partial pedestrian routes in the following sections.

A pedestrian route is a defined sequence of areas and ramps:

- A pedestrian route starts with routing decision in an area. The routing decision is a routing point which is displayed as a red circle by default. The routing decision and its routing point must be located in an area.
- The last routing point of a pedestrian route is referred to as destination. The destination is a routing point which is displayed as a turquoise circle by default. The destination can be located in an **area** or a **ramp** . If a pedestrian has reached his destination area and the area does not contain any additional routing decisions he can use, he is removed from the simulation.
- The other routing points are intermediate destinations. They can be located as intermediate points in an **area** or a **ramp** .

Several pedestrian routes to different destinations can start from the first routing point.

- The destinations of static pedestrian routes which start from one routing point can be located in different areas. A static pedestrian route can also run back to the area from where it started.
- The destinations of partial pedestrian routes which start from one routing point must be located in one area.

If you delete a routing decision, the corresponding routes are deleted.

Note: Start points, intermediate points or the destination of a pedestrian route must not be placed on a link defined as a pedestrian area. (see "Attributes of links" on page 409), (see "Modeling links as pedestrian areas" on page 922). You can specify an **Area** construction element on the link and then position the first routing point, intermediate point or destination of your pedestrian route in this area.

Consideration of a routing decision by pedestrians

A routing decision only applies to pedestrians that have been assigned a pedestrian class and that are without any routing information. A pedestrian already on a route may only accept new routing information after he has reached the destination area of his route. There must be no pedestrian input to this area. Partial routing decisions (static) lead to a different behavior (see "Static pedestrian routes, partial pedestrian routes and pedestrian routing decisions" on page 940).

10.13.2.1 Static pedestrian routes, partial pedestrian routes and pedestrian routing decisions

A pedestrian route starts with routing decision in an area. A pedestrian route may be static or partial. A static pedestrian route starts at a routing decision of the type **Pedestrian routes (static)**. A partial pedestrian route starts at a routing decision of the type **Pedestrian route (partial)**. You select the type in the Network objects toolbar, when inserting the routing decision.

Differences between static pedestrian routes and partial pedestrian routes

- **Pedestrian routes (static)**: Static pedestrian routes lead pedestrians from an area with a pedestrian input and the first routing point of a pedestrian route (a red circle by default) to an area with the destination of the pedestrian route (a turquoise circle by default). Several pedestrian routes may run from the first routing point of a static pedestrian route to different areas. The number of pedestrians (static) is defined by the **Relative volume** attribute. It does not depend on the dynamic status in the simulation.

  In Viswalk, pedestrian inputs, static pedestrian routing decisions and static pedestrian routes define a pedestrian OD matrix that is adhered to. Therefore, pedestrians arrive at the destination of their static pedestrian route and are not influenced by other routing decisions in areas that they pass in the course of their pedestrian route. Only if there is a routing point of a static pedestrian route and a first routing point of one or several partial pedestrian routes in one of these areas, this may have an impact on the pedestrian's remaining route.

  In addition, static pedestrian routing decisions affect areas for which the **Platform edge** attribute is selected.

- **Pedestrian routes (partial)**: Partial pedestrian routes serve the local distribution of pedestrians without changing the pedestrian OD matrix. If several partial pedestrian routes start at the same routing point, their destinations must be located in the same area.

  If the pedestrians fulfills the following conditions, he is assigned a new role:

  - The pedestrian enters an area which has the following properties:
    - a routing point of his original static pedestrian route or partial pedestrian route is located in this area.
    - the first routing point of another partial pedestrian route is located in this area. If the pedestrian is already on a partial pedestrian route, he can choose the other pedestrian partial route.
  - The destination of the new, partial pedestrian route is located in an area which also includes the routing point of his original route.
  - The pedestrian belongs to a pedestrian class which the partial routing decision applies to.

  The pedestrians follow the partial pedestrian route depending on the route choice method selected (see "Defining partial routing decisions of a pedestrian" on page 949). From the destination of the partial pedestrian route, the pedestrians continue to follow their previous, static pedestrian route.

Replacing, creating and adding routing points

- **Replace route points**: Effect of a partial pedestrian route, if its destination is the second next or later routing point of the pedestrian's static route. This also applies for multiple partial pedestrian routes.

- **Inserting routing points**: Effect of a partial pedestrian route, if its destination is the next routing point of a pedestrian's static route. This also applies for multiple partial pedestrian routes.
- **Add routing point at the end of pedestrian route**: Effect of a static pedestrian route. A routing point can only be added when a pedestrian is added to the network or has reached the destination of his former pedestrian route in an area that carries a new routing decision. If his former pedestrian route ends in the area of a new routing decision, this area must not contain a pedestrian input. Otherwise, the routing decision is ignored and the pedestrian is removed from the simulation.

10.13.2.2 Modeling partial routes for pedestrians

A pedestrian routing decision (partial) is either **static**, based on a **formula** or **dynamic**. For dynamic pedestrian routing decisions (partial) you can also select a route choice method.

Static method

Route choice method	Description
Static	Fixed user-defined ratios per partial route for each user-defined time interval (see "Defining partial routing decisions of a pedestrian" on page 949), (see "Attributes of pedestrian partial routing decisions" on page 962). For static partial routes the decision process is the same as for static routes (main routes).

Method used - Formula

Route choice method	Description
Formula	Enter the formula (with the attribute and attribute value), which defines the percentage of pedestrians using the pedestrian route (see "Defining partial routing decisions of a pedestrian" on page 949), (see "Attributes of pedestrian partial routing decisions" on page 962).

Dynamic method

Route choice method	Description
Travel time	Dependence of choice ratios on the travel time of pedestrians who have already finished the partial route (see "Route choice method using the Travel time criterion" on page 952)

Route choice method	Description
Service point selection	For modeling the route choice when confronted with a set of parallel queues, e.g. in front of several counters where the shortest queue should be selected (see "Route selection method Use service point selection" on page 974).
Quantity	Proportion depends on the number of pedestrians in that area (see "Route choice method using the Quantity criterion" on page 953): **Combination method**: Calculation of the number of pedestrians from the route choice areas
Density	Proportion depends on the density of pedestrians in that area (see "Route choice method using the Density criterion" on page 955): **Combination method**: Calculation of the number of pedestrians from the route choice areas

The partial routing decision only affects pedestrians whose current pedestrian route (static or partial) includes an intermediate point or destination in the area that the partial routes, belonging to the respective partial routing decision, have their destination.

A partial routing decision function applies as soon as the pedestrian enters the area where the partial routing decision is made. The area does not have to include an intermediate point of the pedestrian's route.

All partial pedestrian routes starting at the same partial routing decision lead to the same destination area. This does not only apply to partial pedestrian routes whose routing point is located in an area with the **Queuing** attribute. In this case, this routing point is required in order to allow pedestrians to join the queue.

Weighting partial pedestrian routes

You can weight partial pedestrian routes using the attributes below and thus influence the attractiveness of a partial pedestrian route or take into account different capacities of partial pedestrian routes: (see "Attributes of partial pedestrian routes" on page 964):

- **Additional quantity**
- **Additional quantity - service point selection**
- **Additional density**
- **Additional travel time**

10.13.2.3 Use cases for static partial routes of pedestrians

You can for instance use static partial routes for the following use cases:

- Use case 1 distributes the pedestrians with a long static pedestrian route locally, using static partial routes on different paths, but with the same destination. Then the pedestrians continue their route, taking the original, static pedestrian route to their original destination. This way, you do not have to define several similar, static pedestrian routes that differ in just a few route elements.

- Use case 2 benefits from an attribute of the static partial routing decision. This attribute specifies the obligatory inclusion of all pedestrians that enter the area without their intermediate or final destination there. Thanks to this attribute, pedestrians are spatially better distributed if their current position already suggests their next routing variant. In this case, the routing decision often provides one route only. You will find an example of the *Static - Ticket Gates.inpx* file in the directory:

  ..\Examples Training\Pedestrians\Enhanced Routing\Partial Routes

A partial routing decision with the **Decision model** attribute and the **Entering the area** entry may apply to a pedestrian in one of the following cases:

- when entering an area for which a partial routing decision has been specified
- when selecting a new static pedestrian route

There are the following cases in which a partial routing decision is ignored, although you might expect it to be applied:

- Each of the two areas **A** and **B** have a partial routing decision **a** and **b**. For both routing decisions, the **Entering the area** option is selected in the **Decision model** attribute. Area **B** is located completely within area **A**. This means that pedestrians always enter area **A**first before they might enter area **B**. Main and partial routes are defined in such a way that the partial routing decision **b** is valid for pedestrians. However, this does not apply to the partial routing decision **a**. Partial routing decision **a** would, however, become applicable if the main route was replaced by partial route **b**. All the same, partial routing decision **a** is not applied immediately after partial routing decision **b** or when the pedestrian leaves area **B** and enters area **A** .
- In the simulation, one pedestrian is added to the pedestrian input and is assigned a static pedestrian route. At his starting position, however in other areas, there are two additional partial routing decisions **a** and **b**. For both routing decisions, the **Entering the area** option is selected in the **Decision model** attribute. Partial route **a** ends in an area in which the main route includes an intermediate destination, but the partial route **b** does not. Partial route **b**, however, does end in an area in which partial route **a** includes an intermediate destination. Even so, only partial routing decision **a** is carried out. Partial routing decision **b** is ignored, as the condition "check for existing partial routing decisions" requires that a normal routing decision was carried out. If the partial routing decision **b** is not positioned in the area which contains the pedestrian input through which the pedestrian is added, but in an area that the pedestrian will enter in the following time step, the partial routing decision **b** is still carried out.

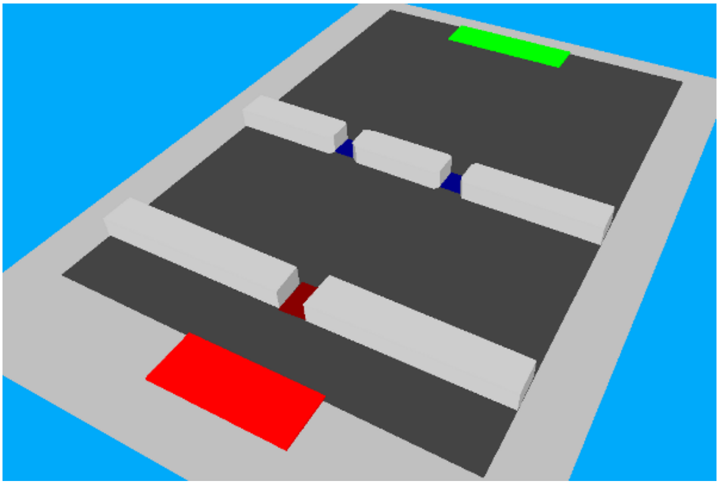
These restrictions prevent endless loops that could occur within a single time step.

Use case 1

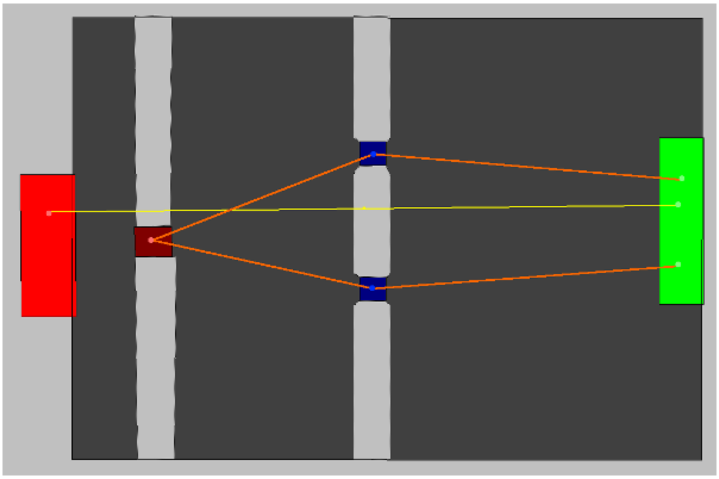
Use case 1 includes pedestrians coming from different origins located relatively far away and following routes with many intermediate points. All these pedestrians will pass through a specific area. This area provides alternative routes, irrespective of the pedestrians' origins or destinations. After having passed through this area, the pedestrians continue their routes, based on their origins, to their distant destinations. On their respective routes, they account for numerous intermediate points.

In this case, it is useful to model the route choice in their common through area with special partial routes. These should only change the route locally, not all the way.

The following two figures show examples of sections of long pedestrian routes, including the section that can be modeled with partial routes. The following figure shows the static pedestrian route through the red area below to the green area above, with the destination of the pedestrian route:



In the following figure, the static pedestrian route is highlighted by a light yellow line that runs horizontally from the red area on the outer left to the destination in the green area on the right. Both partial pedestrian routes are marked by darker, orange lines. They start in the dark red area and run to the destinations in the right area via the areas with the blue intermediate points.



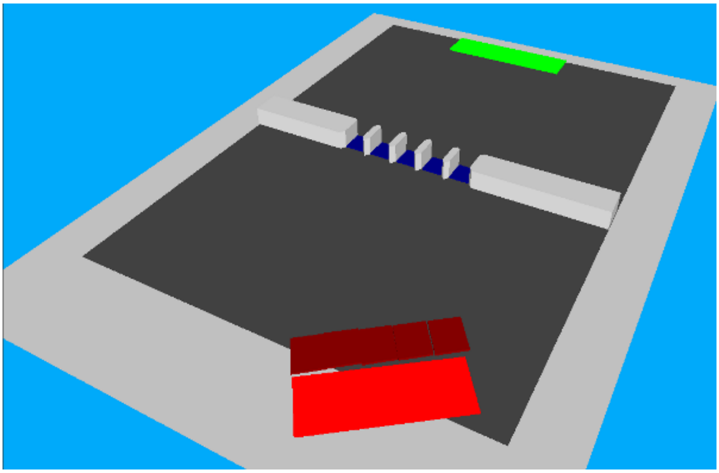
In this example, the static partial route choice function is applied because the destinations of the partial pedestrian routes are located in the same green area to the right as the intermediate point of the static pedestrian route.

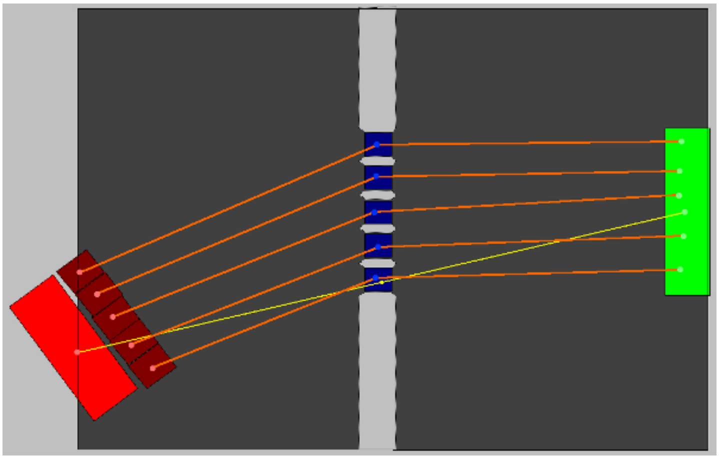
Without static partial routes, each long static route would have to be defined as often as there are spatially limited alternative options in the common area.

Use case 2

The problem with ticket gates is that the simulated pedestrians will mostly choose the shortest path. They will not take a detour to save time - not even if the detour is as quick as in the case of this ticket gate example. If pedestrians do not approach the ticket gates orthogonally, they might cram at one or two of them and ignore the other ticket gates. This would not be realistic behavior. To avoid this, use dynamic partial routing decisions.

As the pedestrian's exact position, which is located a few meters away from the ticket gates, specifies to a certain extent the gate he is likely to use in reality, you can use the so-called “catch all” feature of static partial routing decisions to guide him to a particular gate. This is depicted in the following figures:





In this case, routing decisions are actually not real decisions regarding alternative routes, as each decision is assigned one route only. This again reflects the advantage of the “catch all” feature.

You can also use the **dynamic potential** method to model similar situations in which the shortest or quickest route are relevant (see "Dynamic potential" on page 968).

The dynamic potential method is particularly suited for the following cases:

- when alternative routes show minor differences only, e.g. service desk choice(see "Route selection method Use service point selection" on page 974)
- when alternative routes are not fully discrete, e.g. when a large crowd has to make a 90° turn or a U-turn at a specific point.

10.13.2.4 Defining static pedestrian routes

Add the first routing point as starting point for a pedestrian route (static) to the area of your choice. Define one or several pedestrian routes (static) from this routing point to other areas. Using intermediate points in areas and ramps allows you to model the course of the route realistically.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Pedestrian Routes**.

   The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Pedestrian Routes** again.

   A list box opens.

3. Then click **Static**.
4. Press the CTRL key and in the Network editor, right-click the area of your choice.
5. Release the keys.

   A red circle is added by default. During simulation, the pedestrians are not generated exactly at a circle, but within an area. If for this first routing point you want to insert multiple destinations, carry out the following steps accordingly. By carrying out these steps, you insert one destination.

6. Point the mouse pointer to the destination area of your choice.

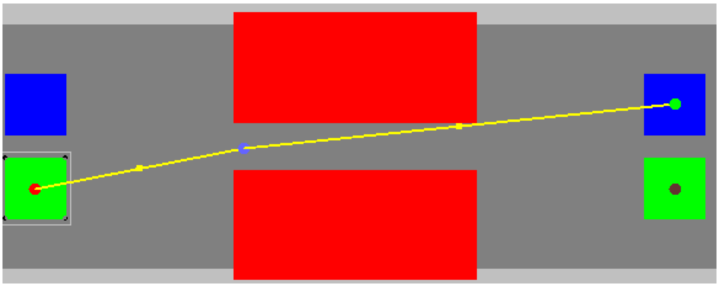
   The border of the destination area is highlighted.

7. Click into the destination area.

   A turquoise dot is inserted by default. The pedestrian area is displayed in red. A yellow band is shown between the first routing point of the pedestrian route and the destination by default.

8. Release the keys.

   *The **Pedestrian routing decision (static)** list opens, if automatic opening of a list after object creation is selected (see "Right-click behavior and action after creating an object" on page 152). You can add additional destinations to areas or stop the addition.*



9. To insert additional destinations, point the mouse pointer to the desired area of your next destination and carry out the steps again.

   When you point the mouse pointer to a destination, a yellow line will appear by default between the first routing point of the pedestrian route and the destination.

10. If you do not want to add any additional destinations, in the Network editor, click in an empty area.

    Especially when defining several pedestrian routes from the first routing point of a pedestrian route, use intermediate points in areas and on ramps to model the course of a route (see "Modeling the course of pedestrian routes using intermediate points" on page 957).

11. You can edit the attributes of a pedestrian route (see "Attributes of static routing decisions for pedestrian routes" on page 960), (see "Attributes of static pedestrian routes" on page 961).

    The attributes are saved to the respective list.

10.13.2.5 Defining partial routing decisions of a pedestrian

Add the first routing point as starting point for a partial pedestrian route to the area of your choice. You may select attributes, such as the decision model, the pedestrian class and the route choice method. From this routing point, define one or several partial pedestrian routes which run through other areas or over other ramps. Using intermediate points in areas and on ramps allows you to model the course of the route realistically. If several partial pedestrian routes start at the same routing point, their destinations must be located in the same area.

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

---

1. On the Network objects toolbar, click **Pedestrian Routes**.

The button is highlighted. The variant of the network object type is displayed in parentheses and the button is shown.

2. Click **Pedestrian Routes** again.

   A list box opens.

3. Then click **Partial**.
4. Press the CTRL key and in the Network editor, right-click the area of your choice.
5. Release the keys.

   For the first routing point, a red circle is inserted by default. If you want to insert multiple pedestrian routes for this first routing point, carry out the following steps accordingly. This allows you to add a destination and pedestrian partial route.

   *The window **Pedestrian Routing Decision (Partial)** opens.*

6. Make the desired changes:

Element	Description
Decision	Number of partial routing decision
Start area	Name of area for which the partial routing system has been defined. In the list box, you can select another area as your start area.
Destination area	is empty before you insert the partial routing decision. Afterwards, it shows the name of the destination area.
Destination ramp	is empty before you insert the partial routing decision. Afterwards, it shows the name of the destination ramp.
Decision models	Select option: ▸ **Entering the area**: This partial routing decision function applies as soon as the pedestrian enters the area. Select this option when decision-relevant criteria are likely to change rather quickly, e.g. when you select the **Static** route choice method which has the same number of route choices for the pedestrian routes. ▸ **Every time step**: The partial routing decision function applies in each time step when the pedestrian is in the area. Select this option when there are only minor or no changes to the decision-relevant criteria, e.g. when you select the **Density** route choice method in order to find an area with the lowest density.

Element	Description
Pedestrian classes	Select option: - All pedestrian types of pedestrian class - individual pedestrian classes
Route choice method	Select the static or dynamic route choice method (see "Modeling partial routes for pedestrians" on page 942): - **Static** (see "Defining static pedestrian routes" on page 948) - **Travel time** (see "Route choice method using the Travel time criterion" on page 952) - **Service point selection** (see "Route selection method Use service point selection" on page 974) - **Quantity** (see "Route choice method using the Quantity criterion" on page 953) - **Density** (see "Route choice method using the Density criterion" on page 955) - **Formula**: Define a formula (with the attribute and attribute value), which defines the percentage of pedestrians using this pedestrian route (see "Attributes of partial pedestrian routes" on page 964).

7. Point the mouse pointer to the destination area of your choice.

   The border of the destination area is highlighted.

8. Press the CTRL key and right-click in the destination area.
9. Release the keys.

   *By default, a turquoise circle is inserted. By default, a yellow band is shown between red circle of the first routing point of the partial pedestrian route and the destination. The **Partial Pedestrian Routing Decisions** list opens if automatic opening of a list after object generation is selected (see "Right-click behavior and action after creating an object" on page 152). You may add further partial pedestrian routes to areas on the basis of the routing point or stop the addition.*

10. To insert additional partial pedestrian routes, point the mouse pointer to the desired area of your next destination and carry out the steps again.

    When you point the mouse pointer to an area, a yellow line will appear by default to the first routing point of the partial pedestrian route.

11. If you do not want to add any partial pedestrian routes, in the Network editor, click in an empty area.

    Especially when defining several pedestrian routes from the first routing point of a pedestrian route, use intermediate points in areas and on ramps to model the course of a route (see "Modeling the course of pedestrian routes using intermediate points" on page 957).

12. You can edit the attributes (see "Attributes of partial pedestrian routes" on page 964), (see "Attributes of pedestrian partial routing decisions" on page 962).

The attributes are saved to the respective list.

13. Define route choice areas for the route choice methods **Quantity** and **Density** or generate them automatically (see "Selecting route choice areas" on page 956).

Route choice method using the Travel time criterion

The following steps are performed:

1. The pedestrians are equally distributed across all pedestrian routes of the routing decision.
2. Vissim evaluates the travel times of the pedestrians who have finished this pedestrian route. You can enter the number of these pedestrians in the field **Average of the last <number> pedestrians**. The default value is 10. If the number of pedestrians is < 10, an even distribution is taken as a basis.
3. *Travel time of route i = Ti* is the average travel times of the last user-defined number of pedestrians who have finished a route. It can also be the average of travel times of pedestrians who have so far arrived at the route destination.

You can select the following **Route choice parameters**:

Option	Meaning
Best Route	The user-defined percentage of pedestrians uses the best route. That is the route with the least travel time. If there are two best routes, the volumes are distributed equally on both routes. The remaining pedestrians are distributed randomly across the other routes. Default value of 90.00%.
Kirchhoff	Kirchhoff exponent: The probability of a route choice is calculated as the reciprocal of the travel time RZ to the power of Kirchhoff exponent E) divided by the sum of these powers for all routes, default value 3.5. For two partial routes, the relative volume p_i for partial route 1 is: $$p_1 = \frac{\left(\frac{1}{Rz_1}\right)^E}{\left(\frac{1}{Rz_1}\right)^E + \left(\frac{1}{Rz_2}\right)^E}$$

Option	Meaning
Logit	Logit - denominator: The probability of a route choice is calculated as exponent **e** to the power of (the negative travel time-RZ divided by the user-defined denominator c), divided by the sum of these powers for all routes, default value 10 seconds: $$p_1 = \frac{e^{\frac{-Rz_1}{c}}}{e^{\frac{-Rz_1}{c}} + e^{\frac{-Rz_2}{c}}}$$ $$p_2 = \frac{e^{\frac{-Rz_2}{c}}}{e^{\frac{-Rz_1}{c}} + e^{\frac{-Rz_2}{c}}}$$
Logit of reciprocal	Logit of reciprocal - numerator z: the probability of a route choice is calculated as exponent **e**, to the power of (the numerator z divided by the travel timeRZ), divided by the sum of these powers for all routes, default value 10 seconds: $$p_1 = \frac{e^{\frac{z}{Rz_1}}}{e^{\frac{z}{Rz_1}} + e^{\frac{z}{Rz_2}}}$$

Route choice method using the Quantity criterion

Vissim calculates the number of pedestrians in the areas of the pedestrian partial routing decisions. Then Vissim calculates the number of pedestrians in areas with the pedestrians' relevant partial routing decisions.

You can select the following **Route choice parameters**:

Option	Meaning
Best Route	The user-defined percentage of pedestrians uses the best route. It follows the pedestrian route whose areas have the lowest number of pedestrians. These areas include the pedestrians' relevant partial routing decisions. If there are multiple best routes, the volumes are distributed equally across them. The remaining pedestrians are distributed randomly across the other routes. Default value of 90.00 %.
Kirchhoff	Kirchhoff exponent E: The probability p_i of a route choice is calculated as the quotient reciprocal of the number N of the relative volumes, to the power of negative Kirchhoff exponent -E, divided by the sum of these powers for all routes, default value 3.5. For two partial routes, the relative volume p_i for partial route 1 is: $p_1 = \frac{N_1^{-E}}{N_1^{E}+N_2^{E}}$
Logit	Logit - denominator c: The probability of a route choice is calculated as exponent **e** to the power of (the negative quantity -N divided by the user-defined denominator) c, divided by the sum of these powers for all routes, default value 1.00: $p_1 = \frac{e^{\frac{-N_1}{c}}}{e^{\frac{-N_1}{c}}+e^{\frac{-N_2}{c}}}$ $p_2 = \frac{e^{\frac{-N_2}{c}}}{e^{\frac{-N_1}{c}}+e^{\frac{-N_2}{c}}}$
Logit of reciprocal	Logit of reciprocal - numerator z: the probability of a route choice is calculated as exponent **e**, to the power of (the numerator z divided by the quantity$N$), divided by the sum of these powers for all routes, default value 1.00: $p_1 = \frac{e^{\frac{z}{N_1}}}{e^{\frac{z}{N_1}}+e^{\frac{z}{N_2}}}$

Combination method: Calculation of the number of pedestrians from the route choice areas

- **Total**: Total number from all route choice areas. When route choice areas overlap, these areas and the pedestrians there are each taken into account only simply.
- **Average**: Average number from all route choice areas
- **Maximum**: Maximum number from all route choice areas
- **Minimum**: Minimum number from all route choice areas

Applies to **average**, **maximum**and **minimum**: When route choice areas overlap, these areas and the pedestrians there are taken into account several times, and so the number of pedestrians for each area is determined separately.

Route choice method using the Density criterion

Vissim calculates the number of pedestrians in the areas of the pedestrian partial routing decisions. Then Vissim calculates the volume of pedestrians in areas with the pedestrians' relevant partial routing decisions on the basis of the route choice parameters.

You can select the following **Route choice parameters**:

Option	Meaning
Best route	The user-defined percentage of pedestrians uses the best route. It traverses the area with the lowest pedestrian density. If there are multiple best routes, the volumes are distributed equally across them. The remaining pedestrians are distributed randomly across the other routes. Default value of 90.00%.
Kirchhoff	Kirchhoff exponent E: The probability of a route choice is calculated as the reciprocal of the density, ϱ to the power of negative Kirchhoff exponent $-E$ divided by the sum of these powers for all routes, default value 3.5. For two partial routes, the relative volume p_i for partial route 1 is: $p_1 = \frac{\varrho_1^{-E}}{\varrho_1^E + \varrho_2^E}$
Logit	Logit - denominator n: The probability of a route choice is calculated as exponent**e** to the power of (negative density $-\varrho$ divided by the user-defined denominator$n$) divided by the sum of these powers for all routes, default value 1.33 pedestrian/m^2: $p_1 = \frac{e^{\frac{-\varrho_1}{n}}}{e^{\frac{-\varrho_1}{n}} + e^{\frac{-\varrho_2}{n}}}$ $p_2 = \frac{e^{\frac{-\varrho_2}{n}}}{e^{\frac{-\varrho_1}{n}} + e^{\frac{-\varrho_2}{n}}}$
Logit of reciprocal	Logit of reciprocal - numerator z: the probability of a route choice is calculated as exponent : **e** to the power of (the numerator$z$ divided by the density ϱ) divided by the sum of these powers for all routes, default value 1.33 pedestrian/m^2: $p_1 = \frac{e^{\frac{z}{\varrho_1}}}{e^{\frac{z}{\varrho_1}} + e^{\frac{z}{\varrho_2}}}$

Combination method: Calculation of the number of pedestrians from the route choice areas

- **Total**: Total density from all route choice areas. When route choice areas overlap, these areas and the pedestrians there are each taken into account only simply.
- **Average**: Average density from all route choice areas
- **Maximum**: Maximum density from all route choice areas
- **Minimum**: Minimum density from all route choice areas

Applies to **average**, **maximum** and **minimum**: When route choice areas overlap, these areas and the pedestrians there are taken into account several times, and so the density of pedestrians for each area is determined separately.

Selecting route choice areas

Through route choice areas Vissim collects the density and number of pedestrians as the basis for the calculation of relative volumes when selecting a route.

You may take into account all areas or ramps in your Vissim network as route choice areas which include intermediate points of the pedestrian route. Or you may select individual areas or ramps to be used as route choice areas.

When you change the course of a partial route and therefore position intermediate points in other areas, repeat the steps.

Select areas with intermediate points as route choice areas

1. Under the **Lists** menu, choose **Pedestrian traffic** > **Partial routes**.
2. Right-click the entry of your choice.
3. In the context menu, select the entry **Automatically generate route choice areas**.

   All areas where the intermediate points of the pedestrian partial route are situated are used as route choice areas. You can select the combination method through which the number or density of the pedestrians on the route choice areas is calculated (see "Modeling partial routes for pedestrians" on page 942).

Tips:

- Alternatively, you can also right-click the desired partial route in the network editor and choose the entry **Generate route choice areas automatically** in the context menu.
- You can also select multiple partial routes and choose the entry **Generate route choice areas automatically** in the context menu.

Select an individual area as route choice area

1. Under the **Lists** menu, choose **Pedestrian traffic** > **Partial routes**.
2. Right-click the entry of your choice.
3. Select **Relations** > **Pedestrian route choice areas**.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

4. Repeat the next steps until you have selected all desired areas or ramps in the list.
5. From the shortcut menu, choose **Add**.
6. In the list on the right, right-click a free area.

   A new row with default data is inserted.
7. Select the desired area or ramp.

10.13.2.6 Modeling the course of pedestrian routes using intermediate points

After you insert intermediate points, a pedestrian route, by default, has the following objects and is displayed as a line:

- a routing decision in the start area
- the destination pedestrian route location in the area where the pedestrian route ends
- The route course is shown as a yellow line.
- The yellow line contains a yellow dot in the middle.

You can insert an intermediate point of the route into each area you want the pedestrian route to traverse. You can move a pedestrian route location to a different area. Examples of use:

- You want the pedestrian route to traverse certain areas. You e.g. want to add additional destinations for the pedestrian route or partial pedestrian routes.
- You want to edit attributes that have an impact on the pedestrian route of an area, e.g. **Cell size**, **Obstacle distance**, **Potential calculation method**, or attributes for dynamic potential.
- You want to assign the pedestrian route location to another area or ramp

In the following cases, a route location on a ramp or stairway defines the direction that the pedestrian takes on the ramp or stairway. In these cases, the pedestrian uses the entire ramp or stairway:

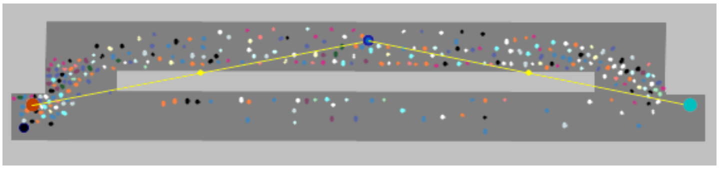
- The previous pedestrian route location of the pedestrian route or the pedestrian’s routing decision must be on the same level of the ramp or stairway. This is the start or end level of the ramp or stairway.
- The previous pedestrian route location of the pedestrian route must no be part of an area for which the **Waiting area for elevators** attribute is selected. Near these waiting areas, the pedestrian always uses the ramp or stairway towards the start - end level.

You can edit the attributes for each pedestrian route location (intermediate point or destination) (see "Attributes of pedestrian route locations" on page 958).

1. On the Network objects toolbar, click **Pedestrian Routes**.
2. In the Network editor, click the destination of the desired pedestrian route.

   A yellow band is shown between the first routing point of the pedestrian route and the destination, including an intermediate point in the middle.

3. Click the intermediate point, hold the mouse button down, and drag the point to the ramp, stairway or area of your choice.
4. Release the mouse button.



The course of the pedestrian route is adjusted. The intermediate point is larger and displayed as a blue circle by default (in the figure in the middle of the upper area). Additional yellow points are displayed (in the figure on the left and right lower margin of the upper area). Using these points, you can model the pedestrian routes in more detail.

Note: Within a pedestrian area, there must not be two adjacent intermediate points of a static pedestrian route or a partial pedestrian route.

10.13.2.7 Attributes of pedestrian route locations

You can edit attributes of pedestrian route locations:

- Destination: A pedestrian route location in a destination area, by default displayed as a turquoise circle. If you are using dynamic potential, it is displayed as a turquoise square.
- Route location: A pedestrian route location in an area the pedestrian route traverses, by default displayed as a blue circle. If you are using dynamic potential, it is displayed as a blue square.

Editing attributes of the destination

The turquoise dot or turquoise square, representing the destination of a pedestrian route, is also shown when the pedestrian route is not selected.

1. In the Network editor, double-click the turquoise point of the pedestrian route.

   *The **Pedestrian Route Location** window opens.*

2. Edit the entries of your choice, see table below.

Editing the attributes of a pedestrian route location

The blue intermediate points of a pedestrian route are displayed when the pedestrian route is selected.

1. On the Network objects toolbar, click **Pedestrian Routes**.
2. In the desired pedestrian route, click the turquoise point of the destination.

   The pedestrian route is displayed as a yellow line with blue route locations.

3. Double-click the desired route location.

   *The **Pedestrian Route Location** window opens.*

4. Make the desired changes:

Column	Description
Decision	Number of routing decision of pedestrian route
Route	Number of routing decision and number of pedestrian route
Area	Area in which the routing point is located In the list box, in the areas list, you can click another route location the pedestrian route runs through.
Ramp	Ramp that lies on the routing point. In the list box, in the ramps list, you can click another ramp for the route location the pedestrian route runs through.
Cell size	Edge length of a grid mesh used for the calculation of distances to the destination area with the static or dynamic potential (see "Defining global model parameters" on page 871). Default value 0.15 m. Avoid values **>** object radius.
Obstacle distance	Distance up to which the nearby walls have a bearing on the distance potential (see "Defining global model parameters" on page 871). Default 0.5 m.
Dynamic potential	**UseDynPot**: Routing takes place along the path with the lowest estimated travel time that pedestrians can take within a single level. Enables the input boxes for the parameters of dynamic potential (see "Dynamic potential" on page 968), (see "Defining the Dynamic Potential for a static pedestrian route" on page 971).

5. Confirm with **OK**.

Tips:
Alternatively, open the **Pedestrian Route Location** window for a route location that is an intermediate point or destination. To do so, in the Network editor, right-click the route location of your choice. Then from the shortcut menu, choose **Edit**.

Editing additional attributes in the attribute list Route locations

In the attribute list **Route locations**, you can edit additional attributes of route locations.

1. From the **Lists** menu, choose **Pedestrian Traffic** > **Static Routes**.

   The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. In the **Static Pedestrian Routes** list, on the toolbar, in the **Relations** list box, click **Route locations**.

   *The **Route locations** list is displayed on the right. By default, you can edit the list (see "Using lists" on page 93).*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

In the **Route locations** list, in addition to the attributes described above, you can show and edit the following attributes:

Long name	Short name	Description
Dynamic potential - calculation interval	**CalcInt**	(see "Defining the Dynamic Potential for a static pedestrian route" on page 971)
Dynamic potential - g (general strength)	**g**	
Dynamic potential - h (direction impact)	**h**	
Dynamic potential - impact	**Impact**	
Dynamic potential	**UseDynPot**	
Pedestrian routes (static)	**PedRoutSta**	Number of routing decision and number of the static pedestrian route
Pedestrian routes (partial)	**PedRoutPart**	Number of the pedestrian routing decision (partial) and number of the partial pedestrian route
Ban elevator use	**BanElevUse**	If a route location of a pedestrian route or partial pedestrian route lies on a ramp or stairs and you selected the attribute **Ban elevator use** for it, the pedestrian will not use the elevator until he has reached the next route location for which this attribute has not been selected.
Point	**Pt**	x and y coordinates of route location
WKT point	**WKTPoint**	x and y coordinates of the route location in the *Well-known text* format
x	**x**	x coordinate of route location
y	**y**	y coordinate of route location

10.13.2.8 Attributes of static routing decisions for pedestrian routes

You may edit the attributes of different variants.

1. From the **Lists** menu, choose **Pedestrian Traffic** > **Static Routing Decisions**.

   The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number
Name	Designation
Area	Name of the area for which a routing decision has been defined.
PedClasses	Pedestrian classes
RouteChoiceMeth	Route selection method for the static pedestrian route: - **Static** (default value): The route is selected on the basis of the **Relative volume** attribute of the pedestrian route. - **Formula:**The route selection is based on a user-defined formula. The formula helps you calculate the percentage of pedestrians using the pedestrian route based on attributes and attribute values.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Pedestrian Classes (see "Attributes of pedestrian classes" on page 879)
- Pedestrian routes (static)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.13.2.9 Attributes of static pedestrian routes

You may edit the attributes of different variants.

1. From the **Lists** menu, choose **Pedestrian Traffic** > **Static Routes**.

   The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number
Name	Designation
RelFlow (0)	Relative volume. Only enabled, if the route choice method **Static** listed in **Static Pedestrian Routing Decisions** is selected.
PedRoutDecSta	Pedestrian Routing Decisions (Static)
Formula	Enter the formula (with the attribute and attribute value), which defines the percentage of pedestrians using this pedestrian route. Only enabled, if the route choice method **Formula** listed in **Static Pedestrian Routing Decisions** is selected. This attribute is not dependent on time intervals.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Route locations

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.13.2.10 Attributes of pedestrian partial routing decisions

You may edit the attributes of different variants.

1. From the **Lists** menu, choose **Pedestrian Traffic** > **Partial Routing Decisions**.

   The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number
Name	Designation
Area	Name of area for which the partial routing decision has been defined.
AllPedTypes	Select this option if you want the partial routing decision to be valid for all pedestrian types of the pedestrian class.
NumPedsForAvg	**Number of pedestrians (for calculation of average)**
NumPedsInQueue	**Number of pedestrians in queue**
PedClasses	Pedestrian classes
BestRoutPerc	Best route percentage for dynamic route choice with the criterion **Travel time** (see "Route choice method using the Travel time criterion" on page 952)
DecModel	**Decision model**: Defines when the route choice affects pedestrians of the pedestrian routing decision area. - **Area walked on**: The route choice affects passengers only in the time step during which the passenger walks in the partial routing decision area. - **Each time step**: The route choice affects passengers from the time step in which they step into the partial routing decision area until the time step when they leave the area.
KirchExp	Kirchhoff exponent for dynamic route choice with the criterion **Travel time**
LogitDenom	Logit - Denominator for dynamic route choice with the criterion **Count**, **Density** or **Travel time**
LogitRecipNum	Logit of reciprocal - numerator for dynamic route choice with the criterion **Count**, **Density** or **Travel time**
RouteChoiceMeth	Route choice method for partial pedestrian routes (see "Defining partial routing decisions of a pedestrian" on page 949)
RoutChoiceAreaCombMeth	Combination method for the route choice methods with the criteria of **Count** and **Density** (see "Route choice method using the Quantity criterion" on page 953), (see "Route choice method using the Density criterion" on page 955)

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Pedestrian Classes (see "Attributes of pedestrian classes" on page 879)
- Pedestrian routes (partial)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.13.2.11Attributes of partial pedestrian routes

You may edit the attributes of different variants.

1. Under the **Lists** menu, choose **Pedestrian traffic** > **Partial routes**.

   The list of attributes opens.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the left may include the following attributes:

Column	Description
No	Number
Name	Designation
RelFlow (0)	Relative volume. Only enabled, if the route choice method **Static** listed in **Static vehicle routing decisions** is selected.
PedRoutDecPart	Partial pedestrian routing decision on the partial route (see "Attributes of pedestrian partial routing decisions" on page 962)
Formula	Enter the formula (with the attribute and attribute value), which defines the percentage of pedestrians using this pedestrian route. Only enabled, if the route choice method **Formula** listed in **Partial Pedestrian Routing Decisions** is selected. The **Formula** attribute is not dependent on time intervals.
You can use the attributes below to weight the partial pedestrian route for the selected route choice method. No value is defined by default, so there is no weighting. Value range > 0.	

Column	Description
AddQuant	**Additional quantity**: Number of pedestrians that is included in the selected combination method for the pedestrian routing decision (partial route). Has the same effect as a constant number of pedestrians in an additional area.
AddQuantSrvPntSel	**Additional quantity - service point selection**: Constant number of pedestrians, which is added to the number of people waiting at the service point. This makes the queue longer and the partial pedestrian route to the service point less attractive.
AddDens	**Additional density**: Density of pedestrians, which is included in the selected combination method for the pedestrian routing decision (partial route). Has the same effect as a constant density of pedestrians in an additional area
AddTravTm	**Additional travel time**: constant proportion of time that is added to the travel time determined dynamically. This results in an extended travel time, making the partial pedestrian route less attractive.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Pedestrian route choice areas
- Route locations

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

The data is allocated.

10.13.3 Using pedestrian attribute decisions

You can use a pedestrian attribute decision to set a pedestrian attribute to a desired value as soon as the pedestrian enters one of the following network objects:

- Area
- Ramp
- Stairway
- Escalator

- Moving walkway
- Elevator

The attribute value can also be based on a distribution defined in Vissim. You can transfer the set attribute value to a pedestrian route. This allows you to select a route based on the attribute value (see "Modeling routing decisions and routes for pedestrians" on page 939). You can restrict the attribute value to the required pedestrian classes.

10.13.3.1 Defining Pedestrian Attribute Decisions

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

You can define the pedestrian attribute decision and allocate it to one of the following network objects:

- Area
- Ramp
- Stairway
- Escalator
- Moving walkway
- Elevator

1. On the Network objects toolbar, click **Pedestrian Attribute Decisions**.
2. Hold down the CTRL key and right-click the desired position of the pedestrian attribute decision in the network object that you want to allocate the pedestrian attribute decision to.

   *A gray circle with a blue contour is inserted. The **Pedestrian Attribute Decisions** list opens.*
3. Edit the attributes:

Short name	Long name	Description
No.	**Number**	Unique number of the pedestrian attribute decision
Name	**Name**	Name of the pedestrian attribute decision

AllPedTypes	**All pedestrian types**	If this option is selected, all pedestrian types will follow the pedestrian attribute decision. The option **All Pedestrian Types** is a virtual pedestrian class that automatically includes all new pedestrian types. This also includes pedestrian types that have not yet been allocated to a pedestrian class.
PedClasses	**Pedestrian classes**	Pedestrian classes to which the pedestrian attribute decision applies
Attr	**Attribute**	Select the pedestrian attribute for which a value is to be set or taken from a distribution defined in Vissim. The **Attribute** is not dependent on the time interval or pedestrian class. The **Attribute** can be a user-defined attribute (see "Using user-defined attributes" on page 210).
DecType	**Decision type**	**Value**: Enables the **Value** box, disables the **Distribution** (**Distr**) box. **Distribution**: Enables the **Distribution** (**Distr**) box, disables the **Value** box.
Value	**Value**	Value to which the **attribute** is set. Only enabled if the **Value** is set in the**Decision type** attribute.
Distr	**Distribution**	Defined distribution in Vissim to which the **Attribute** is set. Only enabled if the **Distribution** is set in the**Decision type** attribute.
TimeFrom	**Time from**	Start of the time interval during which the pedestrian attribute decision is valid (in simulation seconds)
TimeTo	**Time to**	End of the time interval during which the pedestrian attribute decision is valid (in simulation seconds)
Area	**Area**	Number of the area that the pedestrian attribute decision is allocated to
Elevator	**Elevator**	Number of the elevator that the pedestrian attribute decision is allocated to
Ramp	**Ramp/Stairs**	Number of the ramp or stairway that the pedestrian attribute decision is allocated to
x	**x**	x-offset to the center of the rectangle surrounding the area, ramp, stairway or elevator
y	**y**	y-offset to the center of the rectangle surrounding the area

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- **Pedestrian classes**

2. On the list toolbar, in the **Relations** list, click the desired entry.

   The right-hand list is shown. If there is no assignment, only the column titles are shown.

3. Enter the desired attribute values in the right-hand list (see "Attributes of pedestrian classes" on page 879).

The data is allocated.

10.13.4 Dynamic potential

Dynamic potential is a route-based method used to control the search for the best route that pedestrians can take within a level. Thereby pedestrians are not meant to take the shortest distance path, but the path with the least estimated travel time to their destination or intermediate destination. Once you have the selected the Dynamic Potential for a ramp or stairway, this potential will have an impact on the construction element and supports lane formation in pedestrian flows. If a route location on a pedestrian route (with the selected dynamic potential) is on a ramp or stairway, it will affect pedestrians moving towards a ramp or stairway. The route to the construction element will not be affected as pedestrians using a stairway or ramp will automatically be navigated by the static potential. If pedestrians using the ramp or stairway are not to be navigated by the static potential but by the dynamic potential, you can select this option in the attribute list of the ramp or stairway and define the parameters cell size, obstacle distance, impact h direction impact, g general strength and the calculation interval for the ramp or stairway.

The dynamic potential method is spatially continuous and complements the pedestrians' dynamic partial route, which takes the number, volume or travel time of pedestrians into account.

For both the dynamic potential and partial route method, the travel time reduction is the determining factor for walking behavior. Travel time based partial routes that are based on the dynamic potential method, however, provide pedestrians with a discrete choice of different routes at a certain time. If the dynamic potential is active for a destination or an intermediate destination, pedestrians will try to take the route they currently believe is the quickest. This means pedestrians want to move in a direction that according to a heuristic mathematical method is considered the shortest walking time to the next destination or intermediate destination.

Even this rather simplified description of the dynamic potential methods shows its continuous character. There is no specific decision point. Pedestrians continuously aim at optimizing their travel time. This is limited by the simulation time step only. Pedestrians do not try to take the path with the shortest travel time out of a limited number of user-defined routes. With the dynamic potential method, pedestrians choose their trajectory automatically, and thus their route from a continuously unlimited and uncountable number of possible trajectories.

Calculation of the dynamic route potential is very computation time consuming. The dynamic potential field for a specific route is only calculated as long as there are pedestrians that actually use the route.

10.13.4.1 Dynamic potential use cases

The dynamic potential is limited to the level, ramp or stairway you have specified it for. The dynamic potential method cannot be applied to multiple levels and is not meant to find the fastest route across multiple levels. This is where you use partial routes.

Using dynamic potential or travel time based partial routes for a level

There are many use cases where you can apply both methods. Experience has shown that in some cases partial routes may be the better method, whereas in other cases dynamic potential should be preferred. There are use cases for which both methods lead equally to the results desired and others in which neither do.

- When modeling scenarios, partial routes can be used for discrete choices. Partial routes are discrete, whereas the dynamic potential is continuous in several respects.
- When modeling continuous choices, you should opt for the dynamic potential method.
- The dynamic potential supports lane formation in pedestrian flows, in particular on ramps.

Example:

You are modeling a large number of pedestrians at, for example, a 90° corner, using the dynamic potential method. If there are also several service points in the corner, the selected quantity becomes discrete. For these routes, you can use partial routes because the distance from the starting point to the individual service points is different.

Modeling effort and computation time

In general, there is less modeling effort required for the dynamic potential method than for partial routes. Even so, you should avoid modeling use cases with the dynamic potential method that are better suited for the partial routes method. In spite of the higher modeling effort required for partial routes, by default the simulation speed is faster than when you apply the dynamic potential method.

10.13.4.2 Description of the method Dynamic Potential

An important basis for the dynamic potential is the method in which pedestrians generally find their next destination area in a simulation. This is achieved because the driving force of the social force model points towards the next destination as long as the pedestrian does not walk in this direction:

$$\vec{F}_{\alpha}^{0}\left(\vec{v}_{\alpha}, v_{\alpha}^{0}\vec{e}_{\alpha}\right) := \frac{1}{\tau_{\alpha}}\left(v_{\alpha}^{0}\vec{e}_{\alpha} - \vec{v}_{\alpha}\right)$$

Where:

$\vec{v}_{\alpha}$: pedestrians current velocity

v_{α}^{0}: desired speed of the pedestrian based on the user-defined distribution

$\vec{e}_{\alpha}$: Direction of the desired speed (unit vector): from which the multiplication of the desired walking speed results in the current desired walking speed. $\vec{e}_{\alpha}$ is obtained from the Static Potential or Dynamic Potential or a linear combination of both values.

Using the Static Potential $\vec{e}_{\alpha}$ in Vissim always points in the direction of the path with the shortest distance.

Using the Dynamic Potential $\vec{e}_{\alpha}$ points in the direction of the path with shortest time according to the current estimation. This may not be the absolutely right direction of the path with the shortest time in terms of analytics. Due to the fact that in reality pedestrians are often confused regarding which direction at the current time can lead them to their destination the quickest, it is not a big problem that the actual direction of the quickest path cannot be precisely calculated. Therefore, the hypothetical assumption that the direction of the path with the shortest time is known in the simulation and hence the behavior of each pedestrian is optimal, is probably not realistic.

Impact parameter (see "Defining the Dynamic Potential for a static pedestrian route" on page 971), (see "Dynamic potential attributes" on page 972): The value $\vec{e}_{\alpha}{}^{s}$ for the direction of the path with the shortest distance and the value $\vec{e}_{\alpha}{}^{q}$ for the direction of the path with the shortest time. The value $\vec{e}_{\alpha}$ is derived therefrom. $\vec{e}_{\alpha}{}^{s}$ and $\vec{e}_{\alpha}{}^{q}$ are included in accordance with the value of the **impact** parameter as a weighting.

Tip: You will find further information in the document Quickest Paths in Simulations of Pedestrians, **Kretz T.**, **Große A.** u.a., Karlsruhe, 2011

Regardless of whether $\vec{e}_{\alpha}$ points in the direction of the path with the shortest distance (Static Potential is used, the Dynamic Potential is not used) or in the direction of the path with the shortest time (the Dynamic Potential is used at 100 %), when calculating $\vec{e}_{\alpha}$ the first step is to determine the values for the points of the grid which indicate either the distance or the estimated remaining travel time from the respective point to the relevant distance area. The grid is consistent with the potential. Since the distance from a grid point to the destination does not change during the simulation run, the potential that provides the distance values, referred to as static potential, acts as a "Look-up" table. In contrast, with the consideration of all pedestrians in the network at the same time, the estimated remaining travel time to the destination changes continuously for each grid point. Thus this potential is referred to as dynamic potential. If you imagine the values of this potential as rising or increasing values, $\vec{e}_{\alpha}$ points in the direction of the descent, which mathematically represents the (negative) gradient.

As soon as $\vec{e}_\alpha$ has been calculated based on the static or dynamic potential, the value is applied in the driving force term. The total of the driving force and the social forces is included for acceleration of the pedestrian in the respective time step.

10.13.4.3 Defining the Dynamic Potential for a static pedestrian route

You define the dynamic potential on the destination of a pedestrian route (static).

1. On the Network objects toolbar, click **Pedestrian Routes**.
2. In the Network Editor, click on the destination point (default color: turquoise) of the desired static route.

   The course of the static route is shown as a yellow line by default. The routing decision, the destination point, auxiliary points and intermediate points are shown in color.

3. Double-click the destination.

   *The **Pedestrian Route Location** window opens.*

4. Select the option **Use dynamic potential**.
5. Edit the attributes (see "Dynamic potential attributes" on page 972).
6. Confirm with **OK**.

The window closes. The destination point and the intermediate points of the pedestrian route, for which a dynamic potential is defined, are shown as squares.

You can also define the dynamic potential for a ramp or stairway (see "Defining the Dynamic Potential for a ramp or stairway" on page 971).

10.13.4.4 Defining the Dynamic Potential for a ramp or stairway

1. Make sure that the following columns are displayed in the **Ramps & Stairs** list:
   - **UseDynPot**: **Use Dynamic Potential**
   - **g**: **Basic force**
   - **h**: **Direction impact**
   - **Impact**: **Impact**
   - **CalcInt**: **Calculation interval**
   - **ObstDist Obstacle distance**
   - **Cell size**
2. Select the desired option in the row of the required construction element, in the **Use dynamic potential** column.
3. Edit the attributes (see "Dynamic potential attributes" on page 972).

⇨	Tip: Alternatively, you can define the dynamic potential for a ramp or stairway in the **Ramps & Stairways** window on the **Behavior** tab.(see " Attributes of ramps and stairs, moving walkways and escalators" on page 913).

You can also define the dynamic potential for a static pedestrian route. (see "Defining the Dynamic Potential for a static pedestrian route" on page 971).

10.13.4.5 Dynamic potential attributes

You can edit the dynamic potential attributes for a ramp or stairway and for a static pedestrian route in the following elements:

- For a ramp or stairway in the **Ramps & Stairs** list (see "Defining the Dynamic Potential for a ramp or stairway" on page 971).
- For a static pedestrian route in the linked **Route locations** list or in the **Pedestrian Route Location** window (see "Defining the Dynamic Potential for a static pedestrian route" on page 971).

Element	Description
Impact	Deviation from the static potential. Factor influencing $\mathbf{e_{alpha}}$ for weighting when calculating the path selection (see "Description of the method Dynamic Potential" on page 969). • Standard value 100 % = Pedestrians follow the direction which is calculated on the basis of the Dynamic Potential. • 0 % = Pedestrians follow the direction which is calculated on the basis of the Static Potential. The angles inbetween are obtained from the values inbetween. A useful impact depends on the value of the basic force **g**.
Calculation interval	Time interval after which the potential in each case should be updated. As the dynamic potential requires a lot of calculation time, an extension of the calculation interval can ensure an acceptable simulation speed with a slow computer and a lot of active dynamic potential attributes. However, a shorter calculation interval improves the results. If the pedestrian route is part of a pedestrian link, select a sufficiently short calculation interval. If the calculation interval is too large, pedestrians cannot walk around vehicles parked along the route or they have to take an unnecessary detour.

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>g (basic force)</td><td>Defines how the loss time for occupied grid cells is estimated in relation to unoccupied ones, when a pedestrian must walk around a group of pedestrians, forming a circle or square, and there are no obstacles that could extend the path (see "Defining global model parameters" on page 871). Default value for g = 1.5.

Basic force g = 1.5 and impact 100%: Pedestrians no longer want to use the fastest path and only react to other pedestrians in order to avoid collisions.
g > 3 and impact of approximately 100% or only slightly below can lead to unrealistic behavior of pedestrians, for example, to zig-zag movements or stopping short.
If the effect of a dynamic potential with an impact of 100% also appears too weak, increase the value of the parameter g. Do not enter an impact over 100%.
Basic force g = 0: estimated loss time = 0. Impacts simulation as if the dynamic potential was switched off. The estimated travel time is proportional to the remaining path distance. Thus, routing is more likely to take place along the shortest path. Pedestrians only react to other pedestrians in order to avoid collisions.
Basic force g = 1: the estimated loss time is as long as the time it takes to cross this cell in the unoccupied state, i.e. the required time is doubled.
</td></tr>
<tr><td>h (direction impact)</td><td>Influence of the direction of movement of a pedestrian on the calculation of the dynamic potential (see "Defining global model parameters" on page 871).
Default = 0.7

h = 0.0: There is no influence.
h = 1.0: Pedestrians, moving at a free walking speed, are evaluated depending on the direction of the geographically shortest path to the route destination for which the dynamic potential is activated:

not evaluated if they move in the exact direction of the route destination
evaluated twice if they move in the exact opposite direction to the route destination
evaluated once if they move in the exact orthogonal direction to the route destination
Slower pedestrians in the direction of the destination are graded less than once accordingly.
Slower pedestrians away from the destination are graded between once and twice.

h > 1.0: only useful in exceptional cases. Thus a negative value cannot be achieved.
</td></tr>
</table>

10.13.4.6 Route selection method Use service point selection

You can use the **Service point selection** method to perform dynamic pedestrian routing.

Tip: Network files, including examples, can be found in the folder *..\Documents\PTV Vision\PTV Viswalk <Version>\Queuing*:

- *04 Service Points.inpx*
- *04 Service Points 2.inpx*

Use cases for the service point selection method

The route choice method **Service point selection** is suited for the following use cases:

- **Central queue**
  To model a single joint queue for multiple service points. In reality, the "first come – first served" principle is practiced in post offices, at train stations or airports. Pedestrians are not only queuing in front of the service points, but they may also wait in areas in front of these points. The **Queue** attribute of the area which relates to the partial route decision must be selected. The area is thus turned into a queue area.
- **Immediate service point allocation**
  A simple decision model for multiple service points. The pedestrians wait at each service point with a separate queue. The pedestrian has to decide which queue to join. Normally, pedestrians will choose the queue with the shortest waiting time. However, it is not easy to tell which one that is, particularly when there is a large number of service points and/or queues, e.g. at supermarket checkouts or ticket gates. The **Queue** attribute of the area in front of the service points must be selected. The areas are thus turned into queue areas.
- **Survey/interview**
  Individual pedestrians walking by are asked to stop for a minute, e.g. to answer a few questions in a questionnaire. Afterwards, they continue their route.
- You can also model pedestrians who wait in a central area and then go to the service points where queues may occur.

Effects on routing

Pedestrians affected by this partial routing decision can be influenced in their routing behavior as follows:

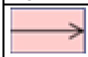
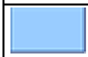
- Routing to a central queue in the area containing a partial routing decision. This area must be a queue area. The waiting time refers to the area containing a partial routing decision.
- Direct routing to a queue in a queue area. The area where the partial routing decision is made must not be a queue area. At least one of the service points should not have reached its maximum number of waiting pedestrians in order to allow pedestrians to join the queue. You can enter them as **Proceed to service point if no more than __ people are queuing there** parameter of the partial routing decision. If all queues are longer, the partial routing decision is ignored and the pedestrian ignores the service points.

The service point is the first queue area on the course of the route that includes an intermediate point of the partial route.

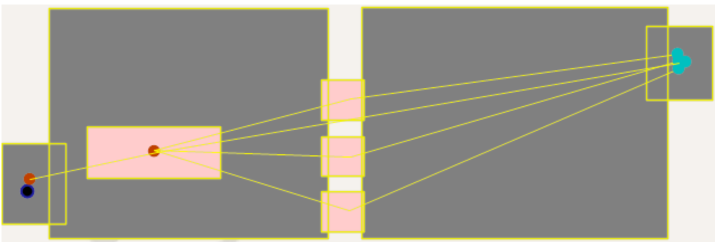
Modeling suggestions for these use cases

Tip: Your Vissim installation provides modeling examples for test applications:
..\Examples Training\Pedestrians\Queuing\08 - Service Desk Partial Routes - Use Cases.inpx

In all figures below, the pedestrian streams go from left to right.

Symbol	Name	Description
	**Pedestrian area**	The **Queue** option is selected.
	**Pedestrian area**	The **Queue** option is not selected.
	**Static routing decision**	Route point where a pedestrian route starts (static). In the pictures below, in the left areas in which pedestrian inputs are defined as well.
	**Partial routing decision**	Partial routing decision using the route choice method **Service point selection**. In the pictures below, in the waiting areas.

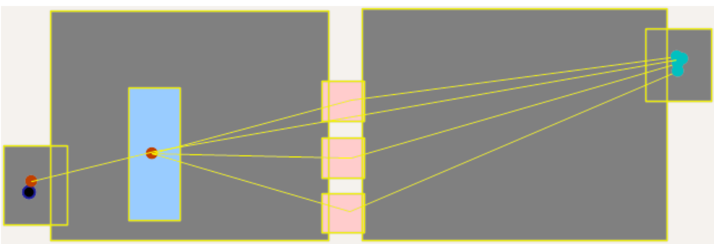
Use case 1: Central queue, the Queue option s selected



Typically, the queue threshold n is as follows: **Proceed to service point if no more than __ people are queuing there** = 0.

This ensures that there is no queue at the service point.

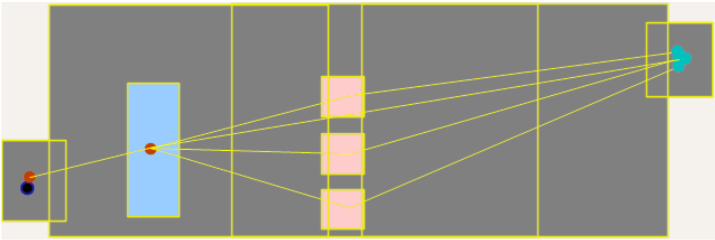
Use case 2: Immediate service point allocation Queue option is not selected



Typically, the queue threshold n is as follows: **Proceed to service point if no more than __ people are queuing there** = 99.

This ensures that all pedestrians join a queue.

Scenario 3: Survey/interview



Typically, the queue threshold n is as follows: **Proceed to service point if no more than __ people are queuing there** = 0.

This ensures that there is no queue at the service point.

10.13.4.7 Prerequisites and requirements of the service point selection method

- The main difference between partial routing decisions with the **service point selection** method and other partial routing decisions:

  For a pedestrian to be able to see the partial route, an intermediate point of the pedestrian's original route must be positioned in the area where the partial routing decision is made. This area is therefore a decision area. It is not necessary to select the **Queue** attribute for this area (see "Attributes of areas" on page 898).

  Pedestrians on a route without an intermediate point in the decision area are not affected by a partial routing decision. These pedestrians are not going to a service point.

- If pedestrians are supposed to go to the service points, each partial route must have an intermediate point in the area for which the **Queue** attribute (queue area) was selected. This way, a queue can be built in this area.
- For a central queue, the partial routing decision must be made in a queue area.
- For immediate service point allocation, the partial routing decision must not be made in a queue area.
- Each queue area must be assigned a wait time distribution via the **Time distribution** attribute (see "Attributes of areas" on page 898).

10.13.4.8 The service point selection method

If the decision area is a queue area with dwell time distribution, the following applies:

- The first pedestrian waits until his waiting time at the decision area has expired. In this case, a wait time distribution must be defined via the **Time distribution** attribute (see

"Attributes of areas" on page 898).

- The first pedestrian waiting continues his way. If there are service points where queues may occur, he will continue his way toward the "best" queue, which is the shortest one. At least at one of the service points should have a queue with no more than n waiting pedestrians. You define this number in the attribute **Proceed to service point if no more than __ people are queuing there.**. If all the queues are full, the pedestrian waits until queue space at one of the service points opens up.
- If people are queuing on or after a while in front of a decision area, pedestrians, whose pedestrian class is not affected by the partial routing, are also affected by the queuing, if an intermediate point of their pedestrian route is part of the decision area. These pedestrians also wait in the same queue until they have reached the decision area, wait the wait time defined in the time distribution, however, do not go to a service point but continue with their original route.

If the decision area is a queue area without dwell time distribution, the following applies:

If people are queuing on or after a while in front of a decision area, pedestrians, pedestrians only queue until they have reached the decision area. They then continue with their original route.

If the decision area is not a queue area, but a time distribution is defined, the following applies:

- A pedestrian of an affected pedestrian class waits until his dwell time has expired and then continues his way.

Calculation method for finding the best queue

- Of all waiting areas the one is selected with no more than n pedestrians waiting in its queue.
- If there is more than one of these queues, the queue with the fewest pedestrians is selected. It is the shortest queue.
- If there is more than one shortest queue, the one with the shortest direct distance to the end of the routing decision that is closest to the pedestrian's coordinates is selected.

Pedestrians on their way to the service point or ready to join the queue are treated as if they already were in the queue.

Note: For each partial route, only the first waiting area after the routing decision is taken into account. Additional waiting areas on the course of the partial route are ignored.

10.13.5 Pedestrian OD matrices

You can enter the pedestrian travel demand for origin-destination relations, i.e. the demand from origin areas to destination areas. This way, you need not define **pedestrian inputs** and **pedestrian routes** in the network. The origin-destination relations between pedestrian areas are displayed in a matrix. In the matrix, you enter the hourly pedestrian volume for each origin-destination relation.

Based on a pedestrian OD matrix, you can save origin-destination data to a file during the simulation run (see "Saving pedestrian travel time measurements from OD data to a file" on page 1048).

Note: Viswalk then internally generates the **pedestrian inputs** and **routing decisions** including **static pedestrian routes**, if these are not available for the origin-destination relation. Thereby the volumes entered are taken into account.

10.13.5.1 Conditions for origin areas and destination areas

Areas, which you select as an origin or a destination for an origin-destination matrix, must meet the following conditions.

Conditions for origin areas in the origin-destination matrix

An area is shown as a row in the origin-destination matrix, if the **Always use as origin area** attribute is selected or if the area contains a pedestrian input for any time interval. The area contains no more than one pedestrian decision. There is no route from this pedestrian routing decision to a stairway/ramp. If the area contains more than one pedestrian routing decision, the **Always use as origin area** attribute has no effect.

Conditions for destination areas in the origin-destination matrix

An area is shown as a column in the origin-destination matrix, if it fulfills at least one of the conditions:

- For the area, the **Always use as a destination area** attribute is selected.
- At least one pedestrian route ends in an area in which an origin area begins.

Notes regarding further network objects

The pedestrian OD matrix only takes into account areas as origins and destinations of pedestrian flows, because pedestrian inputs and routing decisions can only be included in areas.

- Pedestrian routing decisions (partial) and destinations of partial routes in areas are not relevant to the pedestrian OD matrix.
- Stairways, ramps, moving walkways and escalators cannot be the destination of a route in the pedestrian OD matrix, even though the destination of a static pedestrian route can be defined for them.

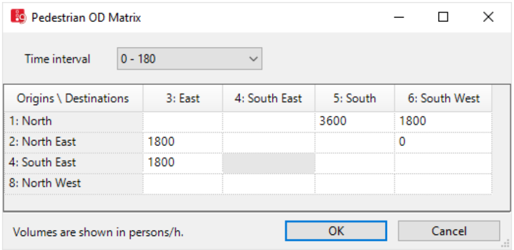
10.13.5.2 Selecting origins and destinations in the Pedestrian OD Matrix

You can select a pedestrian area as origin or destination of pedestrians in the Pedestrian OD Matrix. Specific conditions must be fulfilled for this to occur (see "Conditions for origin areas and destination areas" on page 978). You can also copy and edit existing origins and destinations.

Tip: Alternatively you can select **Always use as origin area** and/or **Always use as destination area** for an area (see "Attributes of areas" on page 898). Thus the area in the Pedestrian OD Matrix is shown as an origin area and/or destination area.

1. From the **Traffic** menu, choose **Pedestrian OD Matrix**.

   *The **Pedestrian OD Matrix** window opens.*



2. Right-click in the window.
3. From the context menu, select the desired entry: **Add Origin** or **Add Destination**.

   A window opens and displays the areas from your network in a list box, which do not contain origins or destinations in the Pedestrian OD Matrix.

4. Select the desired entry.

5. Confirm with **OK**.

   *If you have added the area as an origin, it is shown as a new row in the Pedestrian OD Matrix. The attribute **Always use as origin area** is activated for this area (see "Attributes of areas" on page 898).*

   *If you have added the area as a destination, it is shown as a new column in the Pedestrian OD Matrix. The attribute **Always use as destination area** is activated for this area (see "Attributes of areas" on page 898).*

Notes:

- Vissim does not check if a pedestrian input and a routing decision are in the area.
- In the area, neither a pedestrian input nor a routing decision is generated.

6. If desired, repeat these steps for further areas.
7. Enter the desired volumes in pedestrian/h in the fields (see "Defining the pedestrian demand in the Pedestrian OD Matrix" on page 981).

Tip: You can copy and paste the entries and the matrix via the context menu (see "Defining the pedestrian demand in the Pedestrian OD Matrix" on page 981).

10.13.5.3 Time intervals in the pedestrian OD matrix

The matrix values specified in the pedestrian OD matrix are hourly values referring the respective time intervals. The time intervals of a pedestrian OD matrix are predefined through the time intervals of the following network objects used in the current network:

- Pedestrian Inputs (see "Modeling pedestrian inputs" on page 936)
- Static pedestrian routes (see "Static pedestrian routes, partial pedestrian routes and pedestrian routing decisions" on page 940)

If identical time intervals have been defined for pedestrian inputs and pedestrian routes, you can enter, show, and edit the demand data for each time interval (see "Defining the pedestrian demand in the Pedestrian OD Matrix" on page 981).

You can, e.g., use the Copy-and-Paste command to adopt interval values for another set of time intervals.

If the time intervals are not the same, the first pedestrian input interval and the first pedestrian route are listed above the matrix, in the **Pedestrian OD matrix** window. You cannot edit the time intervals of the other matrix values.

10.13.5.4 Using the matrix values in the pedestrian OD matrix

In a pedestrian OD matrix, the hourly pedestrian demand for each OD relation is displayed in the cells where the Origin column and Destination row overlap.

Each cell value represents the pedestrian volume expected for an origin area to a destination area in the respective time interval.

Demand data in pedestrian OD matrices in [pedestrians/h]

When pedestrian inputs and static pedestrian routes are defined for a network, matrix data is created:

- Total input volume of origin area during input time interval
- Relative volume of static routes to destination area during routing time interval

If no pedestrian inputs or static pedestrian routes have been defined, the program bases the input volume of the origin area for a time interval on user-defined cell data and then generates relative volumes for static routes to the destination for a time interval.

- If you position two routing decisions for different pedestrian classes on a origin area, the origin area is deleted from the origin-destination matrix. If you open the **Pedestrian OD matrix** window, a message is displayed.
- The total input volume is the total of all input volumes of an origin area for a time interval. The pedestrian inputs of an area may consist of different pedestrian compositions (see "Modeling pedestrian inputs" on page 936). The program does not distinguish between different pedestrian types or pedestrian compositions for the total of pedestrian inputs.
- The routing decision of the origin area applies to selected pedestrian classes and/or all pedestrian types (see "Static pedestrian routes, partial pedestrian routes and pedestrian routing decisions" on page 940). For each OD relation there may be several

routes from the routing decision to a destination area. The relative volumes of such parallel routes are added in the pedestrian OD matrix.

The matrix value of a cell includes:

$$Matrix\ value = Input\ volume\ total \cdot \frac{relative\ \ route\ \ volume}{\sum\ relative\ \ route\ \ volumes}$$

Note: You can perform a simulation if one of the following conditions is met:

- The matrix cell value is = 0 as the input value for the origin area in a time interval is 0.
- The matrix cell is empty as there is no pedestrian input or pedestrian routing decision in the origin area, but the option **Always use as origin area** has been selected for the area (see "Defining construction elements as rectangles" on page 890).

10.13.5.5 Defining the pedestrian demand in the Pedestrian OD Matrix

Once you have entered the origins and the destinations, you can define the demand per time interval in the pedestrian OD matrix for all origin-destination relations in the network.

- For origin-destination relations, for which up to now no pedestrian inputs, routing decisions, and/or static pedestrian routes have been defined, pedestrian inputs, routes and routing decisions are generated in Vissim. The entered volume is applied.
- For origin-destination relations, for which pedestrian inputs, pedestrian routing decisions and static pedestrian routes are already defined, the input values and the relevant routing volumes are adjusted respectively. The relationship between unchanged routing volumes is retained.

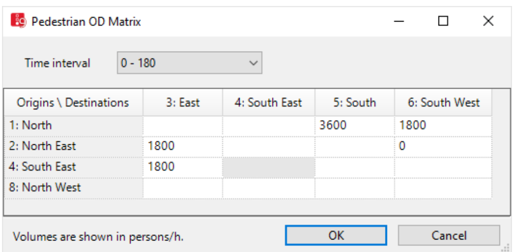
Notes:

- Matrices must not be symmetrical.
- The dimensions of a matrix are the same for all time intervals.

---

1. From the **Traffic** menu, choose **Pedestrian OD Matrix**.

   *The **Pedestrian OD Matrix** window opens. Normally you can edit fields and copy, paste and delete values.*



You may enter values or paste values (e.g. from Microsoft Excel tables used in other programs) from the Clipboard.

Notes:

- The volume per OD relation applies only for the currently selected time interval of the matrix.
- An input value cannot be allocated multiple marked OD relations.

Entering values

1. Enter the desired volumes in pedestrian/h.
2. If you would like to copy values, mark the desired cells.
3. Right-click in the window.
4. Choose the desired entry from the context menu.

Element	Description
Copy matrix	Copy the matrix data of the entire matrix onto the clipboard in order to be able to add it into another time interval or document. The table structure is applied. The labels of the rows and columns are not applied.
Copy matrix incl. headers	Copy the entire matrix onto the clipboard in order to be able to add it to a document. This contains all matrix data including the labels of rows and columns as well as the grid structure.

The value for this relation is applied only for the currently shown time interval.

- If for this relation no values have been entered in another time interval, the value zero is automatically entered for this time interval.
- If values are entered for this relation in other time intervals, the values remain unchanged.

Pasting values from the clipboard

Values from the clipboard are moved from the cell you selected to the cells below and to the right via copy & paste. The number of columns and/or cells from the clipboard must not be higher than the number of columns and/or cells for the origin and destination data used in the pedestrian OD matrix.

Since an area cannot be simultaneously used as an origin and destination, no values are entered in these areas. These areas are gray.

1. Make sure that you have copied the correct values of your choice to the clipboard.
2. In the pedestrian OD matrix, right-click the cell used to copy & paste the values to the cell below and to the right.
3. In the context menu, select **Insert**.

10.13.5.6 Deleting origins, destinations or values in the Pedestrian OD Matrix

You have the option to delete the following data:

- Delete origins line-by-line and destinations column-by-column
- Mark values in multiple cells and delete simultaneously
- normally individual values in fields

Deleting origins or destinations

1. From the **Traffic** menu, choose **Pedestrian OD Matrix**.

   *The **Pedestrian OD Matrix** window opens.*

2. Click on the column header of the destination or the row header of the origin which you would like to delete.
3. Right-click in the list.
4. In the context menu, select **Delete Origin(s)** or **Delete Destination(s)**.

   The marked columns or rows and all OD relations and other pedestrian areas are deleted from the Pedestrian OD Matrix.

   *For the applicable areas, the attributes **Always use as destination area** and **Always use as origin area** are deactivated (see "Attributes of areas" on page 898).*

   *Pedestrian inputs and pedestrian routing decisions, including routes on a deleted origin area, are deleted. Pedestrian routes to a deleted destination area and its relative volumes are deleted. The input volumes on the areas with routing decisions of the deleted pedestrian routes are correspondingly reduced. These changes are immediately applied for **Pedestrian inputs** and **Pedestrian routes** (see "Modeling pedestrian inputs" on page 936), (see "Modeling routing decisions and routes for pedestrians" on page 939).*

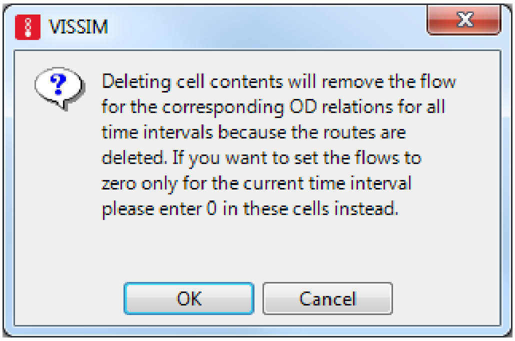
Marking values in multiple cells and deleting simultaneously

1. From the **Traffic** menu, choose **Pedestrian OD Matrix**.

   *The **Pedestrian OD Matrix** window opens.*

2. Mark the cells of the desired OD relations.
3. Press the DEL key.

   A warning is issued. If the content of the selected cells are deleted, the demand data of all other time intervals of these OD relations are also deleted. If for the selected OD relation you would only like to delete the volume for the current time interval, enter 0 in the cell.



4. Confirm with **OK**.

The demand data of the marked relation for all time intervals are deleted. The cells are empty.

10.14 Visualizing pedestrian traffic in 2D mode

You also have the following options for representing the individual pedestrians in color:

- using the default colors (see "Static colors of vehicles and pedestrians" on page 175)
- based on attributes (see "Assigning a color to pedestrians based on an attribute" on page 178)

You can assign display types to the construction elements (see "Defining display types" on page 320).

You can display areas, ramps and stairways on the basis of aggregated values (LOS) (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190).

10.15 Modeling pedestrians as PT passengers

You can model pedestrians as boarding/alighting passengers in PT. The boarding and alighting passengers use PT-vehicles based on PT-specific vehicle types. In the network objects that pedestrians should use as boarding and alighting passengers, select the relevant options for public transport. For example, you may define areas as platform edges or as waiting areas and specify the number of passengers boarding at public transport stops. You may also determine the number of passengers alighting at public transport line stops and define the doors in 2D vehicle models (see "Modeling PT infrastructure" on page 984).

10.15.1 Modeling PT infrastructure

If you model pedestrians as passengers boarding or alighting in PT, they can be going to the station, waiting there for the PT vehicle and boarding once the alighting passengers have left the vehicle. Alighting passengers follow the routes which are assigned to them as soon as they leave a train or a bus. You can easily model such a scenario with waiting areas, platform edges and definitions of the number of boarding and alighting passengers. (see "Quick start: defining pedestrians as PT passengers" on page 987).

You model public transport stops, PT vehicles as well as the associated PT lines by default using the relevant network objects (see "Modeling PT stops" on page 511), (see "Using vehicle types" on page 267), (see "Modeling PT lines" on page 518).

10.15.1.1 Modeling waiting areas and platform edges

- Pedestrians wait in waiting areas of a public transport stop for a PT vehicle of the PT line which they want to use. To define waiting areas for a public transport stop, you must create a pedestrian area and assign the entry **Waiting area** to the attribute **PT usage** (see "Attributes of areas" on page 898). You can also assign multiple public transport stops to a waiting area.
- To define a platform edge, you must create a pedestrian area and assign the entry **Platform edge** to the attribute **Public transport usage** (see "Attributes of areas" on page 898). You can also assign multiple public transport stops to a platform edge.

  Alighting passengers go to the nearest platform edge which they can reach. This depends on the door via which they leave the PT vehicle. If a routing decision is placed on the platform edge, the passenger continues his walk. If no routing decision is placed on the platform edge, the passenger is removed from the network.

  The platform edge and the lane with the public transport stop must be directly adjacent or overlap so that alighting passengers may alight on the platform edge and boarding passengers may board the PT vehicle from the platform edge. You can add a platform edge along a public transport stop in the Network Editor via the context menu (see "Generating platform edges" on page 517). The attributes **Left** and **Right** specify the location of the platform edge parallel to the direction in which you have defined the public transport stop on the link. The number of the platform edge is shown in the attribute **Areas** of the public transport stop.

If a pedestrian is assigned to be at a platform edge or waiting area of a public transport stop for which no boarding passengers are defined, default values are generated.

- These defaults are also generated if a platform edge is added via a context menu.
- Through these default values, each pedestrian who arrives in the waiting area, in the time interval 0 to 99,999, boards a PT vehicle once a PT line serves the public transport stop.

10.15.1.2 Adding alighting passengers

Alighting passengers are added using the setting in the **PT Line Stop** window (see "Editing a PT line stop" on page 526).

- You specify the percentage of alighting passengers per line for each PT line stop.
- You must select the pedestrian composition for the **Alighting composition** according to which the pedestrian types are added to the vehicle.

10.15.1.3 Behavior of boarding and alighting passengers

For each PT stop, you can specify whether pedestrians may board or alight on the right or left side or on both sides (see "Editing a PT line stop" on page 526). By default, all doors on both sides can be used. Boarding passengers wait in the waiting area of the relevant public transport stop. For this waiting area to be reached by the passenger, the destination of a pedestrian route must lie on it.

- The boarding passenger numbers per PT stop result in the PT line with which the waiting passengers want to travel. Once a vehicle from one of these PT lines stops at the PT stop, the passengers walk directly to the nearest door. The pedestrians start to board once all alighting passengers have left the vehicle.
- Passengers can enter until the door is half closed.
- If, depending on the attributes **Door lock duration before departure** and **Door closure delay** of the PT line stop and the **Door closure duration** of the vehicle type, nobody has entered the vehicle, it will leave the PT stop as soon as the departure time is reached. The departure time is based on the timetable or the predefined layover. The degree of **Slack time fraction** of the PT line is taken into account for the scheduled departure time.
- You can select the distribution among the doors in the attribute **Boarding location** of pedestrian area for boarding passengers, and in the PT line stop parameters for alighting passengers (see "Editing a PT line stop" on page 526). Therein, you specify with the option **Late boarding possible**, how a PT vehicle, whose departure time is either prescribed by a schedule or a predetermined time, should respond to a never-ending stream of boarding passengers.
- If a pedestrian is unable to board a PT vehicle, for example for vehicle capacity reasons, the pedestrian returns to a waiting area.
- Default settings are generated for boarding passengers if a public transport stop is assigned to a waiting area or a platform edge for which no boarding passenger share is specified. Due to these default settings, each pedestrian who comes to a waiting area of this public transport stop boards the next PT vehicle that stops.

10.15.1.4 Defining doors for public transport vehicles

A door of a 2D vehicle model has the following properties (see "Defining doors for public transport vehicles" on page 229):

- Position, measured from the front
- Width
- one of the two sides of the vehicle
- For each door, you can specify whether it can be used only for boarding or only for alighting or for both.

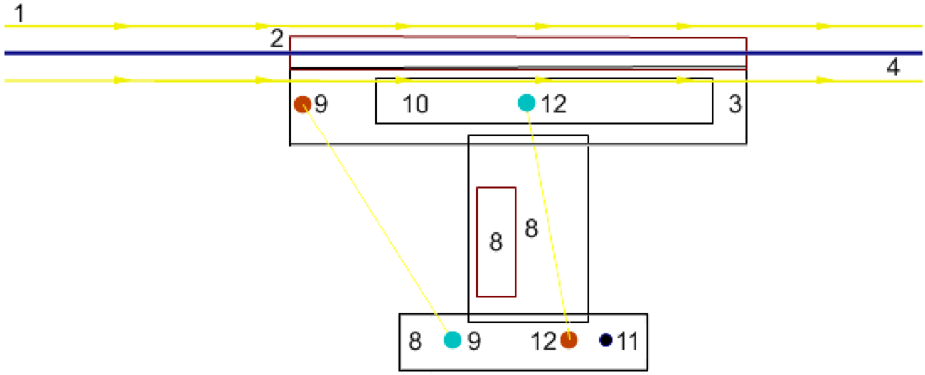
If you use a 3D model, the properties of the door are derived from it.

- You can add or remove doors via the context menu and change the data directly in the base data (see "Defining doors for public transport vehicles" on page 229).
- A warning is issued if changes do not suit the 3D model. You can still confirm these changes. The values in the window are used for the simulation and the values of the 3D model are used only in 3D mode to show the opening and closing of doors.
- If you have not created a door for a vehicle, a door is added temporarily in the middle of the vehicle. Thus pedestrians can still board and alight. They are informed of this via a warning in the trace file.

10.15.2 Quick start: defining pedestrians as PT passengers

The Quick Start uses the example of the definition of a public transport stop to show you the most important steps to define the necessary network objects and to make the necessary settings. The following figure shows a modeling example in wireframe display.

The numbers in the figure correspond to the number of the step:



1. Add a **Link** (in the figure, amber with arrows) (see "Defining links" on page 407).
2. Add a **Public transport stop** (in the figure, with a red outline) (see "Defining PT stops" on page 512).
3. Add an area next to the **Public transport stop** and select the attribute **Platform edge** or create a **Platform edge** for the public transport stop in the network editor (see "Generating platform edges" on page 517).

   If there is more than one **level**, Viswalk tries to determine the correct level according to the **height** of the link.

   When defining the platform edge, the attribute **Public transport usage** > **Platform edge** is selected for the network object **Area** of the platform edge, and the number of PT stops is entered for the attribute **for PT stops**. As a result, Viswalk calculates the occupancy level of the vehicle which departs from the stop from the number of **passengers** which have actually boarded the vehicle during the pedestrian simulation, and not from the number of boarding passengers.

4. Define a **Public transport line** (in the figure, the blue line) (see "Defining PT lines" on page 519)
5. Edit the **PT Line Stop Parameters** (see "Editing a PT line stop" on page 526):
    - Select the attribute **PT stop active**.
    - **Alighting percentage:**: Specify the total ratio of alighting passengers/passengers.
    - **Alighting location**: Distribute the alighting passengers among the doors available for this purpose.
    - Select which sides of the line vehicles should be available for boarding and alighting.
    - If desired, enter a scheduled departure time as a fixed value and/or a minimum dwell time as a distribution. Scheduled departure time, minimum dwell time and slack time fraction collectively specify for how long at least a PT vehicle stops.

      A departure time according to schedule is derived from scheduled departure time and slack time fraction.

      The PT vehicle departs at the earliest at the later time from this time and the time calculated from the minimum dwell time.
    - If the option **Late boarding possible** is selected, the PT vehicle also remains stopped beyond this calculated time in case pedestrians willing to board stream in.

      Other than stemming this flow, the standstill time is limited upwards only by the capacity of the PT vehicle which is defined in the **PT parameters** window (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275).
6. Select a suitable **3D model** or the PT vehicle(see "Defining 2D/3D models" on page 220).
7. Define the attributes of doors of the PT vehicle (see "Defining doors for public transport vehicles" on page 229).
8. Define the geometry of the entire stop using the desired construction elements such as areas, ramps, stairways and obstacles (in the figure, two areas with a black outline and one obstacle with a dark-red outline) (see "Modeling construction elements" on page 880).
9. Define a **Pedestrian routing decision** (in the figure, the upper red circle) in the area of the platform edge. Define at least one **Pedestrian route** which leads to the location that the alighting passengers are supposed to go to (in the figure, from the upper red circle to the lower turquoise circle) (see "Modeling routing decisions and routes for pedestrians" on page 939).
10. Define an area with a waiting area where boarding passengers should wait for the PT vehicle (see "Attributes of areas" on page 898).
11. For the boarding passengers, define a **Pedestrian input** as an origin (in the figure, the black circle)(see "Defining pedestrian inputs" on page 936).
12. Define a **Pedestrian routing decision** in the area of the pedestrian input and a pedestrian route to the waiting area (in the figure, from the lower red circle to the upper turquoise circle) (see "Modeling routing decisions and routes for pedestrians" on page 939).

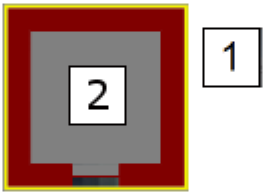
13. Enter the **Capacity** of the PT vehicle (see "Changing attributes for a vehicle type for the duration of boarding and alighting" on page 275).

10.16 Modeling elevators

You can define elevators for vertical transportation of pedestrians. The modeling and simulation of elevators is based on the following elements:

Elevator

The **elevator** is a network object that in the 2D mode is defined and displayed as a rectangular network object (1) (see "Defining elevators" on page 992). The rectangle contains a shaft and a cab (2). The elevator cab moves in the shaft. There is a distance of 30 cm (brown area) between the exterior wall of the shaft and the cab. This area is displayed based on the attribute **Shaft display type** of the elevator.



- Cab

  The elevator cab holds passengers. The cab is part of the elevator, not a separate network object. You can use the following elevator attributes to define the display of the cab: **Floor display type**, **Ceiling display type**, **Wall display type** (see "Elevator attributes" on page 993). The elevator attributes contain additional attributes for the cab, e.g. cab height or capacity. In the cab, a specific walking behavior prevails (see "Walking behavior of pedestrians when using elevators" on page 991).

- Door

  The door is part of the elevator. It is not a separate network object. You can define a door for each of the four cab walls (in the figure on bottom wall). The door is displayed as a thin line in the middle of the wall. The door is positioned automatically:

  - within the first wall of the elevator, when in the Network editor, you create the rectangle for the elevator (see "Defining elevators" on page 992).
  - within the bottom wall of the elevator, when in the Network editor, from the shortcut menu, you choose **Add New Elevator**. By default, the elevator is square.

  You can define the display of the door through the **Door display type** attribute of the elevator (see "Elevator attributes" on page 993). Width and horizontal position of the door within the elevator wall can be defined via the door attributes **Width** and **Center offset (horizontal)**(see "Elevator door attributes" on page 994).

  In the attribute **Active levels** (**ActLvls**), you can assign the elevator door the levels you want it to stop on and open the door (see "Elevator attributes" on page 993). The door only opens on the levels you have selected.

During simulation, the animated door is opened and closed based on the elevator attribute **Motion state**. Shaft door and cab door are displayed as one single door.

Elevator group

An **Elevator group** consists of elevators that you can call (see "Defining an elevator group" on page 995). Each elevator must be assigned to an **Elevator group** (see "Attributes of elevator groups" on page 996). To do so, select the desired elevator group for the elevator attribute **Elevator group** (**ElevGrp**) (see "Elevator attributes" on page 993).

All elevators of an elevator group must be able to stop on the same level. To enable this, in the attribute **Active levels** (**ActLvls**), you can assign the door of each elevator the levels you want it to stop on and open the door (see "Elevator attributes" on page 993). Each level assigned must have a **Waiting area**.

Waiting area

Pedestrians need an area in which they can wait for the elevator. For this area, in the attribute **Waiting area for elevator group**, select the number of the elevator group with the elevators the pedestrians are waiting for (see "Attributes of areas" on page 898). The waiting area can be an area the pedestrians use to enter or exit the elevator or another area they can reach via areas, ramps & stairs. This area must not contain a pedestrian input.

Walkable area

The door must be located at or within a walkable area, so that pedestrians can enter and exit the elevator cab. Otherwise, the simulation will not start. There must be a walkable area on each level the elevator can stop at and pedestrians are supposed to enter or exit the elevator cab.

Pedestrian routes and partial pedestrian routes

Pedestrians use an elevator when their pedestrian route or partial pedestrian route leads from an area on one level to an area on a different level or ends there and using the elevator gets them there quicker. Pedestrians also use an elevator that e.g. only connects levels 2 and 3, even though their route connects levels 1 and 4 without any intermediate points. Pedestrians must be able to reach the elevator and their destination via areas and/or walkable construction elements. For pedestrians to be able to enter and exit the elevator, in the elevator attribute **Active levels** (**ActLvls**), you must assign the elevator door the levels it is supposed to stop on and open the door (see "Elevator attributes" on page 993). These levels must contain areas in which pedestrians can wait for the elevator, enter and exit it and reach their destination. Using intermediate points, you can influence the course of the pedestrian route or partial pedestrian route (see "Modeling the course of pedestrian routes using intermediate points" on page 957).

If a route location of a pedestrian route or partial pedestrian route lies within a waiting area on a level where an elevator can stop, the pedestrian must use its elevator group, even if other elevator groups or ramps & stairs were available.

If a route location of a pedestrian route or partial pedestrian route lies on a ramp or stairs and for this route location you selected the attribute **Ban elevator use**, the pedestrian will not use the elevator until he has reached the next route location for which this attribute has not been selected (see "Attributes of pedestrian route locations" on page 958). This means you can use this option to force pedestrians to use the stairs to reach multiple levels, without having them use the elevator in between. When doing so, you do not have to set a lot of route locations in the stairway.

Level

An elevator must service at least two levels. All elevators of an elevator group must be able to stop on the same level. To enable this, in the attribute **Active levels** (**ActLvls**), you can assign the door of the elevator the levels you want it to stop on and open the door (see "Elevator attributes" on page 993). The door only opens on the levels you have selected.

An elevator stops at a level in the following cases:

- when pedestrians wish to alight: The pedestrian route or partial pedestrian route leads via an area on this level or ends there.
- when pedestrians wish to board: The pedestrian route or partial pedestrian route leads to an area on another level where this elevator stops and can open the door. A waiting area must be defined for the level the pedestrians are waiting on.

10.16.1 Walking behavior of pedestrians when using elevators

Pedestrians use area-based walking behavior that in turn is based on a social force model and walkable construction elements (see " Modeling area-based walking behavior" on page 932), (see "Pedestrian simulation" on page 860).

10.16.1.1 Walking behavior of pedestrians when entering an elevator cab

Pedestrians enter the cab as soon as the door opens and the pedestrians wanting to continue their route on this level have exited the cab. When the elevator **capacity** has been reached, pedestrians stop entering the cab and wait in the waiting area for the next elevator of the elevator group or until the elevator stops again (see "Elevator attributes" on page 993).

When entering the cab, pedestrians use the walking behavior specified in the area behavior type for the area from which they enter the elevator.

10.16.1.2 Walking behavior of pedestrians in an elevator cab

In reality, persons in elevators are willing to accept less space than in other situations where more space is available. In an elevator cab, pedestrians use the walking behavior **Area behavior type (in cab)** (**AreaBehavTypInCab**) of the elevator group the elevator is assigned to. The default walking behavior in this case is **Elevator (in Cab)**.

10.16.1.3 Walking behavior of pedestrians when exiting an elevator cab

Pedestrians exit the elevator as soon as it reaches the floor on which they want to continue with their route and the door opens. If pedestrians cannot alight, e.g. because there are too many pedestrians standing in front of the door or because the door closes too soon, the elevator continues to operate. The stop request for the same floor still exists and the elevator services this floor again as per the control request.

When exiting an elevator, pedestrians use the walking behavior **Area behavior type (alighting)** (**AreaBehavTypAlight**) of the elevator group the elevator is assigned to. The default walking behavior in this case is **Elevator (Alighting)**.

The pedestrians that are not alighting (because they do not have to continue with their route) leave enough space in front of the door for other pedestrians to exit the cab and continue with their route on the respective floor. The doors exercise a “repellent force” on those pedestrians using the walking behavior **Elevator (in the cab)**. This “repellent force” is not exercised on pedestrians with the walking behavior **Elevator (Alighting)**.

10.16.2 Defining elevators

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

When you create an elevator, the cab area and area between the exterior of the shaft and cab are automatically created. The elevator can be inserted into an area or next to it, with the door adjacent to the area.

1. On the Network objects toolbar, click **Elevators**.
2. In the Network editor, point the mouse pointer to the desired position of the first corner point of the elevator.

   In a next step, you define the first wall of the elevator. A door is automatically added to the first wall you define. The size of the elevator must be at least twice the wall thickness. Make sure that the elevator is sufficiently large.
3. Press the CTRL key, hold down the right mouse button and drag the mouse pointer to second corner point of your choice.
4. Release the keys.
5. Drag the corner point open to the desired width or vertically to the desired depth.
6. Double-click.

   Elevator, shaft and doors are displayed in the Network editor. The elevator is selected.

Tip: Alternatively, in the Network editor, from the shortcut menu, choose **Add New Elevator**. By default, a square elevator is added and selected. Using the anchor points, you can drag the elevator open the desired size.

The elevator is automatically assigned to the elevator group with the lowest number. If no elevator group has been defined, Vissim will create one.

7. Edit the attributes of the elevator (see "Elevator attributes" on page 993).
8. Edit the attributes of the area that is meant to be the elevator waiting area (see "Attributes of areas" on page 898).
9. Confirm with **OK**.

The attributes are saved to the elevator list.

10.16.3 Elevator attributes

1. From the **Lists** menu, choose > **Pedestrian Traffic** > **Elevators**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Long name	Short name	Description
Number	**No**	Unique **number** of the elevator
Name	**Name**	Name of elevator
Elevator group	**ElevGrp**	Elevator group to which this elevator belongs (see "Attributes of elevator groups" on page 996)
Queue order	**QueueOrder**	(see "Attributes of areas" on page 898), (see "Selecting network settings for pedestrian behavior" on page 204)
Cab height	**CabHeight**	Inner height of elevator cab [m]. This value defines whether the floor located above the highest floor serviced by the elevator is “broken through”. This way, an obstacle is created there.
Shaft display type	**ShaftDisplTyp**	Display type of area between exterior of elevator shaft and cab
Floor display type	**FloorDisplTyp**	Display type of cab floor
Wall display type	**WallDisplTyp**	Display type of cab walls
Ceiling display type	**CeilDisplTyp**	Display type of cab ceiling
Door display type	**DoorDisplTyp**	Display type of cab door

Long name	Short name	Description
Capacity	**Capacity**	Maximum permitted number of pedestrians per elevator cab. Once the capacity is reached, no additional pedestrians can enter the elevator.
Motion state	**MotionState**	Result attribute with current cab movement: - **Stopping**: Cab comes to a halt - **Going Up**: Cab moves upward - **Going Down**: Cab moves downward
Destination level	**Destlvl**	Next floor the elevator cab stops at. If the elevator is called from outside, the destination level can change.
Travel direction	**TravDir**	Direction in which the elevator is currently moving. When the cab has come to a halt, the direction of the next ride is indicated.
Current level	**CurLvl**	Level the elevator cab is currently on.

By default, the display types are based on the display types selected as default in the network settings (see "Network settings for standard types of elevators and elevator groups" on page 207).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Points: edit coordinates of the corners
- Doors (see "Elevator door attributes" on page 994)

The attributes are described further above.

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

10.16.4 Elevator door attributes

1. From the **Lists** menu, choose > **Pedestrian Traffic** > **Elevators**.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

2. On the list toolbar, in the **Relations** list box, click > **Doors**.

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

Long name	Short name	Description
Edge index	**EdgeIdx**	Position of elevator door as edge index of rectangle that defines the shaft. The edge indices 1 to 4 are counted counterclockwise. 1 = First edge modeled in the Network editor when the elevator was defined. If the elevator is created in the Network editor via shortcut menu command **Add New Elevator**, this is the bottom elevator wall.
Active levels	**ActLvls**	Number of levels on which the door opens. Select at least two levels. The **Main landing level** of the elevator group must be selected to which the elevator is assigned.
Avoid making the total of width and center offset larger than the width of the elevator wall with the door:		
Center offset (horizontal)	**CentOffset**	Horizontal offset of the door's center line to the cab wall's center line [m]. In the Network editor, in the top view, negative values move the door to the left. In the Network editor, in the top view, positive values move the door to the right.
Width	**Width**	Entire door width [m]. With the door open this is the width of its opening. Default value 0.9 m, if permitted by the width of the wall.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

10.16.5 Defining an elevator group

In the **Elevator group** attribute list, you can define new elevator groups and assign elevators to them.

1. Make sure that the desired elevators have been defined (see "Defining elevators" on page 992).
2. From the **Lists** menu, choose > **Pedestrian Traffic** > **Elevator groups**.

   *The **Elevator group** attribute list opens. If no elevator groups have been defined yet, only the column headings are shown.*

   By default, you can edit the list (see "Using lists" on page 93).

   In the list, you can define a new elevator group.

3. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

4. Edit the attributes (see "Attributes of elevator groups" on page 996).

10.16.6 Attributes of elevator groups

1. From the **Lists** menu, choose > **Pedestrian Traffic** > **Elevator groups**.

Tips: The **Elevator groups** list may also be displayed via the following functions:

- Call it from the **Elevators** list: Right-click an elevator in the**Elevators** list and choose **Show Elevator Group List** from the shortcut menu. If **Synchronization** is activated in the toolbar of the **Elevators** list , the elevator group that the elevator is allocated to will be chosen from **Elevator groups** list.
- Call it from the network editor, if an elevator is defined in the network editor: Right-click an elevator and choose **Show Elevator Group List** from the shortcut menu. If **Synchronization** is activated in the toolbar of the **Elevator groups** list , the elevator group that the elevator is allocated to will be chosen from **Elevator groups** list.

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

The list on the left may include the following attributes:

Long name	Short name	Description
Number	**No**	Unique number of the elevator group
Name	**Name**	Name of elevator group
Acceleration	**Accel**	Value for acceleration and deceleration of elevator cabs in shaft [m/s²] assigned to this elevator group
Maximum speed	**MaxSpeed**	Maximum speed of elevator cab assigned to this elevator group [m/s]
Door motion duration	**DoorMotionDur**	Time [s] to open the elevator doors of cabs assigned to this elevator group. This also applies for the closing of elevator doors.
Door closure delay	**DoorClosDel**	Time [s] after which a pedestrian must have cleared the elevator door and the door begins to close. In reality, the time after which the light barrier is no longer interrupted.
Door hold time (minimum)	**DoorHoldTmMin**	Minimum time [s] the door must remain open when cab stops at a level.
Main landing level	**MainLvl**	Number of level on which the elevator cabs assigned to this elevator group are when the simulation is started
Area behavior type (in cab)	**AreaBehavTypInCab**	Area behavior type used by pedestrians in the cab

<table>
<tr><th>Long name</th><th>Short name</th><th>Description</th></tr>
<tr><td>Area behavior type (alighting)</td><td>AreaBehavTypAlight</td><td>Area behavior type used by pedestrians when alighting from the cab</td></tr>
<tr><td colspan="3">Pedestrians who wish to board the elevator may form queues in front of the elevator door and let the pedestrians pass who want to alight the elevator. These queues are formed to the left and the right of the elevator door, on a Vissim-internal area. Pedestrians alighting the elevator move between these two queues. The shape and orderliness of queues are based on the following attributes:</td></tr>
<tr><td>Queue straightness</td><td>QueueStraight</td><td>The Queues straightness defines the shape of the queue. Value range between 0.0 and 1.0. The greater the value, the more straight the queue will look:
0.0: snake shaped queue
1.0: straight queue
Default values are defined in the network settings (see "Selecting network settings for pedestrian behavior" on page 204). These are used if this attribute is not defined for this elevator group.
If no waiting behavior is defined for the waiting areas, the attribute affects all elevators of this elevator group on all floors and for all pedestrian classes.</td></tr>
<tr><td>Queue order</td><td>QueueOrder</td><td>The Queue order defines how orderly the pedestrians line up in the queue. Value range between 0.0 and 1.0. The higher this value, the more orderly pedestrians line up one behind the other:
0.0: Pedestrians are standing together in groups
1.0: Queue of pedestrians lined up one behind the other
Default values are defined in the network settings (see "Selecting network settings for pedestrian behavior" on page 204). These are used if this attribute is not defined for this elevator group.
If no waiting behavior is defined for the waiting areas, the attribute affects all elevators of this elevator group on all floors and for all pedestrian classes.</td></tr>
</table>

By default, the area behavior types are based on the area behavior types selected as default in the network settings (see "Network settings for standard types of elevators and elevator groups" on page 207).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

The list on the right contains attributes and attribute values of network objects, and/or base data allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119):

- Elevators: attributes of the elevators assigned to this elevator group (see "Elevator attributes" on page 993)
- Waiting areas: areas for which the attribute **Waiting area for elevator group** has been selected (see "Attributes of areas" on page 898)

2. On the list toolbar, in the **Relations** list, click the desired entry.

3. Enter the desired data.

10.17 Defining pedestrian travel time measurement

Pedestrian travel time measurement is based on a start point (in a start area) and a destination point (in a destination area). The travel time is determined from entering the start area where the start point is located until entering the destination area where the destination point is located, including wait times. You can show this data in an evaluation (see "Evaluating pedestrian travel time measurements" on page 1046).

Travel time measurements are not shown in 3D mode.

If you define travel time measurements in multistory buildings on levels, for example, check whether the respective level is correct for each pedestrian travel time measurement. Go to the list **Pedestrian Travel Time Measurements** and select the attributes **Start area** > **level** and **End area** > **level**. Check the correct position on the various levels in 2D mode by selecting the visibility for the desired level and hiding the other levels (see "Using the Level toolbar" on page 65).

Notes:

- You can specify that you need not press the CTRL key when adding network objects (see "Right-click behavior and action after creating an object" on page 152).
- For some network objects there are windows in which the attributes of a network object can be defined and edited. There are lists for this, for all network objects. You can choose whether you want to open a window, a list or neither of the two for the definition of network objects in the Network Editor (see "Right-click behavior and action after creating an object" on page 152).

1. On the Network objects toolbar, click **Pedestrian Travel Times**.

In the next step, you may choose the position of the start point in the start area.

2. Press the CTRL key and right-click in the destination area.

   Per default, a pink circle is inserted.

3. Drag the cursor to the destination area.
4. Click.

   *Per default, a green dot is inserted. The **Pedestrian Travel Time Measurements** list opens.*

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

5. Edit the attributes:

Element	Description
No	Number of travel time measurement. You can enter a different number. Use a continuous numbering system for the Vissim network. This simplifies the evaluations.
Name	Name of travel time measurement
StartArea	**Start area**: Number and name of the area where the start point is located
EndArea	**Destination area**: Number and name of the area where the destination point is located
StartPt	**Start point**: Position of the start point based on the coordinates of start point (x) and start point (y)
StartX	**Start point (x)**: coordinate
StartY	**Start point (y)**: coordinate
StartShowLabel	**Start - show label**: ☐ If this option is not selected, the label on the start point for an individual pedestrian travel time measurement is hidden if the label for all pedestrian travel time measurements is selected.
StartLabPosRelPt	**Start label position relative**: Coordinates of the label position when the label on the start section is offset
StartLabPosRelX	**Start label position relative (x)**: X-coordinate of the label position when the label on the start point is offset
StartLabPosRelY	**Start label position relative (y)**: Y-coordinate of the label position when the label on the start point is offset
Endpt	**End point**: Position of the end point based on the coordinates of end point (x) and end point (y)

Element	Description
EndShowLabel	**End - show label**: ☐ If this option is not selected, the label on the destination point for an individual pedestrian travel time measurement is hidden if the label for all pedestrian travel time measurements is selected.
EndLabPosRelX	**End label position relative (x)**: X-coordinate of the label position when the label on the destination point is offset
EndLabPosRelY	**End label position relative (y)**: Y-coordinate of the label position when the label on the destination point is offset
EndLabPosRelPt	**End label position relative**: Coordinates of the label position when the label on the destination point is offset

11 Performing evaluations

Depending on the network objects used, various data is produced during simulations, e.g. information on vehicles, links, areas, nodes, traffic jams, green time distribution or PT waiting times (see "Overview of evaluations" on page 1002). This may be raw data or aggregated data. Depending on the type of data and the desired further use, you can show evaluations in lists and/or windows and save them to text files and/or database files. To do so, select the desired evaluations before the start of the simulation and configure them.

Note: Some output options and individual evaluation results may only be available with add-on modules. You need a license to use the add-on modules.

If you wish to display the result data of evaluations in a database, you must configure the database connection (see "Configuring the database connection for evaluations" on page 1018), (see "Saving evaluations in databases" on page 1021).

Tip: You can also show simulation data in diagrams (see "Creating charts" on page 1113).

1. Define basic settings for the management of the result data: overwrite previous results, configure columns, file or database as location of automatic saving, define percentiles (see "Managing results" on page 1007).
2. From the **Evaluation** menu, choose the desired entry:
   - **Configuration**: Opens the **Evaluation Configuration** window with the following tabs that allow you to make basic settings for result data management and evaluations:
     - **Evaluation output directory** box: By default, the path to the evaluation output directory, the directory to which the currently opened network file **.inpx* is saved. If you are using Scenario Management and a scenario has been opened, you can view the path to the directory in which the scenario is saved.
     - **Result Management** tab: Under Result Management, make the basic settings for managing result data before you configure **Result Attributes** or the **Direct Output** and start the simulation.
     - **Result Attributes** tab: Select evaluations which you wish to display in attribute lists or result lists (see "Configuring evaluations of the result attributes for lists" on page 1014)
     - **Direct Output** tab: Select evaluations which you wish to save to a file or database (see "Configuring evaluations for direct output" on page 1018)
3. Confirm with **OK**.
   - **Measurement Definition**: Opens a respective list with static attributes for the definition of the following measurements:
     - In the **Data Collection Measurements** list, select Data Collection Points (see "Defining a data collection measurement in lists" on page 1011), (see "Generating data collection measurements in lists" on page 1011).

- In the **Delay Measurements** list, select Vehicle travel time measurements (see "Defining delay measurement in lists" on page 1012), (see "Generating delay measurements in lists" on page 1013).
- In the **Area measurements** list, select sections (see "Defining an area measurement in lists" on page 1009), (see "Generating area measurements in lists" on page 1010).

4. If you have configured evaluations and wish to create result data from a simulation, start the simulation (see "Running a simulation" on page 840).

The evaluations are generated and, depending on your settings in the selected program elements, they can be shown or saved to files or databases.

5. If you have configured evaluations for network objects, whose results can be displayed in windows or result lists, from the **Evaluation** menu, select the entry of your choice:
   - **Window**: Opens a window with result attributes of the simulation for the chosen evaluation **Signal Times Table**, **SC detector record** or **Signal changes** (see "Showing evaluations in windows" on page 1022)
   - **Result lists**: opens a list with result attributes from the simulation for the selected evaluation (see "Showing result attributes in result lists" on page 1016)

Notes:

- By default, the currently selected units for distances, speeds, accelerations, decelerations are taken into account for the evaluation (see "Selecting network settings for units" on page 205). You can also choose the unit for attribute values in the attribute selection list in the **Format** column (see "Selecting attributes and subattributes for columns of a list" on page 112).
- Some parameters use different units. These are specified in the evaluation.
- If the script files change attributes that are used in evaluations, the changed values are saved to the evaluations.

11.1 Overview of evaluations

Depending on the selected evaluation, the type of data and the desired further use, you can show evaluations in lists and/or windows and save them to text files and/or database files:

- **Direct Output**: The data of the evaluation is saved as text files. In text files, semicolons are used as a separator. Thus you can import text files into spreadsheet programs, e.g. Microsoft™ Excel™, and use them for calculations, analyses or graphical display.

  You can also save the data of the following evaluations to database files:

  - Vehicle record
  - Vehicle travel times (raw data)
  - Pedestrian record
  - Nodes (raw data)
  - Signal changes

Some evaluations contain raw data. These are already written during the simulation, while the data of other evaluations are written after the simulation.

- **Result Attributes**: Data is aggregated to result attributes during the simulation. You can show the result attributes like static attributes in the following program elements:
  - in result lists
  - in the attribute lists of network objects
  - in the quick view
  - in Network Editors as labeling of network objects
- **Windows**: You may show the data of the following evaluations in windows:
  - SC detector record
  - Signal changes
  - Signal times table
- **Result Lists**: Result lists contain the values of the attributes **Simulation run** and **Time interval** as well as the data of network object-specific attributes. You can save a result list to an attribute file **.att*.

The table under the following list shows the output options for the result data of each evaluation:

- **TXT**: save directly to **.txt* file
- **MDB**: save directly to **.db* file
- **ATT**: Show result attributes in lists and if desired, save to **.att* file
- **Window**: display in a window in Vissim

	Direct output		**ATT**	**Window**
Evaluation	**TXT**	**MDB**		
Discharge record (see "Saving discharge record to a file" on page 1024)	*.dis			
OD pairs (see "Displaying OD pair data in lists" on page 1027)			☑	
Lane changes (see "Saving lane change data to a file" on page 1028)	*.spw			
Vehicle record (see "Saving vehicle record to a file or database" on page 1031)	*.fzp	☑		
Vehicle network performance (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085)			☑	
Vehicle &Travel Times and **Vehicle travel times (raw data)** (see "Evaluating vehicle travel time measurements" on page 1096)	*.rsr	☑	☑	
Vehicle input data	*.fhz			

	Direct output		ATT	Window
Evaluation	**TXT**	**MDB**		
(see "Saving vehicle input data to a file" on page 1110)				
Areas & Ramps (see "Evaluating pedestrian density and speed based on areas" on page 1034)			☑	
Pedestrian grid cells (see "Grid-based evaluation of pedestrian density and speed" on page 1037)			☑	
Pedestrian network performance (see " Pedestrian network performance: Displaying network performance results (pedestrians) in lists" on page 1090)			☑	
Area measurements and **Area measurements (raw data)** (see "Evaluating pedestrian areas with area measurements" on page 1041)	*.merp		☑	
Pedestrian record (see "Saving pedestrian record to a file or database" on page 1053)	*.pp	☑		
Pedestrian travel times and **Pedestrian travel times (raw data)** (see "Evaluating pedestrian travel time measurements" on page 1046)	*.rsrp		☑	
Pedestrian travel times (OD data) (see "Saving pedestrian travel time measurements from OD data to a file" on page 1048)	*.rsmp		☑	
Green time distribution (see "Saving SC green time distribution to a file" on page 1078)	*.lzv			
Nodes and **Nodes (raw data)** (see "Evaluating nodes" on page 1057)	*.knr	☑	☑	
Convergence (see "Saving data about the convergence of the dynamic assignment to a file" on page 1067)	*.cva			
SC detector record (see "Evaluating SC detector records" on page 1070)	*.ldp			☑
Signal changes (see "Evaluating signal changes" on page 1081)	*.lsa	☑		☑
Managed Lanes (see "Saving managed lane data to a file" on page 1084)	*.mle			

	Direct output		ATT	Window
Evaluation	**TXT**	**MDB**		
Vehicle network performance (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085)			☑	
Meso edges (see "Showing meso edges results in lists" on page 1064)			☑	
Public transport waiting times (see "Saving PT waiting time data to a file" on page 1092)	*.ovw			
Data &Collection Measurements and **Data &Collection Measurements (raw data)** (see "Evaluating data collection measurements" on page 1093)	*.mer		☑	
Signal times table (see "Showing signal times table in a window" on page 1098)				☑
SSAM (see "Saving SSAM trajectories to a file" on page 1102)	*.trj			
Queue Counters (see "Showing results of queue counters in lists" on page 1105)			☑	
Links (see "Showing data from links in lists" on page 1103)			☑	
Delays (see "Showing delay measurements in lists" on page 1107)			☑	
Paths (see "Showing data about paths of dynamic assignment in lists" on page 1109)			☑	

11.2 Comparing evaluations of PTV Vissim with evaluations according to HBS

Using the evaluation results obtained with Vissim, you can carry out an evaluation according to HBS (Handbuch für die Bemessung von Straßenverkehrsanlagen, Manual for Road Infrastructure Evaluation), a system of rules applied for economic construction of road infrastructure in Germany.

Thereby, a systematic deviation between the simulation results and the parameters analytically calculated according to HBS might occur:

- The HBS calculation is based on a model of traffic and nodes that is abstract to a certain degree. It cannot always include local particularities, as is possible with the detailed modeling of microscopic simulation in Vissim.

- The evaluation parameters of the HBS refer to a static and thus long-term case, e.g. to wait times. They analytically account for the wait time shares of all vehicles passing the node. During simulation, the recording is limited to the section for which modeled travel time measurements or delay measurements are available. In case of congestion according to HBS calculation, there is a much steeper increase in wait times than shown in the simulation with Vissim. Make sure you account for this when evaluating the results.

Note: Due to the stochastic nature of the simulation, random fluctuations occur in the results of the individual simulation runs. A more reliable assertion is only reached through averaging the results of a sufficient number of simulation runs with different random seeds.

11.3 Performing environmental impact assessments

To perform environmental impact assessments, you may need vehicle emission data. Using Vissim, you have the following options to calculate the exhaust emission of vehicles.

11.3.1 Simplified method via node evaluation

Node evaluation also determines exhaust emissions (see "Evaluating nodes" on page 1057). The basis for these are formed by standard formulas for consumption values of vehicles from TRANSYT 7-F, a program for optimizing signal times, as well as data on emissions of the Oak Ridge National Laboratory of the U.S. Department of energy. The data refers to a typical North American vehicle fleet and does not differentiate between individual vehicle types. This allows you to use node evaluation for a more simple comparison of the emissions produced during different scenarios.

11.3.2 Precise method with EnViVer Pro or EnViVer Enterprise

EnViVer Pro is a program used to calculate exhaust emissions based on detailed vehicle record data from Vissim. Using this add-on module, you can determine the exhaust emissions for individual vehicle types.

The statistical emissions model in EnViVer Pro and EnViVer Enterprise calculates realistic values for CO_2, NO_x and PM10 emissions of different vehicle categories. The calculated emissions are based on speed-time profiles of vehicles. The emission models are updated annually and are kept for future calculations. With the update, new emission models are added and existing models, if available, are updated. EnViVer Pro and EnViVer Enterprise thus provide both current and previous emission models, which allow you to easily compare or recalculate values (see "Overview of add-on modules" on page 38). For questions regarding the purchase of EnViVer Pro and EnViVer Enterprise for Vissim, please contact PTV GROUP under the following e-mail address: traffic.info@ptvgroup.com. For information on the products, please refer to the website of the provider TNO (www.tno.nl).

11.3.3 The COM interface or API approach with EmissionModel.dll

This approach requires programming knowledge, as you must create the *EmissionModel.dll* file or write your own scripts for using the COM interface. It allows you to use your own calculation algorithms and simulation data as input.

11.3.3.1 EmissionModel.dll

For the calculation of emissions, you can access the file *EmissionModel.dll* (see "Accessing EmissionModel.dll for the calculation of emissions" on page 1192). You must provide the *EmissionModel.dll*. It is not part of Vissim. The following data is transferred via the interface for all vehicles and each simulation time step:

- Accelerations
- Speeds
- Weights
- ID numbers
- Vehicle types
- Gradients

11.3.3.2 COM interface

To calculate emissions, you can create your own scripts and/or integrate your own applications into Vissim. Using the COM interface, for each time step, you can access all Vissim attributes, e.g. position, speed and acceleration of the vehicles (see "Using the COM Interface" on page 1189).

11.3.4 Noise calculation

Noise calculations are carried out using specialized external software, e.g. CADNA or SoundPLAN. Vissim can provide specific input data for these programs. Alternatively, you can use the COM port, write your own script or embed your own application into Vissim for noise emission calculation (see "Using the COM Interface" on page 1189).

11.3.5 Calculation of ambient pollution

Concentration values determine the impact of environmental interference factors affecting human health and the natural environment. Concentration values are not calculated in Vissim, but in external software programs that specialize in concentration value calculation. Vissim can provide specific input data for these programs.

11.4 Managing results

Before configuring the desired result data and starting the simulation, you can set the following basic settings for managing the result data:

- Keep or delete data from previous simulation runs
- Configure columns to undertake new simulation runs

- For results which are automatically saved, select as destination Database or File
- Define percentiles

Notes: Evaluation data of previous simulation runs are deleted:

- When you select the option **Delete previous simulation runs**
- When you change the configuration of the evaluation
- When you edit or delete network objects the evaluation is based on

Before Vissim deletes the data, a message is displayed.

---

1. From the **Evaluation** menu, choose > **Configuration** > **Result Management** tab.
2. Make the desired changes:

Element	Description

Keep previous simulation runs	- **None**: Select this option to delete the following data:
	  - The evaluation file **.bew* is overwritten.
	  - The path file **.weg* is overwritten.
	  - The entries of the simulation runs in the **Simulation Runs** results list are deleted.
	  - The files in the folder *..\<Name of network file>.results* are deleted.
	  - If under Simulation Parameters you selected **Number of runs** > 1, the simulation run objects are deleted between the simulation runs. However, the direct output files and automatic list export files of the individual simulation runs remain intact.
	- **Only of current (multiple) simulation**: Select this option to save all simulation run data of the current multiple simulation. Data of previous simulation runs is deleted.
	- **Of all simulation runs**: Select this option to save all simulation run data of the current multiple simulation. Data of previous simulation runs is also kept.

Add new columns in lists	only if the result attributes of previous simulation runs are kept. The option **Only of current (multiple) simulation** or **Of all simulation runs** must be selected: If this option is selected, in the results list of the evaluation, the column settings are adopted for each new simulation run. The results of the new simulation run are displayed in the new lines.
Destination for automatic list export	- **File**: If in a list you selected the **Autosave after simulation** symbol, the results are saved to an Vissim attribute file **.att* at the end of the simulation run (see "List toolbar" on page 97). - **Database**: If in a list you selected the **Autosave after simulation** symbol, the results are saved to a database at the end of the simulation run (see "List toolbar" on page 97).

Element	Description
Available percentiles	Define percentiles which specify a share of each of the values of the total results as an aggregated value over all the simulation runs and time intervals, such as 95% of the queue length in queue counters and turn relations in nodes. One value will be considered for each interval. The value range for a percentile is from 0 to 100%.

3. When you wish to define percentiles, repeat the following steps for each percentile:
4. Right-click in the **Available aggregation percentiles** table.
5. From the shortcut menu, choose **Add**.
6. In the **Value** column, enter the desired value for the percentile as a percentage.

   Once the result attributes and/or the direct output has been defined and the simulation runs performed, the results of the percentiles will be calculated for the attributes and displayed chronologically in the result list for each percentile.
7. Confirm with **OK**.
8. Define the desired result attributes and/or the direct output (see "Configuring evaluations of the result attributes for lists" on page 1014), (see "Configuring evaluations of the result attributes for lists" on page 1014).

11.5 Defining and generating measurements or editing allocated objects

You can define the following measurements in your attribute list. You can define data collection measurements and area measurements or generate them based on existing objects:

- Data collection measurements (see "Defining a data collection measurement in lists" on page 1011), (see "Generating data collection measurements in lists" on page 1011)
- Delay measurements (see "Defining delay measurement in lists" on page 1012), (see "Generating delay measurements in lists" on page 1013),
- Area measurements (see "Defining an area measurement in lists" on page 1009), (see "Generating area measurements in lists" on page 1010)

If you select the measurement in the **Evaluation Configuration** window, you can start your simulation and display the result attributes:

- Data collection measurements (see "Evaluating data collection measurements" on page 1093)
- Delay measurements (see "Showing delay measurements in lists" on page 1107)
- Area measurements (see "Evaluating pedestrian areas with area measurements" on page 1041)

11.5.1 Defining an area measurement in lists

In the **Area Measurement** attribute list you can define new area measurements and assign defined sections to them.

Tip: Alternatively, you can generate area measurements from the defined sections (see "Generating area measurements in lists" on page 1010).

1. From the **Lists** menu, choose **Measurements** > **Area Measurements**.

   *The **Area Measurements** attribute list opens. If no area measurement has been generated or added yet, only the column headings are shown.*

2. Right-click in the list.
3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

4. Make the desired changes:

Element	Description
No	Unique number of the area measurement
Name	Designation of the area measurement
Sections	☑ If this option is selected, the section is entered into the box and taken into account for area measurement.

11.5.2 Generating area measurements in lists

You can generate area measurements in the **Area Measurements** attribute list from the defined sections.

Tip: Alternatively, you can define a new area measurement and select the desired sections for it (see "Defining an area measurement in lists" on page 1009).

1. From the **Evaluation** menu, choose **Measurement Definition** > **Area Measurements**.

   *The **Area Measurements** attribute list opens. If no area measurement has been generated or added yet, only the column headings are shown.*

2. Right-click in the list.
3. Select the entry **Generate all (1:1)**.

   Area measurements are generated from the defined sections and shown in the list.

4. If desired, in the **Sections** column, activate or deactivate sections for area measurement.

11.5.3 Editing sections assigned to area measurements

When the area measurements attribute list is open, you can display and edit the attributes of the assigned sections in the list on the right.

1. From the **Evaluation** menu, choose **Measurement Definition** > **Area Measurements**.
2. In the list on the left, click the desired area measurement.
3. On the toolbar in the **Relations** list box, choose > **Sections**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

4. Click on the desired entry.

5. Enter the desired data.

11.5.4 Defining a data collection measurement in lists

In the **Data Collection Measurements** attribute list you can define new data collection measurements and assign defined data collection points to them.

Tip: Alternatively, you can generate data collection measurements from the defined data collection points (see "Generating data collection measurements in lists" on page 1011).

1. From the **Evaluation** menu, choose **Measurement Definition** > **Data Collection Measurements**.

   *The **Data Collection Measurements** attribute list opens. If no data collection measurement has been generated or added yet, only the column titles are displayed.*
2. Right-click in the list.
3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

4. Make the desired changes:

Element	Description
No	Unique number of the data collection measurement
Name	Designation of the data collection measurement
DataCollectionPoints	**Data Collection Points** list: ☑ If this option is selected, the data collection point is entered into the box and taken into account for data collection measurement.

11.5.5 Generating data collection measurements in lists

In the **Data Collection Measurements** attribute list you can generate data collection measurements from the defined data collection points.

Tip: Alternatively, you can define a new data collection measurement and select the desired data collection points for it (see "Defining a data collection measurement in lists" on page 1011).

1. From the **Evaluation** menu, choose **Measurement Definition** > **Data Collection Measurements**.

   *The **Data Collection Measurements** attribute list opens. If no data collection measurement has been generated or added yet, only the column titles are displayed.*

2. Right-click in the list.
3. Choose the desired entry from the context menu:
   - **Generate all (grouped)**: Generates the data collection measurements and shows them sorted by the links on which the data collection points are defined.
   - **Generate all (1:1)**: Generates the data collection measurements and shows them sorted by data collection point number.

11.5.6 Editing data collection points assigned to data collection measurements

When the data collection attribute list is open, you can display and edit the attributes of the allocated data collection points in the list on the right.

1. From the **Evaluation** menu, choose **Measurement Definition** > **Data Collection Measurements**.
2. In the list on the left, click the data collection point of your choice.
3. On the toolbar in the **Relations** list box, select **Data collection points**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

4. Click on the desired entry.
5. Enter the desired data.

11.5.7 Defining delay measurement in lists

In the **Delay Measurement** attribute list, you can define a new delay measurements and allocate defined vehicle travel time measurements to them.

1. From the **Evaluation** menu, choose **Measurement Definition** > **Delay Measurements**.

   *The attribute list **Delay Measurements** opens. If no delay measurement has been generated or added yet, only the column headings are shown.*

2. Right-click in the list.
3. From the shortcut menu, choose **Add**.

   A new row with default data is inserted.

4. Make the desired changes:

Element	Description
No	Unique number of the delay measurement
Name	Designation of the delay measurement
VehTravTmMeas	**Vehicle Travel Time Measurement**: ☑ When this option is selected, vehicle travel time measurement is specified in the entry box and accounted for during delay measurement.

11.5.8 Generating delay measurements in lists

In the attribute list **Delay measurements**, you can generate delay measurements from the vehicle travel time measurements defined.

Tip: Alternatively, you can define a new delay measurement and select the desired sections for it (see "Defining delay measurement in lists" on page 1012).

1. From the **Evaluation** menu, choose **Measurement Definition** > **Delay Measurements**.

   *The attribute list **Delay Measurements** opens. If no delay measurement has been generated or added yet, only the column headings are shown.*

2. Right-click in the list.
3. Select the entry **Generate all (1:1)**.

   Delay measurements are generated from the defined vehicle travel time measurements and displayed in the list.

4. If desired, in the **VehTravTmMeas** column, you can activate or deactivate vehicle travel time measurements for delay measurement.

11.5.9 Editing vehicle and travel time measurements assigned to delay measurements

When the attribute list **Delay measurements** is open, in the list on the right, you can show and edit the attributes of the vehicle travel time measurements allocated.

1. From the **Evaluation** menu, choose **Measurement Definition** > **Delay Measurements**.
2. In the list on the left, click the desired delay measurement.
3. On the toolbar, in the **Relations** list box, click > **Vehicle travel time measurements**.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

4. Click on the desired entry.
5. Enter the desired data.

11.6 Showing results of measurements

You can show the result attributes of the following evaluations as a list:

- Data collection measurements derived from data collection points
- Delay measurements derived from vehicle travel time measurements
- Area measurements derived from sections

The result attributes can have sub-attributes. You can then filter the display of data in the list.

1. From the **Lists** menu, choose **Results** and then select the desired result list:
   - Data Collection Results (see "Evaluating data collection measurements" on page 1093)
   - Delay Results (see "Showing delay measurements in lists" on page 1107)
   - Area Measurement Results (see "Evaluating pedestrian areas with area measurements" on page 1041)

   The result list opens.
2. Click on the **Attribute selection** icon (see "Selecting attributes and subattributes for columns of a list" on page 112).
3. Choose the desired attributes in the tree structure to the left (see "Selecting attributes and subattributes for columns of a list" on page 112).
4. If desired, click the **Filter** button and filter the data (see " Setting a filter for selection of subattributes displayed" on page 117).
5. If you have filtered data, confirm **Preselection Filter** window with **OK**.

*The result attributes from the simulation are shown in the results list (see "Evaluating pedestrian areas with area measurements" on page 1041), (see "Evaluating data collection measurements" on page 1093), (see "Showing delay measurements in lists" on page 1107). A column with result attributes is shown for each filtered sub-attribute. If you start the simulation again, the average, standard deviation, minimum and maximum values of the simulation data are also shown. You can save the result list to an attribute file *.att (see "List toolbar" on page 97).*

11.7 Configuring evaluations of the result attributes for lists

You can select and configure evaluations whose result attributes you want to show in attribute lists or result lists. The configuration is saved to the file **.inpx*.

Notes: Evaluation data of previous simulation runs are deleted:

- When you select the **Delete previous simulation runs** option from the **Result Management** tab, (see "Managing results" on page 1007)
- When you change the configuration of the evaluation
- When you edit or delete network objects the evaluation is based on

Before Vissim deletes the data, a message is displayed.

1. Ensure that the desired settings are defined for managing the result data (see "Managing results" on page 1007)
2. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
3. Select the desired evaluations in the **Collect data** column.
4. Make the desired changes:

Element	Description
Additionally collect data for these classes	➤ Vehicle classes ➤ Pedestrian Classes By default, the data for all vehicle classes and pedestrian classes is entered together. You can also show the data for certain vehicle classes and/or pedestrian classes separately in the evaluation.
Collect data	Select the desired evaluations before the simulation starts
From time	Simulation seconds in which the evaluation is started
To time	Simulation seconds in which the evaluation is finished
Interval	Duration of the evaluation intervals in which the data is aggregated
More	Evaluation-specific parameters. These are described in the individual evaluations: ➤ Vehicle travel times (see "Evaluating vehicle travel time measurements" on page 1096) ➤ Areas & Ramps (see "Evaluating pedestrian density and speed based on areas" on page 1034) ➤ Nodes (see "Evaluating nodes" on page 1057) ➤ Links (see "Showing data from links in lists" on page 1103) ➤ Queue counters (see "Showing results of queue counters in lists" on page 1105)

5. Confirm with **OK**.
6. Open the desired program elements in which you want to show the result attributes, for example, attribute lists or result lists of network object types (see "Displaying result attributes in attribute lists" on page 1017), (see "Showing result attributes in result lists" on page 1016).
7. If desired, show further columns for the desired attributes in the list.

Notes:

- In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).
- By default, the currently selected units for lengths, speeds, accelerations, decelerations are taken into account for the evaluation (see "Selecting network settings for units" on page 205). You can also choose the unit for attribute values in the attribute selection list in the **Format** column (see "Selecting attributes and subattributes for columns of a list" on page 112).
- Coordinates are always specified in [m].
- Units that are shown in the list box or in the column header with the respective attribute cannot be changed.

8. Start the simulation (see "Running a simulation" on page 840).

The data from the simulation is shown in the attribute lists or result lists.

Tip: After the simulation, you can save lists manually or automatically to an attribute file *.att* (see "List toolbar" on page 97).

11.7.1 Showing result attributes in result lists

Result lists show the values of the attributes **Simulation run** and **Time interval** as well as network object-specific attributes from the simulation. You can save result lists to an attribute file **.att* (see "List toolbar" on page 97).

If you perform multiple simulation runs, these are shown in succession in the result lists.

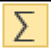
1. Configure the desired evaluation of result attributes (see "Configuring evaluations of the result attributes for lists" on page 1014).
2. Choose the desired entry from the menu **Evaluation** > **Result Lists**.

   The result list opens.

3. If you wish to change the selection of the attributes in the list, click on the **Attribute selection** icon (see "Selecting attributes and subattributes for columns of a list" on page 112).
4. Choose the desired attributes in the tree structure to the left (see "Selecting attributes and subattributes for columns of a list" on page 112).
5. If desired, click the **Filter** button and filter the data (see " Setting a filter for selection of subattributes displayed" on page 117).
6. If you have filtered data, confirm **Preselection Filter** window with **OK**.
7. Start the simulation (see "Running a simulation" on page 840).

   The data from the simulation is shown in the result list. You can switch between the open program elements, for example, between various lists and network editors. You can show *result attributes in result lists after a simulation, provided that you selected and configured the corresponding evaluation before the start.*

8. Make the desired changes (see "List toolbar" on page 97).

Element	Name	Description
	**Show Simulation Run Aggregates**	Shows the following aggregated values for some network object types in the result list or hides them: • Mean • Minimum • Maximum • Standard deviation • Percentile
	**Show Time Interval Aggregates**	Shows the aggregated values across all time intervals for some network object types in the result list or hides them, if several time intervals are defined.

11.7.2 Displaying result attributes in attribute lists

In the attribute list of a network object type, in one of the columns, you can display result attributes. The column lists the attribute values of the result attribute. For example, in the **Vehicles In Network** list, in the **Speed** column, you can display the current speed of the individual vehicles. The result attributes can have sub-attributes. You can then filter the display of data in the list. You can save lists to an attribute file **.att* (see "List toolbar" on page 97).

1. Configure the desired evaluation of result attributes (see "Configuring evaluations of the result attributes for lists" on page 1014).
2. From the **Lists** menu, choose the desired network object type in whose attribute list you want to show the result attributes.

   The Attribute list opens.

3. Click on the  **Attribute selection** icon.

   *The window **<Name Network object type>: Select Attributes** opens.*

4. If desired, click the **Filter** button and filter the data (see " Setting a filter for selection of subattributes displayed" on page 117).
5. If you have filtered data, confirm **Preselection Filter** window with **OK**.
6. Repeat the following steps for all attributes that you want to show in the attribute list.
7. In the section on the left, click the desired result attribute.
8. Click the icon .

   The attribute selected on the left is listed on the right in an additional row. You cannot edit hatched cells or the attribute name.

9. If desired, edit the value in a cell on the right (see "Selecting attributes and subattributes for columns of a list" on page 112).
10. Confirm with **OK**.

11. Start the simulation (see "Running a simulation" on page 840).

The data from the simulation is shown in the list. A column with result attributes is shown for each filtered sub-attribute. You can switch between the open program elements, for example, between various lists and network editors.

11.8 Configuring evaluations for direct output

You can select and configure evaluations which you want to save directly to a file or database. The configuration is saved to the file **.inpx*. Evaluations of raw data always include all attributes. You cannot select individual attributes.

11.8.1 Using the Direct output function to save evaluation results to files

Using Direct output, you can save evaluations to text files. The file format depends on the evaluation (see "Overview of evaluations" on page 1002). If you perform several simulation runs with evaluations in succession, text files are saved for the selected evaluations for each simulation run.

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. For the desired evaluation, select the option **Write to file**.
3. Make the desired changes:

Element	Description
From time	Simulation second in which the evaluation is started
To time	Simulation second in which the evaluation is finished
More	Evaluation-specific parameters. These are described in the individual evaluations.

4. Confirm with **OK**.

5. Start the simulation (see "Running a simulation" on page 840).

The data from the simulation is saved in the files (see "Output options and results of individual evaluations" on page 1023).

11.8.2 Configuring the database connection for evaluations

You must configure the database connection before you save evaluations in a database.

11.8.2.1 System prerequisites for database connections

For data export to Microsoft™ Access™, you need to install an appropriate OLE DB driver:

- Jet 4.0 OLE DB provider: driver for database files of the type (*.*mdb*) for **Access 2003**.
- ACE 12 provider (Microsoft Access Database Engine 2010 Redistributable): driver for database files of the type (*.*accdb*) for **Access 2007** and subsequent versions.

The installation of Microsoft™ Access™ itself is not absolutely necessary for the export.

- All SQL 2003-compliant databases are supported.
- The database connection to SQL is suitable and has been tested for the 64-bit Vis-simeditions.
- The database outputs have been tested with Microsoft™ Access™ and Oracle™.
- You can use the free MS SQL Server 2008 Express Edition. You require administrator rights for the installation:

11.8.2.2 Downlaoding Microsoft® SQL Server® 2008 Express

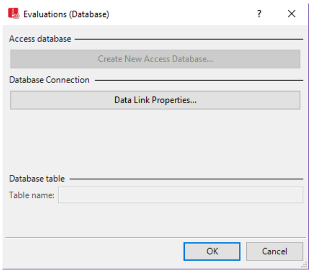
1. Open the website: http://www.microsoft.com.
2. Search for `SQL Server 2008 Express`.
3. Follow the instructions on the Internet page.

11.8.2.3 Creating new Access database

This is only necessary, if you want to create a new Access database. This creates a new database in the Microsoft Access format *.*mdb*. If you select a saved file with the same name, the file is overwritten.

1. From the **Evaluation** menu, choose > **Database Configuration**.

   *The **Evaluations (Database)** window opens.*

2. Check the version of Microsoft™ Access™ that is installed on your computer:
   - **Access 2003**: Install a JET 4.0 driver for database files of the type (*.*mdb*).
   - **Access from 2007**: Install an ACE 12 driver (Microsoft Access Database Engine 2010 Redistributable) for database files of the type (*.*accdb*).
3. Click the button **Create New Access Database...**.

   A window opens.

4. Enter the name of the desired Access™ database.

5. Click the **Save** button.

   *The **Data Link Properties** window opens. On the **Connection** tab, in the **Data source** box, the name of the Access™ database is displayed.*

6. Click the **Test connection** button.
7. Confirm with **OK**.

*In the **Evaluations (Database)** window, the OLE DB **Provider** is displayed. In addition to **Data source**, the name of the datbase is listed.*

11.8.2.4 Configuring data link properties

A database link string is created using the **Data link properties**. This establishes a database connection before the start of the simulation. A database connection can be established only to an existing database. The database connection string is saved encrypted to the **.inpx* file.

1. From the **Evaluation** menu, choose > **Database Configuration**.

   *The **Evaluations (Database)** window opens.*

2. Click the button **Data Link Properties**.

   *The **Data Link Properties** window opens.*

3. Define the settings in the tabs:

Tab	Description
Provider	Select desired provider from the list of installed database providers. Jet Provider and Oracle Provider, etc., were tested with Vissim.

Connection	The connection properties depend on the selected provider. Examples:
	➤ Access 2003 (Jet Provider):
	  ➤ **Database name**: Name of output file **.mdb*
	  ➤ **User name**: If no other name is required, you can accept the default entry.
	➤ Access 2007 (Access Database Engine OLE DB Provider):
	  ➤ **Data source**: Name of data source **.Accdb*
	  ➤ **Storage location**: name of data storage location
	➤ Oracle:
	  ➤ **Server name**: establishes connection to the Oracle server
	  ➤ **User name**: You user name
	  ➤ **Password**: the specified password is saved unencrypted with the Vissim network file.
	  ➤ Select option **Allow saving of password**

Advanced	Provider-specific properties. You can accept the default entries.
All	Provider-specific properties. You can accept the default entries.

Notes: Vissim transfers SQL statements with column identifiers in double quotation marks to the database.

Some ODBC drivers provide the database connection with the option **Use ANSI quotes (double quotation marks)**. If this option does not exist for a database connection, you can use a database that is configured to accept double quotation marks.

Examples:

- SQL Server (Microsoft SQL Server ODBC Driver Version 6.01): Select the option **Use ANSI-quoted identifiers**.
- MySql: Enter the parameter **sql_mode** in **ANSI_QUOTES** either as a start parameter or in the configuration file.

4. Confirm with **OK**.

11.8.3 Saving evaluations in databases

You can use the direct output to save the following evaluations in the **.mdb* file format of SQLite databases (see "Overview of evaluations" on page 1002):

- Vehicle record (see "Saving vehicle record to a file or database" on page 1031)
- Vehicle travel times (raw data) (see "Evaluating vehicle travel time measurements" on page 1096)
- Pedestrian record (see "Saving pedestrian record to a file or database" on page 1053)
- Nodes (raw data) (see "Evaluating nodes" on page 1057)
- Signal changes (see "Evaluating signal changes" on page 1081)

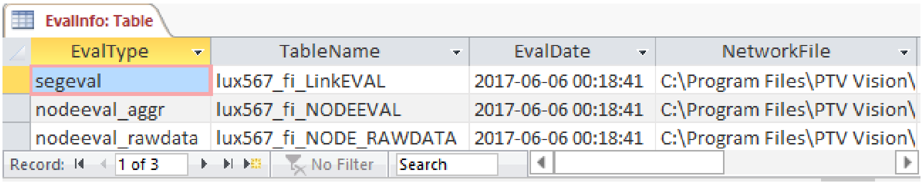
1. Ensure that the database connection is configured (see "Configuring the database connection for evaluations" on page 1018).
2. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
3. For the desired evaluation, select the option **Write database**.
4. Make the desired changes:

Element	Description
From time	Simulation second in which the evaluation is started
To time	Simulation second in which the evaluation is finished
More	Evaluation-specific parameters. These are described in the individual evaluations.

5. Confirm with **OK**.

6. Start the simulation (see "Running a simulation" on page 840).

*The *.mdb file is saved. A table **EvalInfo** is created for the evaluations in the database. The table contains the evaluations with the user-defined name of the table in the **Table name** column. If you wish to execute multiple simulation runs, the data from the simulation runs is saved only in a database table. In the attribute list, multiple columns with the same title name are merged to one column.*


EvalInfo: Table

EvalType	TableName	EvalDate	NetworkFile
segeval	lux567_fi_LinkEVAL	2017-06-06 00:18:41	C:\Program Files\PTV Vision\
nodeeval_aggr	lux567_fi_NODEEVAL	2017-06-06 00:18:41	C:\Program Files\PTV Vision\
nodeeval_rawdata	lux567_fi_NODE_RAWDATA	2017-06-06 00:18:41	C:\Program Files\PTV Vision\

Record: 1 of 3 No Filter Search

Note: If you wish to execute multiple simulation runs and to save the results in a database table, avoid switching the selection of attributes between simulation runs. Otherwise, the table with the previous data simulation runs will be deleted.

*Data from the simulation run are saved after the simulation end to the directory ..\<Name of network file>.results in the default data format *.db.*

11.9 Showing evaluations in windows

You can show the following evaluation windows (see "Overview of evaluations" on page 1002):

- Signal times table
- SC detector record
- Signal changes

SCs must be defined in the network (see "Modeling signal controllers" on page 577).

1. From the **Evaluation** menu, choose > **Windows**.
2. Select the desired evaluation.

   A window opens.

3. Make the desired changes:
   - Signal times table: Specifying display settings for signal times table (see "Showing signal times table in a window" on page 1098)
   - SC detector record: Showing the evaluation of an SC detector record in a window (see "Evaluating SC detector records" on page 1070)
   - Signal changes (see "Evaluating signal changes" on page 1081)
4. Start the simulation (see "Running a simulation" on page 840).

11.10 Importing text file in a database after the simulation

You can use the direct output to save data to a text file during the simulation. You can import this text file in a database after the simulation.

Saving data in a database during the simulation can affect the speed of the simulation.

Example: Importing a **.fzp* file into MS Access 2007

1. Configure the direct output for the vehicle record and select **Write to file**.
2. Open the file **.fzp* using a text editor.
3. Delete all the data above the data block in the file.
4. Save the file with the extension **.txt*.
5. In Access, choose the menu **File** > **New**.
6. Create an empty database.
7. Import the text file using the menu **File** > **Import External Data**.

Note: Make sure that you have selected **Text files** in the **Import** file type window.

8. Click on **Import**.

   *The **Text import Wizard** opens.*
9. Select the option **With Separators**.
10. Click on **Next**.
11. Select the option **Semicolon**.
12. Select the option **First row contains field names**.
13. Click on **Next**.
14. Select the option **In a new table**.
15. Click on **Next**.
16. From the list box for each column, select the correct **data type** double, integer or text.
17. Click on **Next**.
18. Select the option **No primary key**.
19. Click on **Next**.
20. Click on **Finish**.

11.11 Output options and results of individual evaluations

Depending on the selected evaluation, the type of data and the desired further use, you can show evaluations in list and/or windows and save them to text files and/or database files (see

"Overview of evaluations" on page 1002).

Note: Some output options and individual evaluation results may only be available with add-on modules. You need a license to use the add-on modules.

11.12 Visualizing evaluation results

You have additional options for visualizing evaluation results:

- Color schemes for links (see "Assigning a color to nodes based on an attribute" on page 191)
- Link bars (see "List of graphic parameters for network objects" on page 161), (see "Attributes of links" on page 409)
- Turn value visualization (see "Visualizing turn values" on page 685)
- Path volume visualization using flow bundles (see "Visualizing volumes on paths as flow bundles" on page 766)

11.13 Saving discharge record to a file

The discharge record shows the time intervals between vehicles driving away from signal heads. You can use it for purposes such as determining time needs for your SC or calibrating actual measured saturation flow values. Links, vehicle inputs and signal heads must be defined in the network.

In order to determine meaningful time needs, the measured flows should be saturated. You should therefore ensure that the number of vehicles queued behind the stop line is at least as many as can pass during a green phase.

You can output the following data and data formats:

Output	ASCII file	MDB file	Attribute file from attribute list
Aggregated data	-	-	-
Raw data	*.dis	-	-

Defining evaluation of the discharge record results

1. Make sure that the attribute **Discharge record active** is selected in the attributes of the desired signal head.
2. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
3. In the **Discharge record** row, select the option **Write to file**.
4. You can change the time if you wish (see "Using the Direct output function to save evaluation results to files" on page 1018).
5. Confirm with **OK**.
6. Start the simulation (see "Running a simulation" on page 840).

*A *.dis file is saved.*

Result of discharge record evaluation

A **.dis* discharge record may contain the following data:

Element	Description
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
PTV Vissim	Version number, service pack number, build number
Table	Discharge at SC with its number, signal group with its number, and in parentheses data collection point with number

Data block	One line for each SC cycle
	- Column 1: Simulation seconds at start of green time
	- Column 2: Time difference between start of green time and arrival of first vehicle at measuring point
	- Column 3: Time need: time gap between the front end of vehicle 1 (the first vehicle in the queue) and the front end of vehicle 2
	- All other columns contain the time needs of all subsequent vehicles according to their positions in the queue.
	- The end of the green phase is shown by the position of the parentheses. The numbers in parentheses show:
	  - the number of vehicles passing the measuring point during the green phase
	  - the average time needs of these vehicles. Both values are determined without taking vehicle 1 into account, because its time needs depend on the distance between the stop line and the measuring point.
	- Values after the parentheses were measured for vehicles that crossed the stop line after the green phase (during yellow or red).
	- Fourth-to-last line: vehicle position number in the queue (index number) for each cycle
	- Third-to-last line: average time need for the vehicle position concerned
	- Next-to-last line: measured number of vehicles at this position. If saturation is not present for some green phases, smaller numbers are measured for higher index numbers.
	- Last line: total number of vehicles (except those in the first position) and their time needs (measured over all cycles).

Discharge record example: file **.dis*

```
Discharge record
File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Demo\3D - Com-
plex Intersection Karlsruhe.DE\Karlsruhe 3D.inpx
Comment:  Karlsruhe 3D example
```

```
Date:     Monday, June 18, 2018 10:20:30 AM
PTV Vissim 11.00-00* [66098]
Discharge at SC 619, signal group 14, signal head 3
1   (0: 0.0)
80   1.11   1.16   (1: 1.16)
170   1.01   (0: 0.0)
260   (0: 0.0)
350   0.95   0.70   (1: 0.70)
440   1.07   1.31   (1: 1.31)
530   1.09   (0: 0.0)
620   1.05   (0: 0.0)
710   1.17   (0: 0.0)
800   1.15   1.45   (1: 1.45)
890   1.09   (0: 0.0)
980   1.11   (0: 0.0)
1070   (0: 0.0)
1160   0.99   0.25   0.88   (2: 0.56)
1250   (0: 0.0)
1340   1.07   (0: 0.0)
1430   (0: 0.0)
1520   6.95   (0: 0.0)
1610   1.18   1.35   (1: 1.35)
1700   1.18   0.98   (1: 0.98)
1790   1.07   (0: 0.0)
1880   1.06   0.05   1.00   0.11   1.05   1.18   (5: 0.68)
1970   1.11   (0: 0.0)
2060   1.06   1.09   (1: 1.09)
2150   1.08   (0: 0.0)
2240   1.10   1.10   1.08   3.49   (3: 1.89)
2330   1.04   (0: 0.0)
2420   1.09   0.99   (1: 0.99)
2510   1.04   (0: 0.0)
2600   1.13   1.11   (1: 1.11)
2690   (0: 0.0)
2780   (0: 0.0)
2870   1.07   (0: 0.0)
2960   (0: 0.0)
3050   (0: 0.0)
3140   (0: 0.0)
3230   (0: 0.0)
3320   1.11   0.00   1.09   0.18   0.75   0.83   (5: 0.57)
3410   1.06   (0: 0.0)
3500   (0: 0.0)
3590   (0: 0.0)
3680   1.17   (0: 0.0)
3770   1.07   1.47   (1: 1.47)
3860   1.22   (0: 0.0)
3950   1.03   6.26   (1: 6.26)
4040   1.13   (0: 0.0)
4130   1.19   1.16
-----      1      2      3      4      5      6
-----   1.26   1.28   1.01   1.26   0.90   1.00
-----     34     16      4      3      2      2
[27: 1.19]
```

11.14 Displaying OD pair data in lists

In lists, you can show result attributes that are created from traffic data between the origin zones and destination zones of dynamic assignment.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	**ASCII file**	**MDB table**	**Attribute file from attribute list**
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Results of the evaluation of OD pairs

The results list **OD Pair Results** contains the following attributes:

Result attribute Long name	**Short name**	**Description**
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
ODPair	**OD pair**	Zone numbers specify the OD pairs between origin zones, destination zones and origin and destination zones.
Aggregated by departure time:		
TravTmDep	**Travel time (departure)**	Average travel time = *Total of travel times / number of vehicles*
DelayTmDep	**Delay time (departure)**	Average delay time = *Total of delay times / number of vehicles*
DelayRelDep	**Delay (relative) (departure)**	Average relative delay = *Average delay time / average travel time*
VolumeDep	**Volume (departure)**	Number of vehicles
DistTravDep	**Distance traveled (departure)**	*Total distance traveled / number of vehicles*

Result attribute Long name	Short name	Description
Aggregated by arrival time:		
TravTmArr	**Travel time (arrival)**	*Total travel time / number of vehicles*
DelayTmArr	**Delay time (arrival)**	*Total of delay times / number of vehicles*
DelayRelArr	**Delay (relative) (arrival)**	*Average delay time / average travel time*
VolArr	**Volume (arrival)**	Number of vehicles
DistTravArr	**Distance traveled (arrival)**	*Total distance traveled / number of vehicles*

The result attributes are based on the destination parking lot and thus on the destination zone that a vehicle reaches during simulation and where it is then removed from the network. The result attributes are not based on the destination parking lot that is the vehicle's destination at the beginning of the simulation.

11.15 Saving lane change data to a file

You can record the time and place of the lane change that was made. You can restrict the evaluation to vehicle classes and individual vehicles.

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	-
Raw data	*.spw	-	-

Note: This evaluation always uses metric units.

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Lane changes** row select the option **Write to file**.
3. You can change the time if you wish (see "Using the Direct output function to save evaluation results to files" on page 1018).
4. Click the **More** button.

   *The **Lane changes** window opens.*
5. Make the desired changes:

Element	Description
Vehicle filter	- **All vehicles**: takes all vehicles into account in the evaluation - **Filter by classes**: takes only the selected vehicle classes into account - **Filter by sections**: takes only the selected sections into account Sections have to be defined (see "Modeling sections" on page 677). The level with the section and the the level on which vehicles move are the same. The middle of the vehicle's front edge lies outside the section. - **Filter by individual vehicles**: ☑ If this option is selected, you can right-click the list and select vehicles by their number. Only the selected vehicles are taken into account.

6. Confirm with **OK**.
7. Start the simulation (see "Running a simulation" on page 840).

*A *.spw file is saved.*

Result of lane change evaluation

The lane change file **.spw* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
PTV Vissim	Version number, service pack number, build number
Data	Data block with the attributes for each recorded vehicle

The data block of the evaluation file contains the following parameters:

Column	Description
t	Starting time of the lane change. **t** is the end of the time step in which the lane change starts. If you simulate with only one time step per simulation second, the lane change is already running for a second at this time.
VehNr	Vehicle number
v [m/s]	Speed [m/s]
Link No.	Link number
Lane	Number of the old lane
New Lane	Number of the new lane
VF	Vehicle number of old leading vehicle (0 = not available)
v VF	Speed [m/s] of old leading vehicle
dv VF	Speed difference [m/s] of old leading vehicle

Column	Description
dx VF	old leading vehicle: distance [m] between rear end position of the preceding vehicle and front end of the trailing vehicle
VB	Vehicle number of old trailing vehicle (0 = not available)
v VB	Speed [m/s] of old trailing vehicle
dv VB	Speed difference [m/s] of old trailing vehicle
dx VB	old trailing vehicle: distance [m] between rear end position of the preceding vehicle and front end of the trailing vehicle
new VF	Vehicle number of new leading vehicle (0 = not available)
v new VF	Speed [m/s] of leading vehicle
dv new VF	Speed difference [m/s] of new leading vehicle
dx new VF	new leading vehicle: distance [m] between rear end position of the preceding vehicle and front end of the trailing vehicle
new VB	Vehicle number of new trailing vehicle (0 = not available)
v new VB	Speed [m/s] of new trailing vehicle
dv new VB	Speed difference [m/s] of new trailing vehicle
dx new VB	new trailing vehicle: distance [m] between rear end position of the preceding vehicle and front end of the trailing vehicle

Example of evaluation of lane change file *.spw*

```
Lane changes record

File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Demo\3D - Com-
plex Intersection Karlsruhe.DE\Karlsruhe 3D.inpx
Comment:  Example, SC 3-10

Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

t; VehNr; v [m/s]; Link No.; Lane; New Lane;
VF; v VF [m/s]; dv VF [m/s]; dx VF[m]; VB; v VB; dv VB [m/s]; dx VB;
new VF; v new VF; dv new VF; dx new VF; new VB; v new VB; dv new VB; dx new VB
115.10; 203; 9.55; 7;2; 1;
 175; 0.00; 9.55; 164.28; 0; -1.00; -1.00; -1.00;
  198; 12.43; -2.89; 22.37; 0; -1.00; -1.00; -1.00;
128.50; 39; 13.30; 9;3; 2;
 48; 12.77; 0.53; 11.31; 0; -1.00; -1.00; -1.00;
 10; 12.66; 0.64; 9.11; 98; 9.19; 4.10; 0.58;
130.70; 40; 13.56; 9;3; 2;
 48; 13.11; 0.45; 34.45; 47; 13.02; 0.54; 13.28;
  98; 11.51; 2.05; 6.39; 57; 9.53; 4.03; 0.58;
136.50; 180; 12.04; 9;2; 1;
 68; 10.39; 1.65; 34.99; 0; -1.00; -1.00; -1.00;
  102; 12.56; -0.52; 31.48; 105; 15.16; -3.13; 26.58;
141.80; 88; 12.00; 9;3; 2;
```

```
 66; 12.20; -0.19; 18.24; 0; -1.00; -1.00; -1.00;
 58; 10.29; 1.71; 8.42; 115; 12.52; -0.52; 17.11;
144.20; 115; 11.25; 9;2; 1;
 88; 9.80; 1.46; 15.14; 134; 12.27; -1.02; 2.95;
  117; 10.61; 0.64; 25.67; 140; 14.61; -3.36; 105.09;
144.70; 134; 11.73; 9;2; 1;
 115; 11.06; 0.66; 2.51; 0; -1.00; -1.00; -1.00;
 117; 10.20; 1.53; 31.93; 140; 14.53; -2.80; 96.74;
152.20; 272; 8.64; 7;1; 2;
 203; 0.00; 8.64; 156.60; 0; -1.00; -1.00; -1.00;
 265; 12.84; -4.19; 41.51; 0; -1.00; -1.00; -1.00;
164.90; 293; 8.70; 7;1; 2;
 203; 0.00; 8.70; 156.69; 0; -1.00; -1.00; -1.00;
 272; 7.72; 0.99; 131.91; 0; -1.00; -1.00; -1.00;
174.80; 293; 11.48; 7;2; 1;
 272; 0.00; 11.48; 28.49; 0; -1.00; -1.00; -1.00;
 203; 0.00; 11.48; 39.70; 296; 12.99; -1.51; 17.72;
215.20; 183; 10.61; 9;3; 2;
 194; 13.61; -2.99; 21.95; 0; -1.00; -1.00; -1.00;
 204; 12.34; -1.73; 0.64; 165; 10.61; 0.00; 9.79;
…
…
```

11.16 Saving vehicle record to a file or database

The vehicle record outputs the attribute values for each vehicle as raw data in one row per time step. You can restrict the evaluation to vehicle classes and individual vehicles. Links and vehicle inputs must be defined in the network (see "Modeling links for vehicles and pedestrians" on page 406), (see "Modeling vehicle inputs for private transportation" on page 454).

The vehicle record includes link data, if for links the attribute **vehicle record** ☑ has been selected (see "Attributes of links" on page 409).

You can output the following data and data formats:

Output	ASCII file	MDB file	Attribute file from attribute list
Aggregated data	-	☑	-
Raw data	*.fzp	-	-

Tip: You can show many output attributes of the **Vehicle record** in parallel, also during the simulation, as a result list **Vehicles In Network** (see "Displaying vehicles in the network in a list" on page 847).

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Vehicle record** row, select the option **Write to file**.
3. You can change the time if you wish (see "Using the Direct output function to save evaluation results to files" on page 1018).

4. Click the **More** button.

   *The **Vehicle record** window opens.*

5. Make the desired changes:

Element	Description
Resolution	Simulation resolution of time steps
Including parked vehicles	Only with the add-on module **Dynamic Assignment**. ☑ If this option is selected, vehicles in parking lots are also taken into account in the evaluation.
Vehicle filter	➤ **All vehicles**: takes all vehicles into account in the evaluation ➤ **Filter by classes**: takes only the selected vehicle classes into account ➤ **Filter by sections**: takes only the selected sections into account Sections have to be defined (see "Modeling sections" on page 677). The level with the section and the level on which vehicles move are the same. The middle of the vehicle's front edge lies outside the section. ➤ **Filter by individual vehicles**: ☑ If this option is selected, you can right-click the list and select vehicles by their number. Only the selected vehicles are taken into account.
Attribute selection	The **Vehicles In Network: Select Attributes** window opens. You can select attributes for the evaluation (see "Selecting attributes and subattributes for columns of a list" on page 112). For import of the **.fzp* file into the Autodesk 3DS MAX plug-in Civil View, the attributes and subattributes in the following table are relevant. Do not change the default settings specified in the **Decimals** column and the **Format** column, if you choose to import these attributes and subattributes.

Relevant attributes and subattributes for Autodesk 3DS MAX plug-in Civil View:

Attribute\subattribute	**Default value** of column	
	Decimals	**Format**
Simulation second	2	**Default**
Coordinates front	3	**Default**
Coordinates rear	3	**Default**
Vehicle type\Name	0	**Default**
if **Vehicle type\Name** does not exist, Autodesk 3DS MAX Civil View looks for **Vehicle type\Number** and then **Vehicle type**:		
Vehicle type\Number	0	**Default**
Vehicle type	0	**Default**

6. Confirm with **OK**.
7. Start the simulation (see "Running a simulation" on page 840).

*A *.fzp file is saved.*

Notes:

- Stops at PT stops and in parking lots are not counted as stops for the **Number of stops** and **Delay time** in the vehicle record.
- You can filter and record the data of all vehicles to determine the total values of delays or travel time in the network. This is the maximum per vehicle before leaving the network. At the same time, the data of the vehicles that remain after the simulation in the network must be recorded.

Result of vehicle record evaluation

A vehicle record **.fzp* may contain the following data:

Element	Description
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
PTV Vissim	Version number, service pack number, build number
Table	Vehicles In network (see "Displaying vehicles in the network in a list" on page 847)
Column names	Column headers and units
Data block	One row for each vehicle number. The columns correspond to the sequence of the attributes in the selection. The corresponding add-on module must be installed for some of the listed attributes so that correct values can be recorded, for example, the add-on module **Dynamic Assignment**.

Notes:

- By default, the currently selected units for lengths, speeds, accelerations, decelerations are taken into account for the evaluation (see "Selecting network settings for units" on page 205). You can also choose the unit for attribute values in the attribute selection list in the **Format** column (see "Selecting attributes and subattributes for columns of a list" on page 112).
- Coordinates are always specified in [m].
- Units that are shown in the list box or in the column header with the respective attribute cannot be changed.

Your selection of attributes determines which attribute values are included. You may also display these result attributes in the **Vehicles In Network** list (see "Displaying vehicles in the network in a list" on page 847).

Vehicle record example: file **.fzp*

```
$VISION
* File: C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Demo\3D - Complex
Intersection Karlsruhe.DE\Karlsruhe 3D.inpx
* Comment:
*
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]
*
*Table: Vehicles In Network
* SIMSEC: SimSec, simulation second (simulation time [s]) [s]
* NO: No, Number
* LANE\LINK\NO: Lane\Link\Number
* LANE\INDEX: Lane\Index
* POS: Position [m]
* POSLAT: Position (lateral)
*
$VEHICLE:SIMSEC;NO;LANE\LINK\NO;LANE\INDEX;POS;POSLAT
0.20;1;1;1;2.57;0.50
0.40;1;1;1;5.71;0.50
0.40;3;2;1;0.22;0.50
0.60;1;1;1;8.86;0.50
0.60;3;2;1;3.25;0.50
0.80;1;1;1;12.02;0.50
0.80;3;2;1;6.29;0.50
1.00;1;1;1;15.18;0.50
1.00;3;2;1;9.33;0.50
...
```

11.17 Evaluating pedestrian density and speed based on areas

You can determine the density and speed of pedestrians and show them, together with other attributes, in the result lists **Area Results** and **Ramps Results**.

In addition, you have the following options to evaluate pedestrian density and speed:

- grid-based (see "Grid-based evaluation of pedestrian density and speed" on page 1037).
- for each individual pedestrian In this case, density is based on the pedestrians that at the end of an evaluation interval are located within a radius around a pedestrian (see "Showing pedestrians in the network in a list" on page 853).

Density and speed are determined for the following network objects:

- Pedestrian Areas
- Ramps & Stairs
- Links, whose attribute **Is pedestrian area** (option **Use as pedestrian area**) is selected

by default, the data for all pedestrian classes is entered together. You can also show the data for certain pedestrian classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Areas & ramps** row, select **Collect data**.
3. If desired, change the time and/or the interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Confirm with **OK**.

5. Start the simulation (see "Running a simulation" on page 840).
6. If desired, from the **Evaluation** menu, choose > **Result Lists** > **Area Results** or **Ramp Results** (see " Output attributes of area and ramp evaluation" on page 1039).

Attributes of aggregated data output

Result attribute Long name	Short name	Description
Number of pedestrians (maximum)	**NumPedsMax**	Maximum number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians (minimum)	**NumPedsMin**	Minimum number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians (average)	**NumPedsAvg**	Average number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians waiting for PT (maximum)	**NumPedsWaitingPTMax**	Maximum number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs
Number of pedestrians waiting for PT (minimum)	**NumPedsWaitingPTMin**	Minimum number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs

Result attribute Long name	Short name	Description
Number of pedestrians waiting for PT (average)	**NumPedsWaitingPTAvg**	Average number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs
Walk-out count	**WalkOutCnt**	Number of pedestrians leaving the construction element Pedestrians from pedestrian inputs and pedestrians alighting from PT vehicles are not counted.
Density	**Density**	Pedestrian density in area, on ramp or stairs
Walk-in count	**WalkInCnt**	Number of pedestrians walking on construction element Pedestrians from inputs and pedestrians alighting from PT vehicles are not counted.
Density experienced	**DensityExp**	Pedestrian density experienced within the perception radius of a pedestrian: Number of other pedestrians within a radius around the pedestrian.
Area	**Area**	in **Area evaluation results** list only: area number
Speed **All types**	**Speed(All)**	average pedestrian speed, all pedestrian types, is calculated as the harmonic mean
Speed variance	**SpeedVar**	Vectorial speed differences of all pedestrians within the personal environment radius of their own speed (see "Selecting network settings for pedestrian behavior" on page 204)
Ramp/Stairs	**Ramp**	in **Ramp evaluation results** list only: ramp or stairs number
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
If for areas, you selected the **Queues** attribute, you may additionally output the following result attributes via queues:		

Result attribute Long name	Short name	Description
Queue length (maximum)	**QueueLenMax**	Length and time information on the queues
Queue length (minimum)	**QueueLenMin**	
Queue length (average)	**QueueLenAvg**	
Time in queue (maximum)	**TmInQueueMax**	
Time in queue (minimum)	**TmInQueueMin**	
Time in queue (average)	**TmInQueueAvg**	

11.18 Grid-based evaluation of pedestrian density and speed

Note: Grid-based evaluations requires a lot of memory. Make sure that approximately 1 GB of memory space is available.

You can determine the density and speed of pedestrians based on grid cells and show them, together with other attributes, in the result lists **Area Results** and **Ramps Results**. This way, the result data for grid-based visualization of aggregated parameters of pedestrian simulation can also be used after a simulation run. It, for instance, allows you to calculate, show and save an average level-of-service image of multiple simulation runs. Density and speed data form the basis of the LOS color scheme (see "Using LOS schemes for showing aggregated pedestrian values" on page 186), (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190). In addition, you have the following options to evaluate pedestrian density and speed.

- area-based (see "Evaluating pedestrian density and speed based on areas" on page 1034).
- for each individual pedestrian In this case, density is based on the pedestrians that at the end of an evaluation interval are located within a radius around a pedestrian (see "Showing pedestrians in the network in a list" on page 853).

Density and speed are determined for the following network objects:

- Pedestrian Areas
- Ramps & Stairs
- Links, whose attribute **Is pedestrian area** (option **Use as pedestrian area**) is selected

by default, the data for all pedestrian classes is entered together. You can also show the data for certain pedestrian classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Pedestrian Grid Cells** row, select **Collect data**.

   *Select the **Collect data** option to enable the recording of result attributes. If your next steps are to click the **More** button, open the **Pedestrian Grid Cells** window and select **Only last interval**, the data will not be saved and will therefore not be available outside of the simulation run, but only during the simulation run for visualization of the data in a color scheme.*
3. If desired, change the time and/or the interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The **Pedestrian Grid Cells** window opens.*
5. Make the desired changes:

Element	Description
Only last interval	☑ If the option is selected, evaluation of the pedestrian-grid cells is performed exclusively based on the last time interval completed. The values of the result attributes obtained from the last time interval can be visualized in a color scheme for areas and/or ramps & stairs. The values are not stored and will therefore not be available outside of the simulation.
Cell size	Edge length of a grid mesh. Value range 0.01 to max [m], default value 1.00 m
Range of influence	Number of pedestrian-grid cells in each direction (up, down, left, right, diagonal) up until which pedestrians are considered for density measurement. The resulting area created around each pedestrian-grid cell is used by Vissim for the calculation of average LOS. Value range 1 - 5

Element	Description
Threshold	For the following attributes, Vissim records for how long they exceed the threshold. You can enter a threshold value for each attribute. For each attribute, you can show the duration recorded during the simulation graphically in a color scheme of the area or the ramp. This is also possible for a link, if the attribute **Is pedestrian area** is selected for it (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182), (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190), (see "Assigning a color to links based on aggregated parameters" on page 179).
	Density (**DensThresh**): Threshold value for evaluation attribute **Density - duration exceeding threshold**
	Experienced density (**ExperDensThresh**): Threshold value for evaluation attribute **Experienced density - duration exceeding threshold**
	Speed (**SpeedThresh**): Threshold value for evaluation attribute **Speed - duration exceeding threshold**
	Velocity variance: (**VelVarThresh**): Threshold value for evaluation attribute **Velocity variance - duration exceeding threshold**
Filter by sections:	Select the sections for which you want to collect data. If no sections are selected, data is collected for the entire network. Select sections when you want to record and save grid-based evaluations.for large models, small grid cells and many time intervals. Otherwise, if data is then collected for the entire network, your evaluation file will be very large. If a cell is only partly located within a section, the actual area of the cell is used for the calculation of area-based evaluations (density).

6. Confirm with **OK**.
7. Configure the visualization of grid-based data (see "Assigning a color to areas based on aggregated parameters (LOS)" on page 182) (see "Assigning a color to ramps and stairs based on aggregated parameters (LOS)" on page 190).
8. Start the simulation (see "Running a simulation" on page 840).
9. If desired, from the **Evaluation** menu, choose > **Result Lists** > **Area Results** or **Ramp Results** (see " Output attributes of area and ramp evaluation" on page 1039).

11.19 Output attributes of area and ramp evaluation

1. From the **Evaluation** menu, choose > **Result Lists** > **Area Results** or **Ramp Results**.

Result attribute Long name	Short name	Description
Number of pedestrians (maximum)	**NumPedsMax**	Maximum number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians (minimum)	**NumPedsMin**	Minimum number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians (average)	**NumPedsAvg**	Average number of pedestrians that were in the area, on ramp or stairs
Number of pedestrians waiting for PT (maximum)	**NumPedsWaitingPTMax**	Maximum number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs
Number of pedestrians waiting for PT (minimum)	**NumPedsWaitingPTMin**	Minimum number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs
Number of pedestrians waiting for PT (average)	**NumPedsWaitingPTAvg**	Average number of pedestrians who were waiting for a PT vehicle in the area, on the ramp or stairs
Walk-out count	**WalkOutCnt**	Number of pedestrians leaving the construction element Pedestrians from pedestrian inputs and pedestrians alighting from PT vehicles are not counted.
Density	**Density**	Pedestrian density in area, on ramp or stairs
Walk-in count	**WalkInCnt**	Number of pedestrians walking on construction element Pedestrians from inputs and pedestrians alighting from PT vehicles are not counted.
Density experienced	**DensityExp**	Pedestrian density experienced within the perception radius of a pedestrian: Number of other pedestrians within a radius around the pedestrian.
Area	**Area**	in **Area evaluation results** list only: area number
Speed All types	**Speed(All)**	average pedestrian speed, all pedestrian types, is calculated as the harmonic mean

Result attribute Long name	Short name	Description
Speed variance	**SpeedVar**	Vectorial speed differences of all pedestrians within the personal environment radius of their own speed (see "Selecting network settings for pedestrian behavior" on page 204)
Ramp/Stairs	**Ramp**	in **Ramp evaluation results** list only: ramp or stairs number
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
If for areas, you selected the **Queues** attribute, you may additionally output the following result attributes via queues:		
Queue length (maximum)	**QueueLenMax**	Length and time information on the queues
Queue length (minimum)	**QueueLenMin**	
Queue length (average)	**QueueLenAvg**	
Time in queue (maximum)	**TmInQueueMax**	
Time in queue (minimum)	**TmInQueueMin**	
Time in queue (average)	**TmInQueueAvg**	

11.20 Evaluating pedestrian areas with area measurements

You can define or generate area measurements that record data of pedestrian areas with sections. You can display and store result attributes in evaluations and lists. To allow an area measurement to record data, the following conditions must be satisfied:

- At least one section must be defined in the network (see "Modeling sections" on page 677). If sections are placed on top of pedestrian areas, they can record pedestrian area data.
- At least one area measurement must be defined or generated and assigned to at least one section (see "Defining an area measurement in lists" on page 1009), (see "Generating area measurements in lists" on page 1010).
- At least one pedestrian input and one pedestrian route must be defined in the pedestrian area (see "Modeling pedestrian inputs" on page 936), (see "Modeling routing decisions and routes for pedestrians" on page 939).

On a ramp, you can collect data from an area measurement with a section. To do so, in the **Additional Level** (**AddLvl**) attribute of the section, you must select the level leading to the ramp (see "Attributes of sections" on page 679).

by default, the data for all pedestrian classes is entered together. You can also show the data for certain pedestrian classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

Saving results of area measurements

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	*.merP	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)
- Save result attributes to files (see "Using the Direct output function to save evaluation results to files" on page 1018)
- Show result attributes of area measurements in list (see "Showing results of measurements" on page 1014)

Attributes of aggregated data output

Maximum, minimum, average all refer to time interval values.

Result attribute Long name	Short name	Description
Leave time (maximum), (minimum), (average)	**tLeavMax, tLeavMin, tLeavAvg**	first, last, and average point in time all pedestrians leave the sections [simulation second]
Orientation x	**OrientXAvg**	Average of the x component of the orientation vectors
Orientation y	**OrientYAvg**	Average of the y component of the orientation vectors
Walk-out count	**WalkOutCnt**	Number of pedestrians who have left sections This does not include passengers boarding PT vehicles.
Density (maximum), (minimum), (average)	**DensMax, DensMin, DensAvg**	Pedestrian density in sections

Result attribute Long name	Short name	Description
Density experienced (maximum), (minimum), (average)	**DensityExp**	Pedestrian density experienced within the perception radius of a pedestrian
Entry time (maximum), (minimum), (average)	**tEntMax, tEntMin, tEntAvg**	first, last, and average point in time all pedestrians reach the sections [simulation second]
Walk-in count	**WalkInCnt**	Number of pedestrians walking in the sections. Pedestrians from inputs and pedestrians alighting from PT vehicles are not counted.
Area measurements	**AreaMeasurement**	Name and number of area measurement
Pedestrians (maximum), (minimum), (average)	**PedsMax, PedsMin, PedsAvg**	maximum, minimum number of vehicles in section, average number
Total distance (maximum), (minimum), (average)	**TotDistMax, TotDistMin,TotDistAvg**	maximum, minimum and average total distance traveled in sections of all pedestrians who have left the sections during the aggregation interval
Total time gain (average)	**TotTmGainAvg**	average total time delay in sections for pedestrians who have left the sections during the aggregation interval
Total delay (maximum), (minimum), (average)	**TotalDelay**	maximum, minimum and average total time delay in sections for pedestrians who have left the sections during the aggregation interval
Total dwell time (maximum), (minimum), (average)	**TotDwlTmMax, TotDwlTmMin, TotDwlTmAvg**	maximum, minimum and average total dwell time in sections of all pedestrians who have left the sections during the aggregation interval
Speed (maximum), (minimum), (average)	**SpeedMax, SpeedMin, SpeedAvg**	maximum, minimum and average speed
Speed x-component (maximum), (minimum), (average)	**SpeedXMax, SpeedXMin, SpeedXAvg**	maximum, minimum, and average speed of x-component of speed vector

Result attribute Long name	Short name	Description
Speed y-component (maximum), (minimum), (average)	**SpeedYMax, SpeedYMin, SpeedYAvg**	maximum, minimum, and average speed of y-component of speed vector
Speed deviation (average)	**SpeedDevAvg**	average deviation of pedestrian speeds
Source quantity	**SourceQu**	Number of pedestrians walking in the sections. This also includes pedestrians from inputs and pedestrians alighting from PT vehicles.
Desired speed (average)	**DesSpeedAvg**	average desired speed of all pedestrians
World coordinate x (maximum), (minimum), (average)	**WorldXMax, WorldXMin WorldXAvg,**	maximum, minimum and average world coordinate x
World coordinate y (maximum), (minimum), (average)	**WorldYMax, WorldYMin, WorldYAvg**	maximum, minimum and average world coordinate y
World coordinate z (maximum), (minimum), (average)	**WorldZMax, WorldZMin, WorldZAvg**	maximum, minimum and average world coordinate z
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
Destination count	**DestCnt**	Number of pedestrians that will not be leaving the sections, e.g. because their route ends in one of them or they are alighting a PT vehicle.

Output file of pedestrian area result

Structure of output file:

Element	Description
File title	
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
PTV Vissim	Version number, service pack number, build number

Element	Description
List	List of evaluated areas
Column names	Column headers and units
Data block	Data block with measurement data

Column	Meaning
MeasurementAreaNo	Number of section
tEnter	Time pedestrian entered the section [simulation second]
tLeave	Time pedestrian left the section [simulation second]
PedNo	Pedestrian number
PedType	Pedestrian type
DwellTime	Dwell time [s] pedestrians spent in section
vDes	Desired speed in the currently selected unit for the speed: average, minimum and maximum
v	Speed in the currently selected unit for the speed: average, minimum and maximum
DevSpeed	Deviation of pedestrian speed in the currently selected unit for the speed
Density	Pedestrian density in the sections [ped/m²] or [ped/ft²], depending on the unit set for short distances
AvgXOri	Average of x values of orientation vector
AvgYOri	Average of y values of orientation vector
WorldX **WorldY** **WorldZ**	World coordinate X World coordinate Y World coordinate Z
TimeGain	Time gain [s]: Total time gain is determined based on the delay time. The total **time gain** is determined only if the pedestrian is faster than desired.
TimeDelay	Time delay [s]: Total **time delay** is determined from the "time lost" per time step. This is determined from the difference between actual speed and desired speed if the pedestrian is slower than desired. Delays through using a detour are not taken into account in the delay.
Dist	Distance traveled so far in the sections, in the current unit set for short distances
DistNetwork	Total distance traveled so far in the network in currently selected unit for short distances

Example of raw data **.merP*

The data blocks below the file header contain the area results and all column identifiers.

```
Area Measurement (raw data)
```

```
File:    C:\Dokumente und Einstellungen\All Users\Dokumente\PTV Vision\PTV Vissim
11\Examples Demo\KAmerP2013.inpx
Comment:
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]
Section No. : Section number
tEnter      : Time pedestrians enter the section [simulation second]
tLeave      : Time pedestrians leave the section [simulation second]
PedNo       : Pedestrian number
PedType     : Pedestrian type
Dwelltime   : Dwell time [s] in section
vDes        : Desired speed [km/h]
vX          : X component of the speed vector [km/h]
vY          : Y component of the speed vector [km/h]
v           : Speed [km/h]
DevSpeed    : Deviation of pedestrian speed [km/h]
Density     : Pedestrian density in section [ped/m²]
AvgXOri     : Average of x values of orientation vector
AvgYOri     : Average of y values of orientation vector
WorldX      : World coordinate x
WorldY      : World coordinate y
WorldZ      : World coordinate z
TimeDelay   : Time delay [s]
TimeGain    : Time gain [s]
Dist        : Distance [m] pedestrians traveled in sections
DistNetwork : Distance [m] pedestrian traveled in networt so far
MeasurementAreaNo;      tEnter;       tLeave;    PedNo; PedType;  Dwell time;
                2;       9.0;        28.8;      19;   100;          19.8;
       vDes;   vDes;        vDes;          vX;      vX;      vX;           vY;
Average;   Min;        Max; Average;     Min;     Max;    Average;
       5.1;   5.1;        5.1;         4.8;     0.0;     5.0;          -1.2;
        vY;     vY;           v;          v;      v; DevSpeed;     Density;
       Min;    Max; Average;       Min;     Max;          ;Average;
      -1.3;  -0.0;         4.9;        0.0;     5.1;      0.15;       0.00;
    Density; Density; AvgXOri; AvgYOri;   WorldX;    WorldX;     WorldX;
       Min;     Max;                   Average;      Min;        Max;
      0.00;    0.00;    0.9703;   -0.2413; -1178.1;   0.9703;    -0.2413;
     WorldY;  WorldY;      WorldY;     WorldZ;   WorldZ;    WorldZ;Time delay;
Average;    Min;       Max;Average;     Min;      Max;           ;
     115.3;  111.9;     118.6;        0.0;      0.0;      0.0;        0.6;
TimeGain;        TravDist;     DistNetwork;
       0.0;              27.5;              27.5;
```

11.21 Evaluating pedestrian travel time measurements

With the evaluation of the pedestrian travel time, you record pedestrians when they are added in the start areas until they enter the associated destination areas.

> Note: Pedestrian travel time measurements must be defined in the network (see "Defining pedestrian travel time measurement" on page 998).

by default, the data for all pedestrian classes is entered together. You can also show the data for certain pedestrian classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	*.rsrP	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in result lists (see "Showing result attributes in result lists" on page 1016)
- Save result attributes to a file (see "Using the Direct output function to save evaluation results to files" on page 1018)

Result of pedestrian travel time measurements

The file with pedestrian travel times **.rsrp* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
Attribute names	Description of evaluated data
Data block	Measurement data

Tip: You can also create an evaluation based on the pedestrian origin-destination matrix (see "Saving pedestrian travel time measurements from OD data to a file" on page 1048).

Example of pedestrian travel time sections: raw data **.rsrP*

```
Pedestrian travel time measurement (raw data)
File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Train-
ing\Pedestrians\HR.inpx

Comment:
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

t           : Time ped. entered dest. area of travel time measurement [sim.second]
No.         : Travel time measurement number
PedNo       : Pedestrian number
```

```
PedType    : Pedestrian type
TravDist   : Distance traveled from start to destination area [m]
TravTime   : Time traveled from start to destination are [s]
TimeDelay  : Time delay [s] while traveling from start to destination area
TimeGain   : Time delay [s] while traveling from start to destination area
DevSpeed   : Differences [km/h] between actuel speed and v_des

     t; No.; PedNo; PedType; TravDist; TravTime; TimeDelay; TimeGain; DevSpeed;
  17.1;  30;   414;     100;     14.7;     16.1;       5.0;      0.0;      2.7;
  18.4;  29;    56;     100;     17.3;     18.2;       6.3;      0.0;      3.4;
  19.5;  29;    99;     200;     18.0;     19.2;       7.1;      0.0;      3.0;
....
```

Column	Description
t	Time pedestrian entered destination area of travel time measurement [simulation second]
No.	Travel time measurement number
PedNo	Pedestrian number
PedType	Pedestrian type
Dist	Distance traveled from the start area to the destination area
TravTime	Travel time from the start area to the destination area [s]
TimeDelay	Delay [s] from the start area to the destination area: The total **TimeDelay** when traveling this distance results from the "lost time" per time step. This is determined from the difference between actual speed and **desired speed** if the pedestrian is slower than desired. Pedestrians might walk slower than desired when they have to slow down because of other pedestrians, for instance when pedestrian density increases. Delays through using a detour are not taken into account in the delay.
TimeGain	Time gain [s] from the start area to the destination area: The total **TimeGain** when traveling this distance is determined accordingly like the delay. The total "time gain" is determined only if the pedestrian is faster than desired. Pedestrians might walk faster than desired when they are required to do so because of other pedestrians.
DevSpeed	Difference in [km/h] between the actual speed and **desired speed** while walking from the start to the end of the section.

11.22 Saving pedestrian travel time measurements from OD data to a file

If you manage the pedestrian demand of the OD relations between the pedestrian areas as a pedestrian origin-destination matrix, you do not need to define pedestrian inputs or pedestrian routing decisions with pedestrian routes to destination areas (see "Pedestrian OD matrices" on page 977). From a simulation based on a pedestrian origin-destination matrix, you can save the following aggregated data to an **.rsmp* file:

- Travel time
- Delay
- Relative delay
- Volume

Note: You do not need to define any **travel time measurements**. Vissim defines travel time measurements automatically at the start of the simulation. A temporary travel time measurement is generated for each OD relation in the pedestrian origin-destination matrix:

- The temporary travel time measurement starts at the start area.
- The temporary travel time measurement ends at the destination area.

The output data results from the pedestrian volume per OD relation and aggregation interval.

Further routes of a pedestrian in the network are ignored.

Relevant pedestrians for the evaluation

- Pedestrians who are generated in the origin area according to the pedestrian input are taken into account.
- Pedestrians who reach the destination area at the end of their current static route (which began at one of the origin areas) are taken into account.
- Pedestrians who enter an origin area when they leave a PT vehicle are not taken into account.
- Pedestrians who get to the origin area from another area are not taken into account.

  This also includes pedestrians who traveled a partial route between origin and destination, and pedestrians who are located in the current destination area, but are continuing their walk according to a further routing decision, i.e. these pedestrians are also taken into account.
- The arrival time of a pedestrian at the destination area must be in the evaluation period. Thus only the end of the evaluation period belongs to the interval, and not the start of the evaluation period.
- The start time of the pedestrian in the origin area is not relevant.
- Pedestrians who use different static routes for their path from the same origin area to the same destination area are not differentiated. The following thus applies:
  - For each OD relation, the output data results from all relevant pedestrians.
  - There may be multiple static routes for an OD relation.
- A pedestrian who has completed his route from the origin area to the destination area is no longer recorded.

  Examples:

- The pedestrian is no longer relevant if, in the course of his other routes, he once again takes his original static route from the original origin area to the same destination area.
- The pedestrian is also no longer relevant if his next destination area could also be reached from his original origin area via another static route.
- A pedestrian who enters an area on his static route to his destination, which is the destination area of another static route and which also starts at the routing decision of the pedestrian in his origin area, is also not recorded. This pedestrian is only included in the evaluation when he has reached the destination area of his static route.
- Pedestrians with circular routes, which start in area X and also end in this area X, are included in the evaluation. Since pedestrians within an area are not considered, a circular route requires at least one route intermediate point in an area outside area X, which is the origin and destination of the static route. Even area X, which is the origin and destination of a circular route, can be an intermediate point of the circular route if another intermediate point then exists in another area in the circular route before the route destination is reached in area X.

Configuring direct output of the pedestrian travel times (OD data)

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Pedestrian travel time (OD data)** row, select the option **Write to file**.
3. You can change the time if you wish (see "Using the Direct output function to save evaluation results to files" on page 1018).
4. Click the **More** button.

   *The **Pedestrian travel time (OD data)** window opens.*
5. If desired, change the aggregation interval.
6. Confirm with **OK**.
7. Start the simulation (see "Running a simulation" on page 840).

*The *.rsmp file is saved. If the simulation ends before the specified end of the evaluation, the evaluation also ends with the simulation. Thus the last time interval may be incomplete. The corresponding result values for the remaining time interval are still saved.*

Results of the evaluation of pedestrian travel times (OD data)

The output file **.rsmp* contains the following data:

Element	**Description**
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation

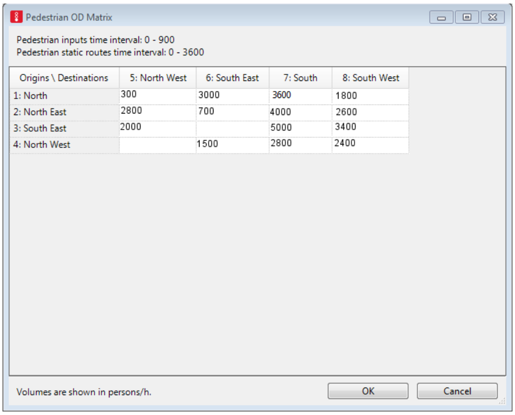
Element	Description
PTV Vissim	Version number, service pack number, build number
Data	Data blocks with the attributes for each OD relation: - for the entire evaluation period - separately for each evaluation interval Header data (one row): - Parameter - Valid time interval in simulation seconds - List of destination areas Output data: - One row per origin area with the output values per attribute and OD relation - For each time interval: separate data block with measurement data per OD relation

Each data block contains the following evaluation data:

Output data	Description
Travel time	Average of all travel times of relevant pedestrians per OD relation.
Delay	Average of all total delay values per OD relation. For each pedestrian, the delay in each simulation step results from: $\text{Time step length} - \frac{\text{Distance walked during time step}}{\text{Desired speed of pedestrian}}$ Example: The delay is 25% of the length of the time step for a pedestrian at 75% of his desired speed. These values are added up over the entire measured distance of the pedestrian. Note: Negative values reduce the total delay value.
Relative delay	Average of all relative delays per OD relation This value is determined separately for each pedestrian as a percentage of the delay in the travel time.
Volume	Number of pedestrians on the basis of which the other result attributes were determined.

Example: Output file **.rsmp*

An output file is generated for the following matrix:



In the example, line breaks in the data block (0s-360s) facilitate readability:

```
Pedestrian travel time measurement (OD data)

File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Train-
ing\Pedestrian OD Matrix.inpx

Comment:
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

Travel time:0s-360s;3;4;5;6;
Delay:0s-360s;3;4;5;6;
Relative delay:0s-360s;3;4;5;6;
Volume:0s-360s;3;4;5;6
1;0.0;0.0;53.5;61.0;
1;0.0;0.0;4.5;4.4;
1;0.00;0.00;0.09;0.07;
1;0;0;166;85
2;31.7;0.0;0.0;73.4;
2;4.1;0.0;0.0;6.0;
2;0.13;0.00;0.00;0.08;
2;97;0;0;97
4;33.2;0.0;0.0;0.0;
4;5.3;0.0;0.0;0.0;
4;0.17;0.00;0.00;0.00;
4;87;0;0;0
8;0.0;77.3;0.0;0.0;
8;0.0;7.4;0.0;0.0;
8;0.00;0.10;0.00;0.00;
8;0;108;0;0

Travel time:0s-180s;3;4;5;6;Delay:0s-180s;3;4;5;6;Relative delay:0s-
180s;3;4;5;6;Volume:0s-180s;3;4;5;6
1;0.0;0.0;51.7;58.0;1;0.0;0.0;4.2;4.0;1;0.00;0.00;0.08;0.07;1;0;0;106;53
```

```
2;31.5;0.0;0.0;0.0;2;4.0;0.0;0.0;0.0;2;0.13;0.00;0.00;0.00;2;80;0;0;0
4;33.1;0.0;0.0;0.0;4;5.6;0.0;0.0;0.0;4;0.17;0.00;0.00;0.00;4;74;0;0;0
8;0.0;0.0;0.0;0.0;8;0.0;0.0;0.0;0.0;8;0.00;0.00;0.00;0.00;8;0;0;0;0

Travel time:180s-360s;3;4;5;6;Delay:180s-360s;3;4;5;6;Relative delay:180s-
360s;3;4;5;6;Volume:180s-360s;3;4;5;6
1;0.0;0.0;56.6;66.0;1;0.0;0.0;5.0;4.9;1;0.00;0.00;0.09;0.07;1;0;0;60;32
2;32.5;0.0;0.0;73.4;2;4.8;0.0;0.0;6.0;2;0.15;0.00;0.00;0.08;2;17;0;0;97
4;33.7;0.0;0.0;0.0;4;3.9;0.0;0.0;0.0;4;0.12;0.00;0.00;0.00;4;13;0;0;0
8;0.0;77.3;0.0;0.0;8;0.0;7.4;0.0;0.0;8;0.00;0.10;0.00;0.00;8;0;108;0;0
```

11.23 Saving pedestrian record to a file or database

Note: You will need the add-on module Viswalk.

The pedestrian record outputs the attribute values for each pedestrian in one row per time step. You can restrict the evaluation to pedestrian classes and individual pedestrians. Pedestrian areas and pedestrian inputs must be defined in the network (see "Modeling construction elements" on page 880), (see "Modeling pedestrian inputs" on page 936). In areas where pedestrians are to be recorded, the **Pedestrian record active** option must be selected (see "Attributes of areas" on page 898).

If under Links, you selected the attribute **Is pedestrian area** (option **Use as pedestrian area**) and want to save the link data in the Pedestrian record, select **Vehicle record active** (see "Attributes of links" on page 409).

You can output the following data and data formats:

Output	ASCII file	MDB file	Attribute file from attribute list
Aggregated data	-	☑	-
Raw data	*.pp	-	-

Tip: You can show many output attributes of the **Pedestrian record** in parallel, also during the simulation, as a result list **Pedestrians in network**.

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Pedestrian record** row, select the option **Write to file**.
3. You can change the time if you wish (see "Using the Direct output function to save evaluation results to files" on page 1018).
4. Click the **More** button.

   *The **Pedestrian record** window opens.*
5. Make the desired changes:

Element	Description
Resolution	Simulation resolution of time steps
Pedestrian filter	▸ **All pedestrians**: takes all pedestrians into account in the evaluation ▸ **Filter by classes**: takes only the selected pedestrian classes into account ▸ **Filter by sections**: takes only the selected sections into account Sections have to be defined (see "Modeling sections" on page 677). The level with the section and the the level on which pedestrians move are the same. The pedestrians' center lies within the section. ▸ **Filter by individual pedestrians**: ☑ If this option is selected, you can right-click the list and select pedestrians by their number. Only the selected pedestrians are taken into account.
Attribute selection	The **Pedestrians in Network: Select Attributes** window opens. You can select attributes for the evaluation (see "Selecting attributes and subattributes for columns of a list" on page 112).

6. Confirm with **OK**.

7. Start the simulation (see "Running a simulation" on page 840).

*A *.pp file is saved.*

Result of pedestrian record

A pedestrian record **.pp* may contain the following data:

Element	Description
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
PTV Vissim	Version number, service pack number, build number
Table	Pedestrians In Network
Column names	Column headers and units
Data block	A row for each pedestrian number and simulation step The columns correspond to the sequence of the attributes in the selection.

Notes:

- By default, the currently selected units for lengths, speeds, accelerations, decelerations are taken into account for the evaluation (see "Selecting network settings for units" on page 205). You can also choose the unit for attribute values in the attribute selection list in the **Format** column (see "Selecting attributes and subattributes for columns of a list" on page 112).
- Coordinates are always specified in [m].
- Units that are shown in the list box or in the column header with the respective attribute cannot be changed.

Your selection of attributes determines which attribute values are included. You may also display these result attributes in the **Pedestrians In Network** list. For example, the following attributes may be included:

Attribute	Definition
Current walking behavior	Walking behavior of the pedestrian type When the walking behavior is area-based, it refers to the walking behavior of the area.
Current destination number	Number of construction element which is the destination
Current destination type	Type of construction element which is the destination
Width	Width of the pedestrian
Level	Number of level
Pedestrian type	Number of pedestrian type
Distance traveled (total)	Total distance traveled so far
Time in network (total)	Total time in network [s] The pedestrian record contains the total time = 0.00, as long as the pedestrian is still in the network.
Speed	Speed at the end of the time step
Height	Size of pedestrian [m]
Construction element number	Number of construction element
Construction element type	Type of construction element
Length	Height of pedestrian [m]
Number	Pedestrian number
Start time	Network entry time [simulation second ss,f], where f (fraction) is a two-digit number
Start time	Start time as time of day [hh:mm:ss,f], where f (fraction) is a two-digit number
Simulation time (time of day)	Simulation time as time of day [hh:mm:ss,f], where f (fraction) is a two-digit number
Simulation second	Simulation time in seconds [ss,f], where f (fraction) is a two-digit number
Static route number	Number of static pedestrian route

Attribute	Definition
Static routing decision number	Number of static pedestrian routing decision
Partial route number	Number of pedestrian route (partial)
Partial routing decision number	Number of pedestrian partial routing decision
Remaining distance	Remaining distance (short length) to the next internal destination. If the next destination or intermediate destination is at the current level of the pedestrian, this is the distance to this destination or intermediate destination. If the next destination or intermediate destination is at another level, the value specifies the distance to the foot of the targeted stairways.
Previous destination number	Number of construction element that was the previous destination
Distance to queue start	Direct distance (short length) to the start of the current queue
Queuing area number	Number of the area in which the current queue is located. 0 = pedestrian is not in a queue
Time in queues (total)	Total waiting time of pedestrian in queues [s]
Time in queue	Waiting time in the last queue [s] in the time step when the pedestrian leaves the queue
Coordinates rear	Coordinates (x), (y), (z) of the rear edge of the pedestrian at the end of the time step
Coordinates front	Coordinates (x), (y), (z) of the front edge of the pedestrian at the end of the time step
Desired speed	Desired speed [km/h]
Time gains	Time gain [s]: Total time gain is determined based on the delay time. The total **time gain** is determined only if the pedestrian is faster than desired.
Time delays	Time delay [s]: Total **time delay** is determined from the "time lost" per time step. This is determined from the difference between actual speed and desired speed if the pedestrian is slower than desired. Delays through using a detour are not taken into account in the delay.

Attribute	Definition
Public transport state	Is the pedestrian waiting, is he alighting from the PT vehicle or walking towards it: - None - Waiting - Approaching - Alighting Instead of the attribute **PT status**, use the attribute **Motion state**, which allows you to record more motion states of pedestrians. You can continue to use existing data with the attribute **PT status**.
Motion state	**MotionState**: The current motion state of a pedestrian shows for how long the pedestrian does a certain activity (see "Showing pedestrians in the network in a list" on page 853).

Pedestrian record example: file *.pp*

```
$VISION
* File: C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Train-
ing\Pedestrians\RiMEA\Rimea 0.inpx
* Comment:
*
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]
*
* Table: Pedestrians In Network
*
* NO: Number
* CONSTRELNO: Construction element number
* CURDESTNO: Current destination number
* DESSPEED: Desired speed [km/h]
* SPEED: Speed [km/h]
*
$PEDESTRIAN:NO;CONSTRELNO;CURDESTNO;DESSPEED;SPEED
1;1;2;5.03;0.00
1;1;2;5.03;0.54
1;1;2;5.03;2.49
1;1;2;5.03;3.54
1;1;2;5.03;4.20
1;3;2;5.03;4.56
…
…
```

11.24 Evaluating nodes

Using the node evaluation, you can record data from nodes of microscopic and mesoscopic simulation in the Vissim network (see "Modeling nodes" on page 705). Node evaluation is used especially to determine specific data from intersections without first having to define all sections manually in order to determine the data.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

Node evaluation also determines exhaust emissions. The basis for these are formed by standard formulas for consumption values of vehicles from TRANSYT 7-F, a program for optimizing signal times, as well as data on emissions of the Oak Ridge National Laboratory of the U.S. Department of energy. The data refers to a typical North American vehicle fleet and does not differentiate between individual vehicle types. Thus node evaluation is used to compare the emissions of different scenarios. To determine emissions for individual vehicle types, use the add-on module EnViVer Pro or EnViVer Enterprise.

Node evaluation accounts for orientations that are not user-defined (see "Attributes of nodes" on page 709).

Queue lengths in the node are recorded by queue counters for each movement and every time step. The queue counters are located at the first signal head or first conflict marker (red) of a priority rule on the link sequence of the turn relation. Queue counters are generated by Vissim. If there is neither a signal head nor a conflict marker (red), Vissim will add the queue counter at the node entrance. Vissim does not record queue lengths of conflict areas with the status red-red or amber-amber.

Delay measurements record the number of vehicles, average delays, average stop delays and the number of stops of a turn relation. Delay measurements consist of one or several travel time measurements. Vissim generates these delay and travel time measurements.

For travel time measurement the following applies:

- Begins upstream of the specified distance before the node entry. For travel time measurements, the result attribute **Start of delay segment before the node** specifies this distance.
- Starts immediately after the closest node, if there are no more than four junctions between the two nodes.
- Ends at node exit

PT stops at public transport stops are not counted as PT stops. Passenger service times of PT vehicles and the waiting time at stop signs are not accounted for in delay times. However, delays caused through braking just before a PT stop and accelerating again afterwards are included in delay times.

If an edge between nodes leads via more than three junctions, it is ignored during node evaluation. Here a junction is any connector that branches off a link upstream of the beginning of another connector or node entry.

If two or more nodes have an identical **FromLink** and **ToLink**, only one queue length is reported.

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	*.knr	☑	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)
- Save raw data to a file or database (see "Configuring evaluations for direct output" on page 1018)

Notes:

Make sure:

- The attribute **UseForEval (Use for evaluation)** of nodes that you want to evaluate, is selected (see "Attributes of nodes" on page 709). Nodes with the selected option **UseForEval** are **active nodes**, while they are **passive nodes** if the option is deactivated.
- Generate a node-edge graph (see "Generating a node-edge graph" on page 718).
- Do not group larger network sections, containing multiple intersections, into a node.

Restrictions for nodes of mesoscopic simulation

- Only the number of vehicles and persons, the loss time per vehicle and per person, and the level of service attributes are recorded and output.
- The values of queue lengths, stop delays, stops and emissions are empty or zero.
- The start section for each internal travel time measurement is placed at the beginning of the first meso edge. The latter must start at least as far upstream of the node as specified in the configuration of the node evaluation, in the **Start of delay segment** attribute.

Defining evaluation of the node evaluation results

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Nodes** row, select **Collect data**.
3. If desired, change the time and interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The window **Node** opens.*

Element	Description
Delay segment start before the node (for node results and node raw data evaluation)	Distance [m] before node, from which delay time is measured
Queue definiton (for queues and node results): Define queue condition	
Begin	A vehicle is in queue if its speed is less than the value **v <** that was entered for the **Begin**.
End	A vehicle remains in queue as long as its speed has not yet exceeded the value **v >** that was entered for the **End**.
Max. headway	Maximum net distance which can occur between two vehicles in queue. The queue is considered to be interrupted if there are larger gaps. Values between 10 and 20m are usual.
Max. length	Maximum queue length. Longer queues may still occur. This attribute is useful if longer queues occur at the next node in the network, but the queues are to be evaluated separately for each junction. When a large value, for example, 4 km, allows a long queue to form, the simulation speed decreases.
Consider adjacent lanes	☑ Select this option to consider adjacent lanes when calculating the queue length for evaluation (see "Showing results of queue counters in lists" on page 1105).

Defining direct output of node evaluation

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Nodes (raw data)** row, click the option of your choice: **Write to file** and/or **Write database**.
3. If desired, change the time and interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The window **Nodes (raw data)** opens.*
5. Make the desired changes:

Element	Description
Start of delay segment before node	Distance [m] before node, from which delay time is measured

Result of node evaluation

The **.knr* file contains the following data:

Attribute	Definition
VehNo	Vehicle number
VehType	Number of vehicle type
StartTime	Simulation second at which the vehicle enters the node
End at	Simulation second at which the vehicle exits the node
StartLink	Link number from which vehicle arrives at node
StartLane	Lane number from which vehicle arrives at node
StartPos	Position from the beginning of the link from which vehicle arrives at node
NodeNo	Node number
Movement	Cardinal points from-to, in which the vehicle moves through the node
FromLink	Number of link that leads to the node Note: This is the link with the shortest path to the node. This does not have to be the link via which the vehicle enters the node.
ToLink	Number of link that leads out of the node. The vehicle has left the node via this link.
ToLane	Number of lane that leads out of the node. The vehicle has left the node via this lane.
ToPos	Position of the node exit on the link which leads out from the node
Delay	Delay time in seconds that it takes to leave the node starting from crossing the start section
StopDelay	StopDelay in seconds within the node, starting from crossing the start section
Stops	Number of stops within the node, starting from crossing the start section
No_Pers	Number of persons in the vehicle

Defining direct output of node evaluation

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Nodes** row, select **Collect data**.
3. If desired, change the time and interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The window **Node** opens.*

Example of node evaluation of raw data **.knr*

```
Node Evaluation (Raw data)
File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
Demo\example.inpx

Comment:  Example, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
```

```
PTV Vissim 11.00-00* [66098]
VehNo; VehType; TStart; TEnd; StartLink; StartLane; StartPos; NodeNo; Movement;
FromLink; ToLink; ToLane; ToPos; Delay; StopDelay; Stops; No_Pers;
   2;   100;        1.7;      7.0;        4;                  1;    0.000;        1;
S-N;       4;       4;                1; 77.268;         0.0;        0.0;     0;        1;
   3;   100;        3.0;      8.2;        4;                  1;    0.000;        1;
S-N;       4;       4;                1; 77.268;         0.0;        0.0;     0;        1;
   1;   100;        1.1;      9.5;        1;                  1;   11.013;        1;
SW-NE;      1;       1;                1; 144.237;        0.0;        0.0;     0;
1;
...
```

Results list Node results

The results list **Node Results** may contain the following data in addition to the above described result attributes of the **.knr* file:

Attribute	Definition
EmissionsCO	**Emissions CO**: Quantity of carbon monoxide [grams]
EmissionsNOx	**Emissions NOx**: Quantity of nitrogen oxides [grams]
EmissionsVOC	**Emissions VOC**: Quantity of volatile organic compounds [grams]
Movement	**Movement**: Number of connectors of a specific inbound link to a specific outbound link of a node. A movement may contain multiple link sequences, for example via parallel connectors. In Node evaluation, various result attributes are automatically calculated for the individual movements. - Number and name of node - Number and name of the **FromLink**, which leads to the node - Position of **FromLink**, at which it passes into the node. - Number and name of the **ToLink**, which leads from the node - Position of **ToLink**, at which it leaves the node. Movements are created for all possible pair options of inbound and outbound links. The **Total** movement contains all movements of each node. The last row of a node movement always contains the **Total** movement. It is only listed specifying the node name and number. To show result attributes of movements, you first need to generate the node-edge graph for evaluations (see "Generating a node-edge graph" on page 718).
Vehs	Number of vehicles
FuelConsumption	Fuel consumption [US liquid gallon]
VehDelay	Vehicle delay (see "Showing delay measurements in lists" on page 1107)
PersDelay	Person delay (see "Showing delay measurements in lists" on page 1107)

<table>
<tr><th>Attribute</th><th>Definition</th></tr>
<tr><td>LOS(All)</td><td>Level of service (transport quality): The levels of transport quality A to F for movements and edges, a density value (vehicle units/mile/lane). It is based on the result attribute Vehicle delay (average). The current value range of vehicle delay depends on the Level of service scheme type of the node Signalized or Non-signalized (see "Attributes of nodes" on page 709). The LOS in Vissim is comparable to the LOS defined in the American Highway Capacity Manual of 2010.
<table>
<tr><th></th><th>Signalized intersection</th><th>Non-signalized intersection</th></tr>
<tr><td>LOS_A</td><td colspan="2">Loss time < 10 s or no volume, as no vehicle is moving, also due to traffic jam</td></tr>
<tr><td>LOS_B</td><td>> 10 s to 20 s</td><td>> 10 s to 15 s</td></tr>
<tr><td>LOS_C</td><td>> 20 s to 35 s</td><td>> 15 s to 25 s</td></tr>
<tr><td>LOS_D</td><td>> 35 s to 55 s</td><td>> 25 s to 35 s</td></tr>
<tr><td>LOS_E</td><td>> 55 s to 80 s</td><td>> 35 s to 50 s</td></tr>
<tr><td>LOS_F</td><td>> 80 s</td><td>> 50 s</td></tr>
</table></td></tr>
<tr><td>LOSVal(All)</td><td>Level-of-service value: Level of transport quality as numbers from 1 to 6 according to the LOS scheme assigned.
No value = no volume.
1 corresponds to LOS_A < 10 s, up to 6 corresponds to LOS_F.
The level-of-service value is more suitable for user-defined formula attributes and color schemes.</td></tr>
<tr><td>Simulation run</td><td>Number of simulation run</td></tr>
<tr><td>Queue length</td><td>Mean of all average queue lengths in a node. Vissim automatically generates queue counters in a node to detect queue lengths. Vissim calculates the average queue length detected by queue counters in a node and then calculates their mean.</td></tr>
<tr><td>Queue length (maximum)</td><td>Maximum queue length. The result depends on the attribute Consider adjacent lanes.</td></tr>
</table>

The attributes will be shown line-by-line for each simulation run and every movement in the node.

For each node, the total results in a later row will show:

- Calculation of the total for vehicles, persons, emissions, fuel consumption
- Calculation of total average for vehicle delay, person delay, stopped delay, queue

length, stops
- Calculation of the maximum queue length (maximum)

11.25 Showing meso edges results in lists

You can display result attributes in lists based on data from the traffic on meso edges during mesoscopic assignment. The delays in the evaluation depend on the meso speed model selected (see "Attributes of links" on page 409). If the Meso speed model **Vehicle-based** is selected, the delays are based on the vehicles' desired speed.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Results of Meso edges evaluation

The results list **Meso edges results** contains the following attributes:

Result attribute Long name	Short name	Description
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Time interval within which the data are aggregated
Meso edge	**MesoEdge**	Number of meso edge
Volume input	**VolInput**	Number of vehicles driving on the edge during the time interval [Veh/h]
Volume dis-charge	**VolDisch**	Number of vehicles exiting the edge during the time interval [Veh/h]
Travel time	**TravTm**	Average travel time on the edge
Delay time	**Delay Tm**	Average delay time based on the difference between actual travel time and travel time at desired speed
Density	**Density**	Vehicle density
Speed	**Speed**	Average speed

Result attribute Long name	Short name	Description
Travel time on inbound meso link	**TravTmInbMesoLink**	Average travel time on inbound link meso edge (output only available for turn meso edges)
Speed on inbound meso link	**SpeedInbMesoLink**	Average speed on inbound link meso edge (output only available for turn meso edges)
Delay time on inbound meso link	**DelayTmInbMesoLink**	Average delay time on inbound link meso edge (output only available for turn meso edges)
Vehicles (entered)	**VehEnter**	Number of vehicles driving on the edge during the time interval
Vehicles (removed)	**VehRemov**	Number of vehicles exiting the edge during the time interval
Demand (latent)	**DemandLatent**	Number of vehicles that could not be deployed in the network (output only available for origin connector meso edges)
Delay (latent)	**DelayLatent**	Total waiting time for vehicles that, since the beginning of the simulation, were not able to enter the network from the origin zone at the time of deployment. (Output only available for origin connector meso edges)

11.26 Showing meso lane results in lists

You can display result attributes in lists that are created from traffic data on meso lanes during mesoscopic simulation. The delays in the evaluation depend on the meso speed model selected (see "Attributes of links" on page 409). If the Meso speed model **Vehicle-based** is selected, the delays are based on the vehicles' desired speed.

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Results of meso lane evaluation

The results list **Meso lanes results** contains the following attributes:

Result attribute Long name	Short name	Description
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Time interval within which the data are aggregated
Meso lanes	**MesoLn**	Number of meso lanes
Volume input	**VolInput**	Number of vehicles driving on the lanes during the time interval [Veh/h]
Volume dis-charge	**VolDisch**	Number of vehicles exiting the lane during the time interval [Veh/h]
Travel time	**TravTm**	Average travel time on a lane
Delay time	**Delay Tm**	Average delay time based on the difference between actual travel time and travel time at desired speed
Density	**Density**	Vehicle density
Speed	**Speed**	Average speed
Queue length	**QLen**	Average queue length: In each time step, the cur-rent queue length is measured and the arith-metic mean is calculated per time interval.
Travel time on inbound meso link	**TravTmInbMesoLink**	Average travel time on inbound link meso edge (output only available for turn meso edges)
Speed on inbound meso link	**SpeedInbMesoLink**	Average speed on inbound link meso edge (output only available for turn meso edges)
Delay time on inbound meso link	**DelayTmInbMesoLink**	Average delay time on inbound link meso edge (output only available for turn meso edges)
Vehicles (entered)	**VehEnter**	Number of vehicles driving on the lanes during the time interval
Vehicles (removed)	**VehRemov**	Number of vehicles exiting the lane during the time interval
Demand (latent)	**DemandLatent**	Number of vehicles that could not be deployed in the network (output only available for origin connector meso edges)
Delay (latent)	**DelayLatent**	Total waiting time for vehicles that, since the beginning of the simulation, were not able to enter the network from the origin zone at the time of deployment. (Output only available for origin connector meso edges)

11.27 Saving data about the convergence of the dynamic assignment to a file

You can save the results of a dynamic assignment in a convergence evaluation file **.cva*.

Note: The evaluation can be performed only with the add-on module **Dynamic Assignment**.

The file contains, for each time interval, the distribution of changes in volume and travel time for all edges and paths, the number of edges/paths that were traversed in this iteration for the first time, the share of edges/paths in percent that met the convergence criterion as well as the info whether the simulation run has converged or not.

Volume changes and travel time changes are divided into classes:

- 10 volume classes for volume changes
- 14 travel time classes for travel time changes

The number of paths and edges is output for each class for which changes have arisen in terms of volume or travel time. This data can be used to determine whether the dynamic assignment has converged (see "Using the dynamic assignment add-on module" on page 692).

You can save only aggregated data in ASCII format as a file **.cva*.

Note: The name of the evaluation file contains the index of the simulation run in the following cases:

- if Vissim is started with the command line parameter -s <n>
- if the simulation run is started via COM and `RunIndex` is set to a value > 0

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Convergence** row, select the option **Write to file**.
3. Confirm with **OK**.

4. Start the simulation (see "Running a simulation" on page 840).

*A *.cva file is saved.*

Result of convergence evaluation

A convergence evaluation file **.cva* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation

<table>
<tr><th>Element</th><th>Description</th></tr>
<tr><td>Date</td><td>Time of evaluation</td></tr>
<tr><td>Vissim</td><td>Version number, service pack number, build number</td></tr>
<tr><td>Data block</td><td>The results of the convergence evaluation are displayed in a table which compares the volumes and travel times of all edges and paths for each iteration. The table is divided into the following blocks:
- Volume difference
  - All edges, except closed edges
  - All paths, except detours
- Travel time difference:
  - Edges
  - Paths: Only the paths used during the last simulation run.
- Shares: Shares in percent:
  - of the converged paths <code>ShrConvPathTT</code>
  - of the converged edges (after travel time) <code>ShrConvEdgeTT</code>
  - of the converged edges (after volume) <code>ShrConvEdgeVol</code>
  - The fulfilment of the convergence condition <code>AssignConv</code> is indicated by a + or -.</td></tr>
<tr><td></td><td>Each row of the evaluation blocks refers to a time interval:
For example, 600.0; 1,200.0; means the interval from simulation second 600 to 1,200.
For each column, each row contains the number of edges and/or number of paths that are contained in the respective class (intervals of volume difference and intervals of travel time difference).
The class boundaries <code>Class from</code> and <code>Class to</code> are contained in the header of a block. <code>Class to</code> belongs to the interval. For example, <code>Class from 3 to 5</code> for edges means: all edges with volume changes greater than 3 vehicles and up to and including 5 vehicles are contained in this class. In the example below, the value 7 is for the time interval 0-600.
- <code>ShrConvPathTT</code>: The share of the paths in percent that has met the convergence criterion if the convergence criterion Travel time on paths has been selected. The percentage weighted by volume is specified in parentheses: Total volume (across all time intervals) of all converged paths / total volume of all paths used
- <code>ShrConvEdgeTT</code>: The share of the edges in percent that has met the convergence criterion if the convergence criterion Travel time of edges has been selected. The percentage weighted by volume is specified in parentheses: Total volume (across all time intervals) of all converged paths / total volume of all paths used
- <code>ShrConvEdgeVol</code>: The share of the edges in percent that has met the convergence criterion if the convergence criterion Volume on edges has been selected.</td></tr>
</table>

Element	Description
	➤ `AssignConv`: The simulation run has converged (**+**), has not converged (**-**) Column `New`: edges/paths entered for the first time in this iteration

Example of convergence evaluation: file **.cva*

```
Convergence evaluation
File:C:\Users\Public\Documents\PTV Vision\11\Examples Training\Dynamic Assign-
ment\Detour\Detour.inpx
ID:  Dynamic Assignment routing example
Date:   Monday, June 18, 2018 10:33:29 AM
PTV Vissim 11.00-00* (64 bit) [66871]
From time;   To time; Volume difference;
(Class from) ;; 0;  1;  3;    6;   11;   26;   51;   101;   251;   501;
(Class to) ;; 0;  2;  5;   10;   25;   50;  100;   250;   500;   ~;
Edges:
0.0;       300.0;    1;   3;   5;   9;   5;   0;   0;   0;   0;   0;
300.0;     600.0;    0;   2;   4;   8;   8;   1;   0;   0;   0;   0;
600.0;     900.0;    3;   6;   8;   4;   2;   0;   0;   0;   0;   0;
900.0;    1200.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1200.0;   1500.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1500.0;   1800.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1800.0;   2100.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2100.0;   2400.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2400.0;   2700.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2700.0;   3000.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
3000.0;   3300.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
3300.0;   3600.0;   23;   0;   0;   0;   0;   0;   0;   0;   0;   0;
Paths:
0.0;       300.0;    0;   1;   4;   7;   0;   0;   0;   0;   0;   0;
300.0;     600.0;    2;   3;   2;   3;   2;   0;   0;   0;   0;   0;
600.0;     900.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
900.0;    1200.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1200.0;   1500.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1500.0;   1800.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
1800.0;   2100.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2100.0;   2400.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2400.0;   2700.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
2700.0;   3000.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
3000.0;   3300.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
3300.0;   3600.0;   12;   0;   0;   0;   0;   0;   0;   0;   0;   0;
From time;   To time; Travel time difference;
(Class from);;0%; 5%;10%;15%; 20%; 30%; 40%; 50%; 60%; 70%; 80%;  90%; 100%; 200%;
(Class to);;5%;10%;15% 20%; 30%; 40%; 50%; 60%; 70%; 80%; 90%; 100%; 200%; ~; New;
Edges:
0.0;     300.0; 8; 6; 6; 0; 3; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
300.0;   600.0; 5; 2; 6; 4; 3; 1; 0; 1; 1; 0; 0; 0; 0; 0; 0;
600.0;   900.0; 8; 5; 2; 2; 2; 1; 2; 0; 0; 1; 0; 0; 0; 0; 0;
900.0;  1200.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
1200.0; 1500.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
1500.0; 1800.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
```

```
1800.0; 2100.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2100.0; 2400.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2400.0; 2700.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2700.0; 3000.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
3000.0; 3300.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
3300.0; 3600.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
Paths:
0.0;     300.0; 6; 2; 3; 1; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
300.0;   600.0; 4; 2; 3; 0; 2; 1; 0; 0; 0; 0; 0; 0; 0; 0; 0;
600.0;   900.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
900.0;  1200.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
1200.0; 1500.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
1500.0; 1800.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
1800.0; 2100.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2100.0; 2400.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2400.0; 2700.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
2700.0; 3000.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
3000.0; 3300.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
3300.0; 3600.0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0; 0;
             ShrConvPathTT;             ShrConvEdgeTT; ShrConvEdgeVol; AssignConv;
75.00%(weighted:  75.63%); 43.48%(weighted:  43.88%);         65.22%;         -;
```

11.28 Evaluating SC detector records

You can use the SC detector record to check control logic of external control procedures, especially VS-PLUS and TL-PDM/C. For each SC, you can show a freely configurable, precise record of the SC values and detector values as well as internal parameters of the control procedure.

The record can be generated for simulation runs and test runs. Detector requirements or macros must be created for test runs.

SCs must be defined in the network (see "Modeling signal controllers" on page 577). You can configure the SC detector record for these SCs (see "Configuring an SC detector record in SC window" on page 1071).

You can save the following data and data formats:

Output	ASCII file	MDB table	Window
Aggregated data	-	-	
Raw data	*.ldp	-	☑

- Save data to **.ldp* file (see "Using the Direct output function to save evaluation results to files" on page 1018)
- Show data in a window (see "Showing a signal control detector record in a window" on page 1072)

All essential values of the parameters and variables of the control procedure are recorded precisely.

11.28.1 Configuring an SC detector record in SC window

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

2. Right-click the desired SC.
3. From the shortcut menu, choose **Edit**.

   *The **Signal Controller** window opens.*

4. Select the **SC Detector Record Config.** tab.

Tip: For external control procedures, more information such as, for example, stage states can be shown in addition to signal groups and detectors. For further details on displaying additional data, refer to the documentation for the respective control program.

---

5. Make the desired changes:

Element	Description
List on the left **Type (Category)**	Select the **Type** whose data you want to show in the detector log. Click the symbol to add the type to the section on the right. The list box contains all types listed in the *.wtt files that are available for an evaluation. For some types, you must choose a corresponding object number. The object number is listed in the box next to them, **SG No.**, **Port No.** or **CP No.**. The heading of the list box depends on the type you have selected, e.g. **SG No.**.
List in the middle **SG No., Det No., CP No.**	If there are several numbers available, from both sections on the right, select a type-specific number, e.g. a detector no. or signal group no. To select entries in long lists, use the PAGE UP, PAGE DOWN, POS1, END and arrow keys.
	Click the corresponding button to add the selected entry to the list on the right and specify which data is shown in the detector log.
List on right	Shows all objects selected in the defined sequence for output in the evaluations window, chosen in both sections on the right, under **Type (categorie)** .
Short title	Abbreviate column title in the SC detector record file **.ldp*
Signal group label:	Labeling of the rows in the detector log for signal groups: - **Number**: By default, the number for labeling signal groups is used. - **Name**: In the evaluation windows, signal groups of the selected SC are labeled with a name.
Detector label:	Labeling of the rows in the detector log for detectors. - **Port number**: By default, the number for labeling detectors is used. - **Name**: In the evaluation windows, detectors of the selected SC are labeled with a name.

6. Confirm with **OK**.

7. Start the simulation (see "Running a simulation" on page 840).

For each SC selected, you can open an ***SC Detector Record*** *window. The SC number is displayed in the window title bar (see "Showing a signal control detector record in a window" on page 1072).*

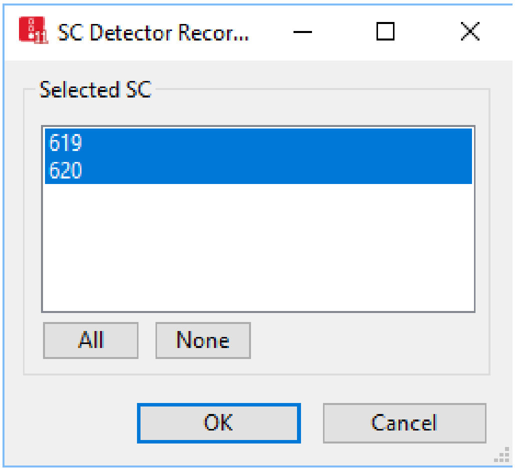
11.28.2 Showing a signal control detector record in a window

The SC detector record contains data of the last 100 seconds of a simulation run or tests. Before showing the SC detector record, configure the SC detector record of the SC (see "Configuring an SC detector record in SC window" on page 1071).

Tip: Alternatively to the next step, you can open the SC detector record for a specific SC. In doing so, you are skipping the next step and are not selecting any settings for labeling. In the **Signal Controllers** list, right-click the desired signal control and from the shortcut menu, choose **Show SC Detector Record**.

1. From the **Evaluation** menu, choose > **Window** > **SC Detector Record**.

   The ***SC Detector Record - Windows*** *opens.*



2. Make the desired changes:

Element	Description
Selected SC	Select SC
All	Select all defined SCs. The evaluation for each SC is shown in a window.
None	Cancel selection of all SCs

3. Confirm with **OK**.

*The **SC Detector Record - window** closes. The **SC<Number> Detector Record - window** for the selected SC opens. When you start the simulation, the SC detector record is displayed depending on the settings made in the **SC Detector Record Config.** tab.*

Results of SC detector evaluation

During a simulation or during a test run, the values of all data specified in the configuration in the last 100 simulation seconds are shown in the SC Detector Record window.

The type of values which can be recorded depends on the control procedures of each SC. You can refer to the documentation of the control program in this case.

If in the **Direct Output** tab you selected SC Detector Record and the simulation or test run have been completed, the **.ldp* file contains the following data: The SC detector record is constructed as a table with one row per simulation second and one column per selected type of values. The record can show up to 1,000 values and up to 3,000 characters per row.

Element	Description
Row 1	Name of evaluation with time of evaluation
Row 2	Comment on simulation
Row 3	SC number, control files, program number, simulation run or test
Other rows	Evaluation data. The columns correspond to the sequence of values in the Configuration window.

Note: The values are saved after the run of the control logic.

When you perform multiple simulation runs using different random seeds, the number of the respective simulation run is added to the name of the evaluation file **.ldp*.

For VS-PLUS, you may also output the following value types in the SC detector record (**SG** = signal group; **VS** = Verkehrsstrom - German for traffic volume):

Value type	Meaning
Active flag of traffic volume	Active flag of traffic volume, VS-Plus specific
GreenMax of traffic volume	GreenMax of traffic volume, VS-Plus specific
Main indicator PE	Next traffic volume of main sequence of the relevant priority element (1-6) to be processed
Control time of traffic volume	Control time of traffic volume, VS-Plus specific

Value type	Meaning
Empty column	Distance from a space
Priority class of traffic volume	By default: 1: PrT, 2: PT, 3: Emergency vehicles
Priority level of traffic volume	Priority level of traffic volume: - 1: normal - 2: maximum reasonable waiting time exceeded
Priority value of traffic volume	Priority level of traffic volume, VS-Plus specific
Signal state of SG	- **.** Red - **=** Red/Amber - **|** Green - **/** Amber
Simul.second	Simulation second between 1 and 999,999,999
Separator	vertical line between two columns
Cycle second	Tx second between 1 and cycle time
DET waiting time	Counter which adds up the time from the first passing of a detector to the next release
Waiting time of traffic volume	Time in seconds since detection of the traffic volume (such as DET waiting time - deceleration time)
State of DET	- **+**: within a simulation second, a vehicle has either completely passed the detector or a vehicle has left the detector and another has reached it (rising and falling edge in one time step) - **|**: at the end of the time step there is a vehicle on the detector and no vehicle has left the detector within this time step. - **.**: The detector is not occupied during the time step.
State of calling point	State of a calling point
State of traffic volume	Coded state of the traffic volume, for example: - 1: detected - 9: Green command given - 10: Switched to green - 21: Red command given

Example of SC detector record file *.ldp*

```
Signal control detector record   [6/6/2016 14:56:45]
Luxembourg, SC 3-10
```

```
SC 5;  program file: vap216.dll;  import files: l05_11hp.VAP, l05.pua;  program no.
1;  simulations run
```

```
      SSSSSSSSS
      iiiiiiiii
      ggggggggg
      .........
      DDDDDDDDD      SSSS
s    CiiiiiiiiiSSSSSStttt
i    ysssssssssttttttaaaa
m    cpppppppppaaaaaatttt
u    lllllllllltttttteeee
l    eaaaaaaaaaeeeeee
.     yyyyyyyyy      DDDD
s    s         DDDDDDEEEE
e    eSSSSSSSSSEEEEEETTTT
c    cGGGGGGGGGTTTTTT
o    o               5555
n    n   2222551133331122
d    d1231357122312341919

 1.0  1.0III...I............
 2.0  2.0III...I............
 3.0  3.0III...I............
 4.0  4.0III...I............
 5.0  5.0III...I............
 6.0  6.0///...I............
 7.0  7.0///...I............
 8.0  8.0///...I............
 9.0  9.0......I............
10.0 10.0...IIII............
11.0 11.0...IIII............
12.0 12.0...IIII............
13.0 13.0...IIII............
14.0 14.0...IIII............
15.0 15.0...IIII............
16.0 16.0...IIII............
17.0 17.0...IIII............
18.0 18.0...IIII............
19.0 19.0...IIII............
20.0 20.0...IIII............
...
```

11.28.3 Results of SC detector evaluation

During a simulation or during a test run, the values of all data specified in the configuration in the last 100 simulation seconds are shown in the SC Detector Record window.

The type of values which can be recorded depends on the control procedures of each SC. You can refer to the documentation of the control program in this case.

If in the **Direct Output** tab, you selected SC Detector Record and the simulation or test run have been completed, the **.ldp* file contains the following data: The SC detector record is constructed as a table with one row per simulation second and one column per selected type of values. The record can show up to 1,000 values and up to 3,000 characters per row.

Element	Description
Row 1	Name of evaluation with time of evaluation
Row 2	Comment on simulation
Row 3	SC number, control files, program number, simulation run or test
Other rows	Evaluation data. The columns correspond to the sequence of values in the Configuration window.

Note: The values are saved after the run of the control logic.

If simulations run several times, the respective Random Seed is attached as an index of the simulation run to the name of the evaluation file **.ldp*.

For VS-PLUS, you may also output the following value types in the SC detector record (**SG** = signal group; **VS** = Verkehrsstrom - German for traffic volume):

Value type	Meaning
Active flag of traffic volume	Active flag of traffic volume, VS-Plus specific
GreenMax of traffic volume	GreenMax of traffic volume, VS-Plus specific
Main indicator PE	Next traffic volume of main sequence of the relevant priority element (1-6) to be processed
Control time of traffic volume	Control time of traffic volume, VS-Plus specific
Empty column	Distance from a space
Priority class of traffic volume	By default: 1: PrT, 2: PT, 3: Emergency vehicles
Priority level of traffic volume	Priority level of traffic volume: - 1: normal - 2: maximum reasonable waiting time exceeded
Priority value of traffic volume	Priority level of traffic volume, VS-Plus specific

Value type	Meaning
Signal state of SG	- **.** Red - **=** Red/Amber - **|** Green - **/** Amber
Simul.second	Simulation second between 1 and 999,999,999
Separator	vertical line between two columns
Cycle second	Tx second between 1 and cycle time
DET waiting time	Counter which adds up the time from the first passing of a detector to the next release
Waiting time of traffic volume	Time in seconds since detection of the traffic volume (such as DET waiting time - deceleration time)
State of DET	- **+**: within a simulation second, a vehicle has either completely passed the detector or a vehicle has left the detector and another has reached it (rising and falling edge in one time step) - **|**: at the end of the time step there is a vehicle on the detector and no vehicle has left the detector within this time step. - **.**: The detector is not occupied during the time step.
State of calling point	State of a calling point
State of traffic volume	Coded state of the traffic volume, for example: - 1: detected - 9: Green command given - 10: Switched to green - 21: Red command given

Example of SC detector record file *.ldp

```
Signal control detector record   [6/6/2016 14:56:45]
Luxembourg, SC 3-10
SC 5;  program file: vap216.dll;  import files: l05_11hp.VAP, l05.pua;  program no.
1;  simulations run
```

```
      SSSSSSSSS
      iiiiiiiii
      ggggggggg
      .........
      DDDDDDDDD      SSSS
s    Ciiiiiiiiissssssstttt
i    ysssssssssttttttaaaa
m    cppppppppppaaaaaatttt
u    llllllllllltttttteeee
l    eaaaaaaaaaeeeeee
.     yyyyyyyyy      DDDD
s    s         DDDDDDEEEE
e    eSSSSSSSSSEEEEEETTTT
c    cGGGGGGGGGTTTTTT
o    o               5555
n    n   2222551133331122
d    d1231357122312341919

 1.0  1.0III...I............
 2.0  2.0III...I............
 3.0  3.0III...I............
 4.0  4.0III...I............
 5.0  5.0III...I............
 6.0  6.0///...I............
 7.0  7.0///...I............
 8.0  8.0///...I............
 9.0  9.0......I............
10.0 10.0...IIII............
11.0 11.0...IIII............
12.0 12.0...IIII............
13.0 13.0...IIII............
14.0 14.0...IIII............
15.0 15.0...IIII............
16.0 16.0...IIII............
17.0 17.0...IIII............
18.0 18.0...IIII............
19.0 19.0...IIII............
20.0 20.0...IIII............
...
```

11.29 Saving SC green time distribution to a file

You can evaluate the absolute frequencies of the occurrence of green durations and red durations for each signal group. The evaluation also includes the calculated averages of both. SCs must be defined in the network (see "Modeling signal controllers" on page 577).

You can save the following data and data formats:

Output	ASCII file	MDB table
Aggregated data	*.lzv	-
Raw data		

- Save data to file (see "Configuring evaluations for direct output" on page 1018)

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **Green time distribution** row, select the option **Write to file**.
3. Confirm with **OK**.

4. Start the simulation (see "Running a simulation" on page 840).

*After the simulation run, an *.lzv file is saved.*

Result of evaluation of SC green time distribution

The file of the SC green time distribution **.lzv* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number

Data	The evaluation contains the following data blocks:
	- Data block with average green times for each SC
	- Separate data blocks with green times and red times for each SC:
	  - The columns contain the individual signal groups *j*
	  - The rows contain the green times and red times *i* (to 120 s)
	Each table entry *ij* indicates how often the signal group *j* received a release duration of *i* seconds.
	- Separate data blocks of all green times and red times with information about their frequency and average for each SC and each signal group.
	In addition, the frequencies are represented graphically in a simple form.

Note: For the graphic representation of green time distribution, you can import the file **.lzv* into a spreadsheet program, e.g. Microsoft™ Excel. The block with the tabular green times is particularly suitable in this case.

Example of SC green time distribution file *.lzv

```
Distribution of Signal Times
Datei:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
```

```
Demo\example.inpx

Comment:  Example, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

Time:     0.0 - 300.0

SC 7, Average Green Times:
   Signal group;         t;
              1;       8.5;
              2;      42.9;
              3;      41.9;
             21;      17.5;
             23;      17.0;
             25;      22.1;
             27;      73.6;

SC 6, Average Green Times:
   Signal group;         t;
              1;      70.3;
              2;      12.0;
...

SC 7, Green Times:
t|SG;    1;    2;    3;   21;   23;   25;   27;
  0;     0;    0;    0;    0;    3;    2;    0;
  1;     0;    0;    0;    0;    0;    0;    0;
  2;     0;    0;    0;    0;    0;    0;    0;
  3;     0;    0;    0;    0;    0;    0;    0;
  4;     1;    1;    1;    0;    0;    0;    0;
  5;    67;    0;    2;    7;    3;    0;    1;
...
118;     0;    0;    0;    0;    0;    0;    0;
119;     0;    0;    0;    0;    0;    0;    0;
120;     0;    0;    0;    0;    0;    0;   15;

SC 7, Red Times:
t|SG;    1;    2;    3;   21;   23;   25;   27;
  0;     0;    0;    0;    0;    0;    0;   20;
  1;     0;    0;    0;    0;    0;    0;    0;
...
 12;    13;    0;    0;    0;    0;    0;    0;
 13;    10;    0;    0;   67;    0;    0;    0;
...
118;     0;    0;    0;    0;    0;    0;    0;
119;     0;    0;    0;    0;    0;    0;    0;
120;     0;    0;    0;    0;    0;    0;    0;

SC 6, Green Times:
```

```
t|Sg;  1;  2;  4;   5;   6;  11;  21;  22;  23;  24;  25;  26;  27;  28;  31;
   0;  0;  0;  0;   0;   0;   0;  10;  10;   0;   0;   0;   0;   0;   0;   9;
   1;  0;  0;  0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;
   2;  1;  0;  0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;   0;
...

SC 7, Signal group 1, Green Times:  (Mean:   8.5)
  4   1  *
  5  67  *******************************************************************
  6   8  ********
...
 22   1  *
 31   1  *

SC 7, Signal group 1, Red Times:  (Mean:  22.3)
  1   1  *
 10   7  *******
 11   9  *********
...
 65   1  *
 79   1  *

SC 7, Signal group 2, Green Times:  (Mean:  42.9)
  4   1  *
  6   2  **
...

SC 6, Signal group 1, Green Times:  (Mean:  70.3)
  2   1  *
  5   1  *
...
```

11.30 Evaluating signal changes

You can record all changes of the individual signal groups from all light signal controls. SCs must be defined in the network (see "Modeling signal controllers" on page 577).

You can save the following data and data formats:

Output	ASCII file	MDB table	Window
Aggregated data	-	-	
Raw data	*.lsa	☑	☑

- Save raw data to a file or database (see "Configuring evaluations for direct output" on page 1018)
- Show data in a window (see "Showing evaluations in windows" on page 1022)

Showing evaluation of signal changes in a window

1. From the **Evaluation** menu, choose > **Windows** > **Signal Changes**.

   *The **Signal changes** window opens in a tab.*

2. Start the simulation (see "Running a simulation" on page 840).

*In the **Signal changes** window, the current values of the following data is displayed:*

Element	Description
SimSec	Simulation second [s]
CycSec	Cycle time [s]
SC	SC number
SG	Signal group number
State	New signal state
old	Time since the last signal state change (= duration of the previous signal state)
Crit	SC type
bcSG	Signal group which has caused the current signal change. This is supported only by certain controllers for vehicle actuated signals.

*A *.lsa file is saved.*

Result of evaluation of signal changes

The record file of signal changes **.lsa* contains the following information:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
SC block	All signal groups
Data block	A row is output for each change process of each signal group. The columns contain the following data (from left to right): - Simulation second [s] - Cycle time [s] - SC number - Signal group number - New signal state - Time since the last signal state change (= duration of the previous signal state) - SC type - Signal group which has caused the current signal change. This is supported only by certain controllers for vehicle actuated signals.

Example of evaluation of signal changes file *.lsa

```
Signal Changes Protocol

File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
Demo\example.inpx

Comment:  Example, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim:   11.00 [66098*]

SC 6  SGroup   1  Link  10283  Lane  1  At   18.0
SC 6  SGroup   2  Link    277  Lane  1  At  300.2
SC 6  SGroup   4  Link  10279  Lane  1  At    9.8
SC 6  SGroup   4  Link  10279  Lane  2  At    3.5
SC 6  SGroup   5  Link    272  Lane  1  At  245.6
SC 6  SGroup   6  Link    273  Lane  1  At  154.0
SC 6  SGroup  11  Link    249  Lane  2  At   66.8
SC 6  SGroup  11  Link    249  Lane  3  At   66.9
SC 6  SGroup  11  Link    249  Lane  1  At   66.8
SC 6  SGroup  21  Link    280  Lane  1  At    2.1
SC 6  SGroup  22  Link    281  Lane  1  At    2.8
SC 6  SGroup  23  Link    285  Lane  1  At    1.0
SC 6  SGroup  24  Link    284  Lane  1  At    3.6
SC 6  SGroup  25  Link    287  Lane  1  At    1.4
SC 6  SGroup  26  Link    286  Lane  1  At    2.0
SC 6  SGroup  27  Link    283  Lane  1  At    2.9
SC 6  SGroup  28  Link    282  Lane  1  At    2.9
SC 6  SGroup  31  Link     73  Lane  1  At    2.8
SC 6  SGroup  31  Link    406  Lane  1  At    2.1
SC 10  SGroup   1  Link  10305  Lane  1  At    8.7
SC 10  SGroup   2  Link      4  Lane  2  At   16.7
...
     1.0;      1.0;      6;      11;  green;          1.0;  VAP;       0;
     1.0;      1.0;      6;       1;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;      32;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;      31;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;      25;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;       3;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;       2;  green;          1.0;  VAP;       0;
     1.0;      1.0;     10;       1;  green;          1.0;  VAP;       0;
     1.0;      1.0;      9;      53;  red/amber;       1.0;  VAP;       0;
     1.0;      1.0;      9;      52;  red/amber;       1.0;  VAP;       0;
     1.0;      1.0;      9;      51;  red/amber;       1.0;  VAP;       0;
     1.0;      1.0;      9;      25;  green;          1.0;  VAP;       0;
     1.0;      1.0;      9;      22;  green;          1.0;  VAP;       0;
     1.0;      1.0;      9;      11;  green;          1.0;  VAP;       0;
     1.0;      1.0;      9;      10;  green;          1.0;  VAP;       0;
     1.0;      1.0;      9;       1;  green;          1.0;  VAP;       0;
```

```
         1.0;       1.0;       8;       52;  red/amber;       1.0;  VAP;       0;
...
```

11.31 Saving managed lane data to a file

During a simulation run, you can save attribute values of managed lanes, general purpose lanes and other attribute values of managed lane facilities in the Vissim network to an *.mle* file. The following must be defined:

- Managed lanes routing decision (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459)
- Toll pricing calculation model (see "Defining toll pricing calculation models" on page 331)
- Decision model for managed lanes facilities (see "Defining decision model for managed lane facilities" on page 329)
- Managed lanes facility (see "Defining managed lane facilities" on page 327)

You can save the following data and data formats:

Output	ASCII file	MDB table
Aggregated data	-	-
Raw data	*.mle	-

- Save data to file (see "Using the Direct output function to save evaluation results to files" on page 1018)

Result of managed lanes evaluation

The managed lanes evaluation file *.mle* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
Managed lanes facility	Name and number of the Managed Lanes
Data block	Data block with a column for each attribute

Example of managed lanes evaluation file *.mle*

```
Managed Lanes Evaluation

File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
Demo\example.inpx

Comment:  Example, SC 3-10
```



```
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

ManagedLanesFacility 1:           Miller Road
Managed Lanes Routing Decision 2:    Managed Lanes Routing Decision Miller

Time            : Simulation time [s]
FacilityNo      : Managed lanes facility number
TTS             : TTS [seconds]
AvsML           : Average speed on managed lanes route [mph]
AvsGP           : Average speed on general purpose route [mph]
SOV             : Toll user class SOV
HOV2            : Toll user class HOV2
HOV3+           : Toll user class HOV3+
Revenues        : Total toll revenue (at managed lanes routing decision)
VehML(All)      : Number of vehicles at end of managed lanes route, All vehicle types
VehGP(All)      : Number of vehicles at end of general purpose route, All vehicle
types

Time; FacilityNo; TTS; AvsML; AvsGP; SOV; HOV2; HOV3+; Revenues; VehML(All); VehGP
(All);
 0.0;   1; 0; 96.56; 96.56; 3.00;  2.00;  0.00;   0.00;  0;   0;
60.0;   1; 0; 96.56; 96.56; 3.00;  2.00;  0.00;  21.00;  0;   0;
120.0;  1; 0; 51.14; 53.18; 3.00;  2.00;  0.00;  36.00;  8;  14;
180.0;  1; 0; 50.37; 52.50; 3.00;  2.00;  0.00;  33.00; 12;  17;
240.0;  1; 0; 51.02; 52.85; 3.00;  2.00;  0.00;  36.00; 10;  19;
300.0;  1; 0; 50.06; 28.78; 3.00;  2.00;  1.00;  39.00; 14;   7;
Total; 1; 0; 50.56; 49.87; 3.00;  2.00;  0.00; 165.00; 44;  57;
```

11.32 Vehicle network performance : Displaying network performance results (vehicles) in result lists

You can show vehicle specific attributes of the entire network compiled in lists.

In microscopic simulation, the network performance is updated with every time step. In mesoscopic simulation, the network performance is updated at the end of the evaluation interval.

Via the COM Interface, you can show and access user-defined attributes in the Network Performance (Vehicles) Results list. User-defined formula attributes are shown, if you use the subattribute **Simulation run**. User-defined attributes are saved when you save the network file.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Result of vehicle network performance

All attributes take into account the vehicles which have already left the network or reached their destination parking lot and the vehicles that are still in the network at the end of the evaluation interval.

The total demand of the input flows and origin-destination matrices during the simulation time results from:

Total = Vehicles In Network + vehicles which have left + vehicles which could not be used (immediately)

The results list **Vehicle network performance results** contains the following attributes. Emissions are specified in grams:

Result attribute Long name	Short name	Description
Emissions (evaporation) HC	**EmissionsEvaporationHC**	Evaporated hydrocarbons
Emissions 1,3-Butadiene	**Emissions13BUT**	1,3-Butadiene (also vinyl ethylene)
Emissions 2,2,4-trimethylpentane	**Emissions224**	2,2,4-trimethylpentane
Emissions Acetaldehyde	**EmissionsAcetaldehyde**	Acetaldehyde (ethanal)
Benzene emissions	**EmissionsBenzene**	Benzol (benzene)
Emissions CH4	**EmissionsCH4**	CH4 (methane)
Emissions CO	**EmissionsCO**	Carbon monoxide
Emissions CO2	**EmissionsCO2**	Carbon dioxide
Emissions ethyl benzene	**EmissionsEthylBenzene**	Ethylbenzene (ethyl benzene, phenyl ethane)
Emissions Formaldehyde	**Emissions13Formaldhyde**	Formaldehyde (methanal)
Emissions HC	**EmissionsHC**	Burned hydrocarbons
Emissions Hexane	**EmissionsHexane**	Hexane
Emissions elemental carbon	**EmissionsElementalCarbon**	Elemental carbon

Result attribute Long name	Short name	Description
Emissions methyl tert-butyl ether	**EmissionsMTBE**	2-methoxy-2-methyl propane
Emissions N2O	**EmissionsN2O**	Nitrous oxide
Emissions Naphtalene gas	**EmissionsNaphtaleneGas**	Naphthalene
Emissions NH3	**EmissionsNH3**	NH3 ammonia
Emissions NMHC	**EmissionsNMHC**	Non-methane hydrocarbons without oxygen compounds
Emissions NMOG	**EmissionsNMOG**	Non-methane hydrocarbons with oxygen compounds
Emissions NO3	**EmissionsNO3**	Nitrates
Emissions NOx	**EmissionsNOx**	Nitrogen oxides
Emissions particulates	**EmissionsParticulates**	Particulate matter
Emissions PM10 brakes	**EmissionsPM10brakes**	Brake abrasion according to standard PM_{10}
Emissions PM10 total	**EmissionsPM10total**	Total quantity of emissions according to standard PM_{10}
Emissions PM10 tires	**EmissionsPM10tires**	Tire abrasion according to standard PM_{10}
Emissions PM2.5 brakes	**Emissions PM2.5 brakes**	Brake abrasion according to standard $PM_{2.5}$
Emissions PM2.5 total	**EmissionsPM2.5total**	Total quantity of emissions according to standard $PM_{2.5}$
Emissions PM2.5 tires	**EmissionsPM25tires**	Tire abrasion according to standard $PM_{2.5}$
Emissions soot	**EmissionsSoot**	Soot particles
Emissions SO2	**EmissionsSO2**	Sulfur dioxide
Emissions sulfate particulates	**EmissionsSulfateParticulates**	Sulfates
Emissions total organic gases	**EmissionsTOG**	Organic gases
Emissions toluene	**EmissionsToluene**	Toluene (toluene, methyl benzene, phenyl methane, methyl benzene)
Emissions VOC	**EmissionsVOC**	Volatile organic compounds
Emissions xylene	**EmissionsXylene**	Xylene

Further result attributes of result list **Vehicle network evaluation results**:

Result attribute Long name	Short name	Description
Vehicles (active)	**VehAct**	Total number of vehicles in the network at the end of the simulation. Vehicles arrived **VehArr** (**Vehicles (arrived)**) and vehicles not being used are not included in the attribute **Vehicles (active)**. Mesoscopic simulation does not include vehicles on zone connector edges.
Vehicles (arrived)	**VehArr**	Vehicles arrived: Total number of vehicles which have already reached their destination and have been removed from the network before the end of the simulation. In dynamic assignment and mesoscopic simulation, this also includes vehicles that have reached their destination parking lot.
Speed (average)	**SpeedAvg**	Average speed [km/h] or [mph] *Total distance DistTot / Total travel time TravTmTot*
Stops (total)	**StopsTot**	Total number of stops of all vehicles that are in the network or have already arrived. The following stops are not included in the attribute **Stops (total)**: - Scheduled stop times of buses and trains at public transport stops - Parking times in parking lots The attribute does not contain any values in mesoscopic simulation. A stop is counted if the speed of the vehicle at the end of the previous time step was greater than 0 and is 0 at the end of the current time step.
Stops (average)	**StopsAvg**	Average number of stops per vehicle: *Total number of stops / (Number of veh in network + number of veh that have arrived)* The attribute does not contain any values in mesoscopic simulation.
Fuel consumption	**FuelConsumption**	Fuel consumption [US liquid gallon]

Result attribute Long name	Short name	Description
Demand (latent)	**DemandLatent**	Number of vehicles from meso origin connector edges, vehicle inputs and parking lots that could not be used. Number of vehicles that were not allowed to enter the network from vehicle inputs and parking lots until the end of the simulation. These vehicles are not counted as vehicles in the **VehAct** network.
Travel time (total)	**TravTmTot**	Total travel time of vehicles traveling within the network or that have already left the network.
Simulation run	**SimRun**	Number of simulation run
Delay (total)	**DelayTot**	Total delay: Total delay of all vehicles in the network or of those that have already exited it. The delay of a vehicle in a time step is the part of the time step that must also be used because the actual speed is less than the desired speed. For the calculation, the quotient is obtained by subtracting the actual distance traveled in this time step and desired speed from the duration of the time step. Loss time, for instance, includes stop times at stop signs. The following times are not included in the attribute **Loss time (total)**: - Scheduled stop times of buses and trains at public transport stops - Passenger service times - Parking times in parking lots
Delay (latent)	**DelayLatent**	Total delay of vehicles that cannot be used (immediately) Total waiting time of vehicles from input flows and parking lots that were not used at their actual start time in the network. This value may also include the waiting time of vehicles that enter the network before the end of the simulation.
Delay (average)	**DelayAvg**	Average delay per vehicle: *Total delay / (Number of veh in the network + number of veh that have arrived)*

Result attribute Long name	Short name	Description
Delay stopped (total)	**DelayStopTot**	Total standstill time of all vehicles that are in the network or have already arrived. Standstill time **=** time in which the vehicle is stationary (speed = 0) The following times are not included in the attribute **Loss time continuous (total)**: - Scheduled stop times of buses and trains at public transport stops - Parking times, regardless of parking lot type The attribute does not contain any values in mesoscopic simulation.
Delay stopped (average)	**DelayStopAvg**	Average standstill time per vehicle: *Total standstill time / (Number of veh in network + number of vehicles that have arrived)* The attribute does not contain any values in mesoscopic simulation.
Distance (total)	**DistTot**	Total distance of all vehicles in the network or of those that have already exited it.

11.33 Pedestrian network performance: Displaying network performance results (pedestrians) in lists

You can show pedestrian specific attributes of the entire network compiled in lists.

Via the COM Interface, you can show and access user-defined attributes in the Network Performance (Pedestrians) Results list. User-defined formula attributes are shown, if you use the subattribute **Simulation run**. User-defined attributes are saved when you save the network file.

by default, the data for all pedestrian classes is entered together. You can also show the data for certain pedestrian classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Network performance results for pedestrians

All attributes take into account the pedestrians who have already left the network or reached their destination and the pedestrians who are still in the network at the end of the evaluation interval.

The results list **Network performance results for pedestrians** contains the following attributes:

Result attribute Long name	Short name	Description
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
Pedestrians (added)	**PedEnt**	Pedestrians that have newly been inserted into the network
Pedestrians (arrived)	**PedArr**	Pedestrians arrived: Total number of pedestrians who have already reached their destination and have been removed from the network before the end of the simulation.
Pedestrians (act-ive)	**FgAct**	Total number of pedestrians in the network at the end of the simulation. Pedestrians arriving **PedArr** (**Pedestrians (arrived)**) and pedestrians not used in the network are not included in the attribute **Pedestrians (active)**.
DichDensity (average)	**DensAvg**	Average pedestrian density: ratio of pedestrians in the network to walkable areas.
Speed (average)	**SpeedAvg**	Average speed [km/h] or [mph] *Total distance DistTot / Total travel time TravTmTot*
Flow (average)	**FlowAvg**	Product of current speed, averaged over all pedestrians and the current density
Travel time (average)	**TravTmAvg**	Average travel time of pedestrians traveling within the network or who have already been removed from the network.
Flow towards destination (average)	**FlowToDestAvg**	Product of current speed, averaged over pedestrians and current density, accounting for static potential and position of each pedestrian.
Speed towards destination (average)	**SpeedToDestAvg**	Average speed [km/h] or [mph] *Total distance DistTot / Total travel time TravTmTot* accounting for the static potential and position of each pedestrian
Stops (average)	**StopsAvg**	Average number of stops per pedestrian: *Total number of stops / (Number of ped in the network + number of ped that have arrived)*

Result attribute Long name	Short name	Description
Delay stopped (average)	**DelayStopAvg**	Average time of stop
Normalized Speed (average)	**NormSpeedAvg**	Ratio of actual speed over desired speed, averaged over pedestrians and time steps.

11.34 Saving PT waiting time data to a file

This record contains the duration of each stop, which is not due to boarding and alighting or due to a stop sign, for each PT vehicle. The evaluation is performed in chronological order. You can further edit the evaluation, for example, in a spreadsheet program.

You can save the following data and data formats:

Output	ASCII file	MDB table
Aggregated data	-	-
Raw data	*.ovw	-

Note: This evaluation always uses metric units.

- Save raw data to a file (see "Using the Direct output function to save evaluation results to files" on page 1018)

Result of evaluation of PT waiting times

A PT waiting time evaluation **.ovw* contains the following information:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
Data	The data block includes: - one column per attribute - one row per PT vehicle stop which is not used for boarding and alighting or caused by a stop sign.

The data block of the evaluation file always shows the following attributes in this sequence:

Column	Description
Time	Simulation second at the end of wait time
VehNo	Number of the vehicle
Line	Number of PT line

Link	Number of link
At	Exact position on the link in [m]
Duration	Wait time [s]

Example of PT evaluation file **.ovw*

```
Table of PT waiting times
File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
Demo\example.inpx

Comment:  Example, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

Time  VehNo  Line    Link  At   Duration
   55    72   206    2006   92.20      2
   96     2   218     103  320.78     16
   98    72   206     103  254.35      8
  122     3   101     110  537.28      5
  140     1  1110     104   10.66      3
  142    72   206     103  320.79     26
  180    72   206     102   91.89     15
  203    72   206     102   93.38     20
  233     2   218     106   82.44      1
  259   221  1103     220  250.52     80
  260   219  1109     220  236.18     64
  265   325 12121     220  203.25      7
  265   220  1105     201   56.41     40
  271   315   106     107  216.91      8
  275     2   218     109   69.59     15
  290   326  2105     346    9.22     51
  291   220  1105       9   29.68     13
```

11.35 Evaluating data collection measurements

- At least one data collection point on a link must be defined in the network (see "Defining data collection points" on page 446).
- At least one data collection measurement must be defined, and at least one data collection point must be assigned to it (see "Defining a data collection measurement in lists" on page 1011), (see "Generating data collection measurements in lists" on page 1011).
- At least one vehicle input and one vehicle route must be defined on the link (see "Defining vehicle inputs" on page 456), (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459).

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

Saving results of data collection measurements

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	*.mer	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)
- Save raw data to a file (see "Using the Direct output function to save evaluation results to files" on page 1018)
- Show result attributes of measurements in list (see "Showing results of measurements" on page 1014)

Result of evaluation of Data collection measurements

The results list **Data Collection Results** contains the following attributes:

Result attribute Long name	Short name	Description
Simulation run	**SimRun**	Number of simulation run
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated
Data Collection Measurement	**DataCollMeas**	Number of data collection measurement and name of its data collection point
The following result attributes refer to all vehicles in the network that have been recorded during data collection measurement:		
Acceleration	**Acceleration**	Average acceleration of the vehicles
Distance	**Dist**	Distance covered [m] by the vehicles
Length	**Length**	Average length [m] of the vehicles
Vehicles	**Vehs**	Total number of vehicles
Persons	**Pers**	Total number of occupants of the vehicles
Queue delay	**QueueDelay**	Total time in [s] that the vehicles have spent so far stuck in a queue, if the queue conditions are met.
Speed	**Speed**	Average speed of the vehicle at the data collection point
Speed (arithmetic mean)	**SpeedAvgArith**	Arithmetic mean of speed of the vehicles

Result attribute Long name	Short name	Description
Speed (harmonic mean)	**SpeedAvgHarm**	Harmonic mean of speed of the vehicles
Occupancy rate	**OccupRate**	Share of time [0% bis 100%] in the last simulation step, during which at least one data collection point of this data collection measurement was busy.

The **.mer* file contains the following data:

Value	Description
t(enter)	Time at which the front end of the vehicle has passed the data collection point. Time - 1.00: The front end has already passed the section in a previous time step.
t(leave)	Time at which the rear end of the vehicle has passed the data collection point. Time -1.00: The rear end of the vehicle has not yet reached the data collection point.
VehNo	Internal number of the vehicle
Type	Vehicle type, for example, 100 = car
Line	PT line, only for PT vehicle types, otherwise = 0
v[km/h]	Speed
b[m/s²]	Acceleration
Occ	Occupancy: Time in s that the vehicle has spent above data collection point in this simulation second
Pers	Number of persons in the vehicle
tQueue	Queue time: Total time in [s] which the vehicles have spent so far stuck in a queue, if the queue conditions are met.
VehLength [m]	Vehicle length in [m]

Example: file **.mer*

```
Data Collection (Raw Data)

File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples Demo\lux3_
10.inpx

Comment:  Luxembourg, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

Data collection point  3131: Link    46 Lane 1 at  179.168.
Data Collection Point  3151: Link 10065 Lane 1 at    2.568 m.
Data Collection Point  3211: Link    42 Lane 1 at  197.590 m.
Data Collection Point  3231: Link    49 Lane 1 at  197.617 m.
Data Collection Point  3311: Link 10063 Lane 1 at    6.208 m.
```

```
Data Collection Point  3321: Link 10062 Lane 1 at    5.514 m.
Data Collection Point  3351: Link 10064 Lane 1 at    3.096 m.
…

Measurement; t(enter); t(leave); VehNo; Type; Line; v[km/h]; a[m/s2]; Occ; Pers;
tQueue; VehLength[m];
  6311    16.95    -1.00    10    17   0    7.9    -2.83   0.05   1 0.0    4.55
  6311    -1.00    17.60    10    17   0    6.0    -2.83   0.00   1 0.0    4.55
  6312    19.90    -1.00    15    11   0    5.3    -2.68   0.10   1 0.0    4.11
  6321    20.03    -1.00    14    14   0   13.5    -0.99   0.07   1 0.0    4.11
  6321    -1.00    20.34    14    14   0   13.2    -0.99   0.04   1 0.0    4.11
  6312    -1.00    20.94    15    11   0    2.6    -2.68   0.04   1 0.0    4.11
…
```

11.36 Evaluating vehicle travel time measurements

A vehicle travel time measurement consists of a From Section and a To Section. The mean travel time from traversing the From Section up to traversing the To Section, including the waiting time and/or holding time, is calculated as well as the distance traveled between the start section and destination section.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can also evaluate delays with travel time measurements (see "Showing delay measurements in lists" on page 1107).

Note: Vehicle travel time measurements must be defined in the network (see "Defining vehicle travel time measurement" on page 447).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	*.rsr	☑	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in result lists (see "Showing result attributes in result lists" on page 1016)
- Save result attributes to files or databases (see "Configuring evaluations for direct output" on page 1018)

Result of evaluation of travel time measurements

The results list **Vehicle Travel Time Results** contains the following attributes:

Result attribute Long name	Short name	Description
Vehicles	**Vehs**	Number of vehicles recorded
Vehicle travel time measurement	**VehTravTmMeas**	Name of vehicle travel time measurement
Travel time	**TravTm**	Average travel time [s] of vehicles in the network
Simulation run	**SimRun**	Number of simulation run
Distance traveled	**Dist**	Average distance traveled [m] between the start section and destination section of travel time measurement. If there is only one path leading from the start section to the destination section, its value corresponds to attribute **Distance** value of travel time measurement (see "Attributes of vehicle travel time measurement" on page 448).
Time interval	**TimeInt**	Duration of the evaluation intervals in which the data is aggregated

The travel time file *.rsr* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
Data block	One row for each travel time measurement: - **Time**: simulation second - **No.**: number of simulation run - **Veh**: number of vehicle - **VehType**: number of vehicle type - **TravTm**: travel time in [s] - **Dist**: distance in [m] - **Delay**: delay time in [s], difference to optimal (ideal, theoretical) travel time

Example of travel time measurement file *.rsr*

```
Table of Travel Times

File:    C:\Program Files\PTV Vision\PTV Vissim 11\Examples Demo\Manual\lux3_10.inpx

Comment:  Manual, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
```

```
PTV Vissim 11.00-00* [66098]

  Time;   No.;     Veh; VehType;  TravTm;  Delay;
  75.7;  4031;      3;    402;    4.8;  0.0;
  99.2;  4102;      2;    402;  39.2;  0.0;
 106.0;  4041;      3;    402;  18.5;  0.0;
 118.8;  4092;      2;    402;  13.1;  0.0;
 124.2;9063035;    15;     11; 113.8;  0.0;
 126.4;  4051;      3;    402;  19.9;  1.3;
 127.6;9063035;    23;     16; 112.5;  0.6;
 137.2;4035051;     3;    402;  65.1;  1.1;
 140.3;9063035;    94;     15;  81.4;  1.7;
 145.2;  4102;     72;    401;  73.2;  1.3;
..
```

11.37 Showing signal times table in a window

You can show the current signal states and detector states during a simulation or during interactive tests of signal control logic in a window. Therein, the green times, yellow times and red times are represented graphically along a horizontal time axis for each selected signal control.

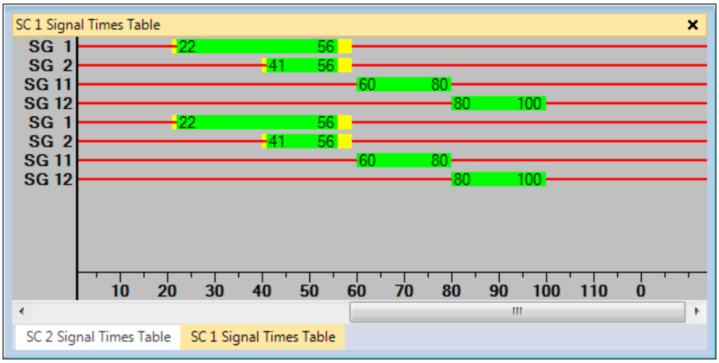
SCs must be defined in the network (see "Modeling signal controllers" on page 577).

1. Ensure that the signal times table is configured according to your requirements (see "Configuring signal times table on SC" on page 1100), (see "Configuring the display settings for a signal times table" on page 1102)
2. From the **Signal Control** menu, choose > **Signal Controllers**.
3. Right-click the desired SC.
4. From the shortcut menu, choose **Show signal times table**.

*The **Signal Times Table Windows** window closes. The **SC <no.> Signal times table** window opens.*

When you start the simulation, colors indicate the current status of the respective signal group.

The signal times table is processed from right to left. The processing speed depends on your simulation speed (see "Defining simulation parameters" on page 840). The state of the current time step is represented at the right edge of the window. The numbers specify the start time and end time of signal states.



If the signal times table also contains detectors, the color of the detector indicates the occupancy state:

Detector color	Description of occupancy state
Change from black line (empty) to light blue	A vehicle passes the loop within a time step, rising and falling pulse edge within a simulation second.
Change from navy blue to light blue	A vehicle leaves the loop, and another vehicle enters the loop within the same time step, rising and falling pulse edge within the same simulation second.
Light blue (several seconds)	Several vehicles pass the loop within a time step, rising and falling pulse edge within a simulation second.
Navy blue	A vehicle is located on the detector at the end of the time step. Thus another vehicle enters the loop in the relevant second, but does not leave it again within the same second. A long navy blue bar means that a vehicle is on the detector. This corresponds to the character ‘|’ in the SC detector record (see "Evaluating SC detector records" on page 1070).

Displaying the time difference

Thus you can determine, for example, the length of time from a detector call to the release of a signal group.

1. Click the **Simulation single step** button .
2. Click a time, and keep the mouse button pressed.
3. Click a another time, and keep the mouse button pressed.

At the bottom left of the window, the time difference between these two times is displayed.

Using the Signal times table window

In the window, an SC time step is four pixels wide at one call/simulation second. If the controller is updated twice per simulation second, a time step is only two pixels wide, the width is only one pixel if it is updated three times or more.

For a fixed time controller, you enter the number of SC time steps per simulation second as an **SC frequency** (see "Defining SC with frequency" on page 638). For external controllers, see the further information of the documentation on the respective control program.

The label of the time axis indicates the cycle second every ten simulation seconds. In addition, a tick mark indicates each 5 seconds. The time axis remains unchanged when the controller runs with multiple time steps per second.

If the controller DLL or the **.exe* file indicate the same simulation second multiple times in succession, only the first simulation second is used for the axis label.

A maximum of 5,000 time steps of the controller can be shown.

11.37.1 Configuring signal times table on SC

1. From the **Signal Control** menu, choose > **Signal Controllers**.

   *The **Signal Controllers** list opens.*

2. Right-click the desired SC.
3. From the shortcut menu, choose **Edit**.

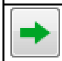
   *The **Signal Controller** window opens.*

4. Select the **Signal Times Table Config.** tab:

Tip: For external control procedures, more information such as, for example, stage states can be shown in addition to signal groups and detectors. For further details on displaying additional data, refer to the documentation for the respective control program.

---

5. Make the desired changes:

Element	Description
Automatic signal times table con-figuration	The option is selected automatically if no user-defined configuration for the SC is available. In this case, the configuration contains all signal groups and detectors listed in the **.wtt* files and cannot be modified. Deselect this option to configure the signal times table manually. The following areas are shown. Then in the sections **Type (category)** and the list next to it, you can select entries and organize them in the section on the right. This allows you to specify the data you want to show in the signal times table.
List on the left **Type (Category)**	Select the **Type** whose data you want to show in the signal times table. Click the symbol to add the type to the section on the right. The list box contains all types listed in the **.wtt*files that are available for the evaluation. For some types, you must choose a corresponding object number from the list box on the right **SG No.**, **Port No.** or **CP No.**. The heading of the list box depends on the type you have selected, e.g. **SG No.**.
List in the middle **SG No., Det No., CP No.**	If there are several numbers available, select a type-specific number, e.g. a detector no. or signal group no. To select entries in long lists, use the Page Up, Page Down, Pos1, End and arrow keys.
	Click the corresponding button to add the selected entry to the list on the right and specify which data is shown in the signal times table.
List on right	Shows all objects selected in the defined sequence for output in the evaluations window, chosen under **Type (category)**.
Signal group label:	Labeling of the rows in the signal times table for signal groups: - **Number**: By default, the number for labeling signal groups is used. - **Name**: In the evaluation windows, signal groups of the selected SC are labeled with a name.
Detector label:	Labeling of the rows in the signal times table for detectors: - **Port number**: By default, the number for labeling detectors is used. - **Name**: In the evaluation windows, detectors of the selected SC are labeled with a name.

6. Configure the display of the signal times table (see "Configuring the display settings for a signal times table" on page 1102).
7. Confirm with **OK**.

8. Start the simulation (see "Running a simulation" on page 840).

   *For each selected SC, an **SC Signal Times Table** window opens with the SC no. in the window title (see "Showing signal times table in a window" on page 1098). Signal states and detector states are displayed dynamically.*

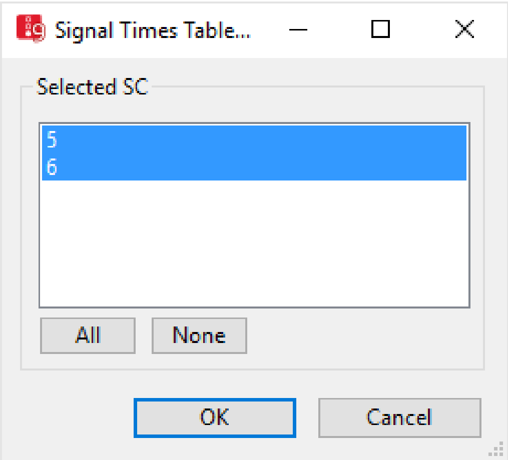
11.37.2 Configuring the display settings for a signal times table

Tips:

- You can save signal times tables with the **SC detector record** to a file (see "Evaluating SC detector records" on page 1070).
- Alternatively, for the next step, open the signal times table for a specific SC without opening the **Signal Times Table Windows**: In the **Signal Controllers** list, right-click the desired SC. Then, from the shortcut menu, choose **Show Signal Times Table**.

1. From the **Evaluation** menu, choose > **Windows** > **Signal Times Table**.

   *The **Signal Times Table Windows** window opens.*



2. Make the desired changes:

Element	Description
Selected SC	Select SC
All	Select all defined SCs. The evaluation for each SC is shown in a window.

3. Confirm with **OK**.

*The **Signal Times Table Windows** window closes. The **SC <no.> Signal times table** window opens (see "Showing signal times table in a window" on page 1098).*

11.38 Saving SSAM trajectories to a file

You can save a binary file **.trj* that contains trajectories. Trajectories describe the course of vehicle positions through the network. This includes the z coordinates of a vehicle. You can upload the file **.trj* to the Surrogate Safety Assessment Model (SSAM) of the Federal Highway Administration Research and Technology of the U.S. Department of Transportation. SSAM is used to evaluate the road safety of transport routes. The content of a binary file **.trj* cannot be interpreted in a text editor.

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	-
Raw data	*.trj	-	-

1. From the **Evaluation** menu, choose > **Configuration** > **Direct Output** tab.
2. In the **SSAM** row, select the option **Write to file**.
3. Confirm with **OK**.

4. Start the simulation (see "Running a simulation" on page 840).

*A file *.trj is saved.*

11.39 Showing data from links in lists

Using the Link evaluation, you can record the result attributes of vehicles based on segments or lanes of links and connectors for the defined time interval. You can restrict the evaluation to vehicle classes and individual vehicles. The following must be defined in the network:

- Links (see "Modeling links for vehicles and pedestrians" on page 406)
- Connectors (see "Modeling connectors" on page 420)
- Vehicle inputs (see "Modeling vehicle inputs for private transportation" on page 454)

Note: Ensure that the attribute **Link evaluation** of links and connectors - which you want to evaluate - is selected (see "Attributes of links" on page 409).

If you assign links a color based on aggregated parameters, for the **Links** evaluation, select **Collect data** (see "Assigning a color to links based on aggregated parameters" on page 179).

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Links** row, select the option **Collect data**.

3. If desired, change the time and/or the interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The **Links** window opens.*
5. Make the desired changes:

Element	Description
Collect link results	➤ **per link segment**: the data is entered in the segments for all lanes ➤ **per lane segment**: the data is entered in the segments for each lane

6. Confirm with **OK**.
7. Start the simulation (see "Running a simulation" on page 840).

Result of evaluation of links

▸ From the **Evaluation** menu, choose > **Result Lists** > **Link Segment Results**.

A link evaluation contains the following data:

Attribute	Definition
Volume	Volume [veh/h] In mesoscopic simulation, for link segments outside the sections of microscopic simulation, the average number of vehicles is displayed that have entered and exited the sections on the meso edge.
Density	Vehicle density
Speed	Average speed
Emissions	For add-on module **API package** only: Results of emission calculation for selected file *EmissionModel.dll* (see "Activating emission calculation and emission model for a vehicle type" on page 274). Emission values are also displayed in: ➤ **Vehicle Network Performance Evaluation** (see "Vehicle network performance : Displaying network performance results (vehicles) in result lists" on page 1085) ➤ **Vehicle record** (see "Saving vehicle record to a file or database" on page 1031) ➤ **Vehicles in Network** list (see "Displaying vehicles in the network in a list" on page 847)
SimRun	**Simulation run**: Number of simulation run
LinkEvalSegs	**Link evaluation segment**: Number of link evaluation segment
DelayRel	**Delay (relative)**: Total delay divided by total travel time of all vehicles in this link segment during this time interval
TimeInt	**Time Interval**: Duration of the evaluation intervals during which the data is aggregated

You can also show the following attributes of the Cartesian world coordinates in the list as indirect attributes of link evaluation segments:

Attribute	Definition
StartCoord	**Start coordinates:** coordinates (x), (y), (z) at which the link evaluation segment begins
StartCoordX	**Start coordinate (x)**: x value of the start coordinate
StartCoordY	**Start coordinate (y)**: y value of the start coordinate
StartCoordZ	**Start coordinate (z)**: z value of the start coordinate
EndCoord	**End coordinates**: coordinates (x), (y), (z) at which the link evaluation segment ends
EndCoordX	**End coordinate (x)**: x value of the end coordinate
EndCoordY	**End coordinate (y)**: y value of the end coordinate
EndCoordZ	**End coordinate (z)**: z value of the end coordinate
Link	Number of the link on which the link evaluation segment is located
StartPos	**Start position**: Position in meters at which the link evaluation segment on the link begins
EndPos	**End position**: Position in meters at which the link evaluation segment on the link ends

11.40 Showing results of queue counters in lists

Queue counters must be defined in the network (see "Defining queue counters" on page 451).

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

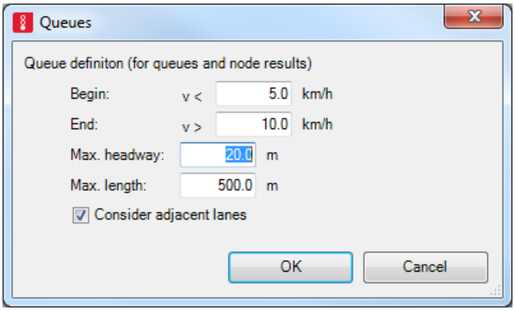
You can save the following data and data formats:

Output	ASCII file	MDB file	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

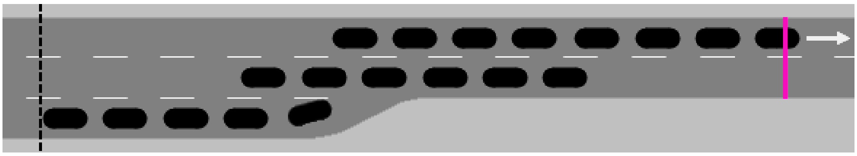
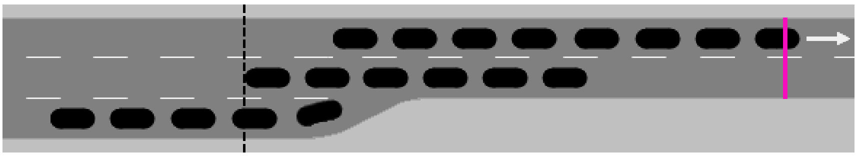
- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

1. From the **Evaluation** menu, choose > **Configuration** > **Result Attributes** tab.
2. In the **Queue counters** row, select the option **Collect data**.
3. If desired, change the time and interval (see "Configuring evaluations of the result attributes for lists" on page 1014).
4. Click the **More** button.

   *The **Queue counters** window opens.*



5. Define the settings for the desired output.

Element	Description
Begin	Define queue condition: A vehicle is in queue if its speed is less than the value **v <** that was entered for the **Begin**.
End	A vehicle remains in queue as long as its speed has not yet exceeded the value **v >** that was entered for the **End**.
Max. headway	Maximum net distance which can occur between two vehicles in queue. The queue is considered to be interrupted if there are larger gaps. Values between 10 and 20m are usual.
Max. length	Maximum queue length. Longer queues may still occur. This attribute is useful if longer queues occur at the next node in the network, but the queues are to be evaluated separately for each junction. When a large value, for example, 4 km, allows a long queue to form, the simulation speed decreases.
Consider adjacent lanes	☑ Select this option to also take adjacent lanes of the previous link into account for calculation of the queue end. In the figure, the dashed line on the left indicates the position of the queue end:  ☐ If this option is not selected, the adjacent lanes of the previous link are not taken into account for calculation of the queue end. In the figure, the dashed line indicates the position of the queue end: 

6. Confirm with **OK**.

7. Start the simulation (see "Running a simulation" on page 840).
8. If you wish, choose in the **Evaluation** menu **Result Lists** > **Queue Results**.

Result of the evaluation of queue counters

Column	Description
QLen	Maximum distance between the traffic counter and the vehicle that meets the queue conditions defined. The queue length is specified as average queue length: With each time step, the current queue length is measured upstream by the queue counter and the arithmetic mean is thus calculated per time interval. This also includes zero values, if there is no vehicle that meets the queue condition. The queue length is the maximum distance between the traffic counter and the vehicle that meets the queue conditions defined.
QLenMax	Queue length (maximum): In each time step, the current queue length is measured upstream by the queue counter and the maximum is thus calculated per time interval.
QStops	Number of queue stops. A queue stop is where one vehicle that is directly upstream or within the queue length exceeds the speed of the **Begin** attribute defined for the queue condition.

11.41 Showing delay measurements in lists

In a delay measurement, the average delay is calculated for all observed vehicles compared to a trip without any other vehicles, signal controls or other required stops.

- At least one vehicle travel time measurement on a link must be defined in the network (see "Defining vehicle travel time measurement" on page 447).
- At least one delay measurement must be defined and at least one vehicle travel time measurement must be assigned to it (see "Defining delay measurement in lists" on page 1012).
- At least one vehicle input and one vehicle route must be defined on the link (see "Defining vehicle inputs" on page 456), (see "Modeling vehicle routes, partial vehicle routes, and routing decisions" on page 459).

All vehicles that pass the vehicle travel time measurements are recorded by delay measurement.

Note: If a vehicle is recorded by several travel time measurements, it is correspondingly often entered in the delay measurement.

by default, the data for all vehicle classes is entered together. You can also show the data for certain vehicle classes separately in the evaluation (see "Configuring evaluations of the result attributes for lists" on page 1014).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in result lists (see "Showing result attributes in result lists" on page 1016)

Result of evaluation of delay measurements

A delay measurement may include the following attribute values:

Column	Description
SimRun	**Simulation run**: Number of simulation run
TimeInt	**Time Interval**: Duration of the evaluation intervals during which the data is aggregated
DelayMeasurement	**Delay measurement**: Number and name of delay measurement
StopDelay	Average stopped delay per vehicle in seconds without stops at PT stops and in parking lots
VehDelay	**Vehicle delay**: Average delay of all vehicles The delay of a vehicle in leaving a travel time measurement is obtained by subtracting the theoretical (ideal) travel time from the actual travel time. The theoretical travel time is the travel time which could be achieved if there were no other vehicles and/or no signal controls or other reasons for stops. Delay time does not account for deceleration in reduced speed areas (see "Using reduced speed areas to modify desired speed" on page 435). To calculate the loss time caused by a desired speed decision, Vissim calculates a theoretical speed and compares it with the current speed (see "Using desired speed to modify desired speed decisions" on page 440). The actual travel time does not include any passenger service times of PT vehicles at stops and no parking time in real parking lots. The delay due to braking before a PT stop and/or the subsequent acceleration after a PT stop are part of the delay.
Stops	Average number of vehicle stops per vehicle without stops at PT stops and in parking lots
Vehs	Number of vehicles
PersDelay	**Person delay**: Average delay [s] of all occupants of the vehicles
Pers	Number of occupants in the vehicles: *number of vehicles * average occupancy rate*

11.42 Showing data about paths of dynamic assignment in lists

You can show data about paths from dynamic assignment in lists.

Note: The **Paths** list can only be displayed using the add-on module Dynamic Assignment (see "Using the dynamic assignment add-on module" on page 692).

You can save the following data and data formats:

Output	ASCII file	MDB table	Attribute file from attribute list
Aggregated data	-	-	☑
Raw data	-	-	-

- Show result attributes in attribute lists (see "Displaying result attributes in attribute lists" on page 1017)
- Show result attributes in a result list (see "Showing result attributes in result lists" on page 1016)

Results of path evaluation

The **Paths** results list contains the following attributes:

Result attribute Long name	Short name	Definition
Volume (old)	**VolOld**	Number of vehicles started on the path, including all vehicle types and/or each vehicle type selected during the last iteration for which the Path file was updated.
Volume (new)	**VolNew**	Number of vehicles started on the path, including all vehicle types and/or each vehicle type selected during the current iteration for which the Path file was updated.
Distance	**Dist**	Distance
Is detour	**Is detour**	Path is a detour
Edge sequence	**EdgeSeq**	Number of edges in sequence
Edge travel time (old)	**EdgeTravTmOld**	Travel Time on the edges of the last iteration for which the Path file was updated.
Edge travel time (new)	**EdgeTravTmNew**	Travel Time on the edges of the current iteration for which the Path file was updated.
Converged	**Conv**	Path converged
To parking lot	**ToParkLot**	Number of To parking lot
Number	**No**	Number of path

Result attribute Long name	Short name	Definition
Static cost	**StaticCost**	Static cost
Path travel time (new)	**PathTravTmNew**	Determined average travel time of path: - prior to a simulation run: smoothed travel time, aggregated from path file by departure time - during a simulation run: changing, incomplete, newly recorded average travel time - after a simulation run: travel time smoothed again with the average newly recorded travel time of all vehicles on this path and aggregated by departure time.
Path-travel time (old)	**PathTravTmOld**	Travel time of previous simulation run, from the start of the simulation run. This value can be used for path search, if in the **Dynamic assignment: Parameters** window, in the **Choice** tab, you selected **Use volume (old)** (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771).
From dynamic vehicle routing decision	**FromDynVehRoutDec**	Number of last dynamic routing decision of path
From parking lot	**FromParkLot**	Number of From parking lot
Route guidance vehicle	**DestinationVeh**	Number of route guidance vehicle

11.43 Saving vehicle input data to a file

You can record the vehicles that are used with vehicle inputs in the network. Links and vehicle inputs must be defined in the network (see "Modeling links for vehicles and pedestrians" on page 406), (see "Modeling vehicle inputs for private transportation" on page 454).

You can save the following data and data formats:

Output	ASCII file	MDB table
Aggregated data	-	-
Raw data	*.fhz	-

- Save raw data to a file (see "Using the Direct output function to save evaluation results to files" on page 1018)

Result of vehicle input data evaluation

A vehicle input evaluation **.fhz* contains the following data:

Element	Description
File title	Name of evaluation
File	Path and name of network file
Comment	Comment on simulation
Date	Time of evaluation
Vissim	Version number, service pack number, build number
Data block	Data block with the following information: - one row per vehicle that enters the Vissim network - one column per attribute

The data block of the evaluation file contains the following attributes in this sequence:

Column	Description
Time	Simulation second
Link	Number of link on which the vehicle is used.
Lane	Number of lane on which the vehicle is used. If a vehicle in the dynamic assignment drives away from a parking lot, the value `0` is output in the **Lane** column.
VehNo	Number of the vehicle
VehType	Number of vehicle type
Line	Number of PT line. No PT line: value `0`
DesSpeed	Desired speed of vehicle

Example evaluation of the vehicle input data file **.fhz*

```
 Table of vehicles entered
 File:    C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\Examples
Demo\example.inpx

Comment:  Example, SC 3-10
Date:     Monday, June 18, 2018 12:23:33 PM
PTV Vissim 11.00-00* [66098]

  Time;     Link;          Lane; VehNo; VehType;   Line; DesSpeed;
  0.1;     1110;             1;     1;      32;   1110;     51.6;
  0.1;     2008;             1;     2;     402;    218;     50.0;
  0.1;     1001;             1;     3;     402;    101;     50.0;
  1.6;      277;             1;     4;      10;      0;     46.3;
  1.8;      267;             1;     5;      10;      0;     46.2;
  2.4;        8;             1;     6;      10;      0;     45.7;
  2.8;      272;             1;     7;      10;      0;     45.6;
  2.9;       70;             1;     8;      17;      0;     54.2;
  3.1;       69;             2;     9;      17;      0;     53.5;
```

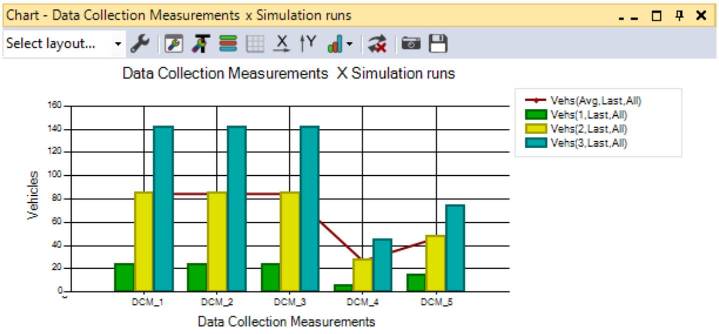
```
  3.4;      274;             1;    10;      17;        0;      53.6;
…
```

12 Creating charts

You can show data graphically in charts, which are convenient for comparison, analysis or use in reports or presentations.

In Vissim you can show the following data in bar charts or line charts:

- Data containing the characteristics of your Vissim network, such as base data or attributes of network objects. This includes data from user-defined attributes.
- Data arising from simulations, such as result attributes from result lists



12.1 Presenting data

Depending on your requirements for data comparison or data analysis, you can create a line chart or a bar chart. As the basis for the chart, select the desired network object type, base data type or result data type as well as at least one object of the selected type and one attribute of the object. The attribute values are plotted on the vertical y-axis.

You can also combine a line chart with a bar chart by using the data series graphic parameters (see "Adjusting how the chart is displayed" on page 1127).

12.1.1 Dimension on the x-axis

The objects from one of the following dimensions are plotted on the x-axis. Select the desired dimension, which for the chosen network object type, base data type or result data type determines whether network objects, attributes, simulation runs, time intervals, vehicle types or pedestrian types are plotted on the x-axis. Then choose the objects in this category to be plotted on the x-axis.

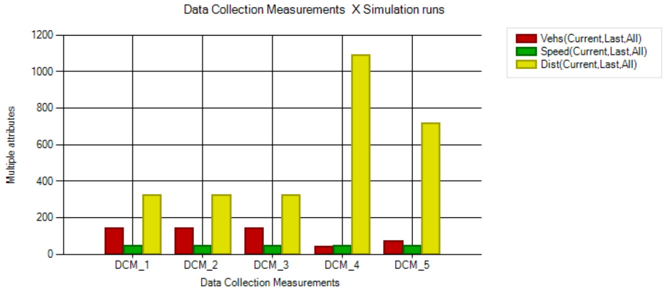
You can choose from the following dimensions and objects in these dimension:

- **Network objects**: You must choose at least one object of the selected network object type, base data type or result data type. You can select an option to choose all objects of the selected network object type, base data type or result data type.
- **Attributes**: You must choose at least one attribute for the selected object or objects.

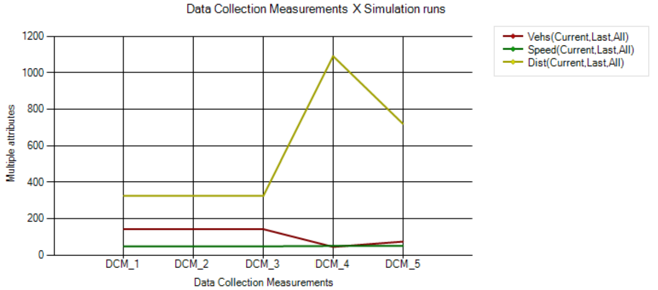
- **Simulation runs**: If you have performed simulation runs, you can choose the desired simulation runs to be used as data sources for the chosen objects and chosen attributes in the chart. You can also select an option to use the data from all simulation runs.
- **Time intervals**: If time intervals are available, you can choose the intervals to be used as data sources for the chosen network objects and attributes in the chart. You can also select an option to use data from all time intervals.
- **Vehicle classes** or **Pedestrian classes**: You can choose vehicle classes or pedestrian classes to be used as data sources for the chosen network objects and attributes in the chart. You can also select an option to use data from all vehicle classes or pedestrian classes.

12.1.2 Attribute values on the y-axis

On a **bar chart** the height of the bar corresponds to the attribute value of the object plotted on the x-axis. If you have chosen several attributes for the objects on the x-axis, a bar for each attribute is shown on the bar chart. The bars are distinguished by a different color for each attribute.



On a **line chart** the vertical position of the data point corresponds to the attribute value of the object plotted on the x-axis. The data points of multiple objects are connected by a line.



For data in percentages, Vissim divides the y-axis into segments from 0 to 100. If the data of only one attribute is shown in percentages, the unit [%] is added to the y-axis labeling.

12.1.3 Presentation of data during an active simulation

If a chart containing dynamic data is displayed during an active simulation, the chart shows the data at each time step. As a result, the chart can change continuously as long as the simulation is running.

12.2 Creating a chart quick-start guide

The quick-start guide shows you the key steps for selecting the desired chart type and data you want to display.

12.2.1 Making preselections or selecting all data

You can choose from the following options for creating a chart:

You want a chart based on a network object type selected from the network objects toolbar

(see "Creating charts from a network object type" on page 1119)

1. On the network object toolbar, right click the network object type of your choice.
2. From the context menu, choose **Create Chart**.

*The **Create Chart** window opens. You can select data and configure the chart.*

You want a chart based on data from network objects shown in a list

(see "Creating charts from data in a list" on page 1121)

1. Open the desired list of network objects, base data or result attributes.
2. Select the desired entries.
3. From the context menu, choose **Create Chart** > **for this attribute**.

*If you have selected enough attribute values in list cells to create a chart, the chart is displayed. The **Create Chart** window does not open.*

You want to select all the data for a chart yourself

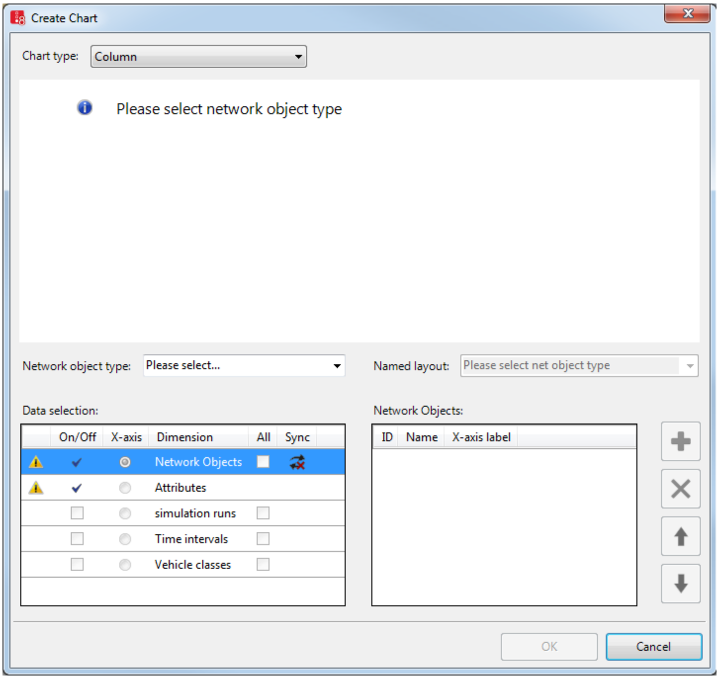
(see "Creating a chart without preselection" on page 1123)

- From the **View** menu, choose > **Create Chart**.

*The **Create Chart** window opens. You can select data and configure the chart.*

12.2.2 Configuring the chart

In the **Create Chart** window, select more data so that the chart can be created.



1. Select the desired entry in the **Chart type** list box.
2. Make sure that the desired entry has been selected in the **Network object type** list box.
3. If you want to add objects in the **Network objects** dimension for the selected network object type in the area on the right, perform the following four steps:
4. Click the **Network objects** dimension.
5. Click the  button.

   A selection window opens.
6. Select the desired objects.
7. Confirm with **OK**.

   The selected objects are shown in the area on the right.
8. If you do not wish the x-axis to be labeled with the object name, in the **X-axis label** column, enter the name of your choice.
9. Repeat the last steps for the **Attributes** dimension.

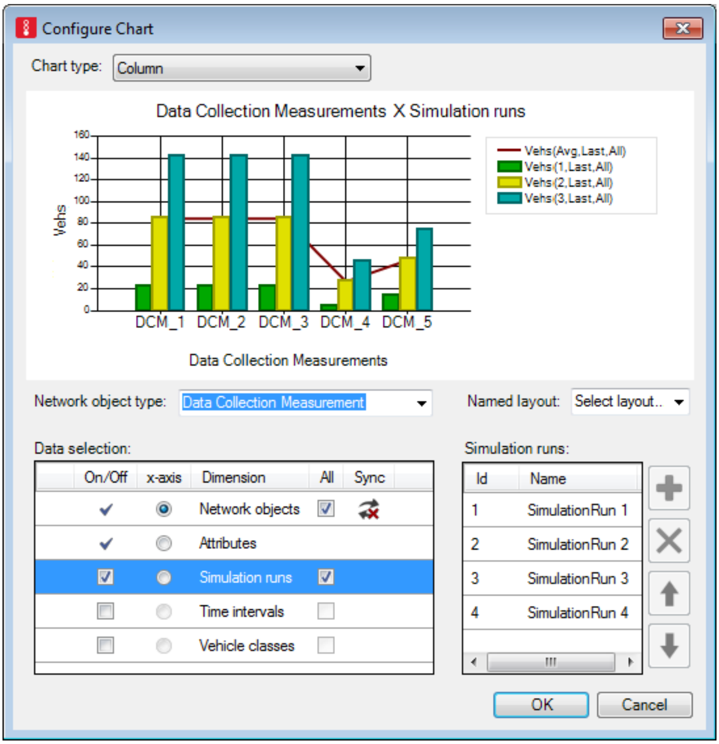
   The selected attributes are shown in the area on the right. A chart preview is displayed.

   Next you can limit the selected objects and attributes to the desired simulation runs, time intervals, vehicle classes or pedestrian classes. You can only select these if the selected *network object type, base data type or result data type has attributes related to the desired dimension.*

10. Repeat the following four steps for the desired dimensions:
11. Make sure that the **On/Off** option is selected for the dimension in the **Data selection** list box.
12. Click the  button.

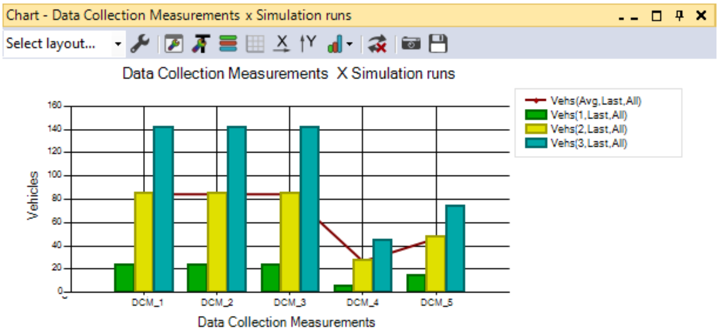
    A selection window opens.
13. Select the desired objects.
14. Confirm with **OK**.
15. In the **Data selection** list, select **x-axis** for the dimension, whose objects you want to plot on the x-axis.

    The selected objects are shown in the area on the right. A chart preview is displayed.



16. Confirm with **OK**.

    The chart is displayed.



17. Edit the chart if you so wish (see "Configuring a created chart" on page 1126).
18. Reuse the chart if you so wish (see "Reusing a chart" on page 1132).

12.3 Charts toolbar

Element	Name	Description
Select layout list box	**Chart layout selection**	➤ Save named chart layout (see "Using named chart layouts" on page 1130) ➤ Select named chart layout
	**Configure Chart**	Open the **Create Chart** window and select data (see "Configuring the chart type and data" on page 1127)
	**Edit window graphic parameters**	Use graphic parameters to define how the various elements of the chart are displayed (see "Adjusting how the chart is displayed" on page 1127)
	**Edit title graphic parameters**	
	**Edit legend graphic parameters**	
	**Edit drawing area graphic parameters**	
	**Edit x-axis graphic parameters**	
	**Edit y-axis graphic parameters**	
	**Edit data series graphic parameters**	

Element	Name	Description
	Synchronization	The chart is synchronized with all network editors and synchronized lists. When you select or deselect bars or network objects in the chart, they are also selected or deselected in other windows, and vice versa. The **Sync** option in the **Create Chart** window is linked to **Synchronization**. The **Sync** option is therefore selected when Synchronization is selected, and vice versa.
	No synchronization	The chart is not synchronized with other program elements. The **Sync** option in the **Create Chart** window is not selected.
	Copying an image to the clipboard	(see "Reusing a chart" on page 1132)
	Export image (Screenshot)	(see "Reusing a chart" on page 1132)

12.4 Creating charts with or without preselection

In the following program elements you can select objects to be used to generate a chart:

- Network objects toolbar (see "Creating charts from a network object type" on page 1119)
- Network editor (see "Creating charts from network objects in the network editor" on page 1120)
- Lists, such as the attribute list of the network object of a network object type, results lists or base data lists (see "Creating charts from data in a list" on page 1121)

You can also generate a chart without using these program elements for preselection. In that case, select all the desired data in the **Create Chart** window (see "Creating a chart without preselection" on page 1123).

12.4.1 Creating charts from a network object type

On the network object toolbar you can select a network object type and use it to configure the data for the chart.

1. On the network object toolbar, right click the network object type of your choice.
2. In the context menu, choose **Create Chart**.

   *The **Create Chart** window opens. The network object type is automatically entered in the **Network object type** list box.*
3. Select the **x-axis** option for **Network objects** in the **Data selection** area.
4. Click the button.

   A selection window opens.

5. Select the desired network objects.
6. Confirm with **OK**.

   The selected objects are shown in the area on the right.
7. Select the **x-axis** option for **Attributes** in the **Data selection** area.
8. Click the button.

   A selection window opens.
9. Select the desired attributes.
10. Confirm with **OK**.

    The selected objects are shown in the area on the right. A chart preview is displayed if the data selection is valid.
11. If you do not wish the x-axis to be labeled with the object name, in the **X-axis label** column, enter the name of your choice.
12. If you wish, repeat the data selection procedure for simulation runs, time intervals, vehicle classes or pedestrian classes. You can only select these if the selected network object type, base data type or result data type has attributes related to the desired dimension.

    A chart preview is displayed if the data selection is valid.
13. Confirm with **OK**.

    *In the Vissim graphical user interface, the **Chart <Chart title>** window opens. The chart is displayed.*
14. Edit the chart if you so wish (see "Configuring a created chart" on page 1126).
15. Reuse the chart if you so wish (see "Reusing a chart" on page 1132).

12.4.2 Creating charts from network objects in the network editor

In the network editor you can select one or more network objects and use them to configure other data for the chart.

1. In the network editor, select the desired network objects of a particular network object type.
2. In the context menu, choose **Create Chart for Selected Objects**.

   *The **Create Chart** window opens. The network object type of the selected network object is automatically shown and selected in the **Network object type** list box.*

   *The **x-axis** option for **Network objects** is automatically selected in the **Data selection** area.*

   *The network objects you selected in the network editor are automatically shown in the **Network objects** area on the right.*
3. Select the **x-axis** option for **Attributes** in the **Data selection** area.
4. Click the button.

A selection window opens.

5. Select the desired attributes.
6. Confirm with **OK**.

   The selected objects are shown in the area on the right. A chart preview is displayed if the data selection is valid.

7. If you do not wish the x-axis to be labeled with the object name, in the **X-axis label** column, enter the name of your choice.
8. If you wish, repeat the data selection procedure for simulation runs, time intervals, vehicle classes or pedestrian classes. You can only select these if the selected network object type, base data type or result data type has attributes related to the desired dimension.

   A chart preview is displayed if the data selection is valid.

9. Confirm with **OK**.

   *In the Vissim graphical user interface, the **Chart <Chart title>** window opens. The chart is displayed.*

10. Edit the chart if you so wish (see "Configuring a created chart" on page 1126).
11. Reuse the chart if you so wish (see "Reusing a chart" on page 1132).

12.4.3 Creating charts from data in a list

You can select data in a list and use it to create a chart. Depending on whether you select columns, rows or cells in the list, different menu options are shown in the context menu for creating the chart and different data is copied.

1. Open the desired list of network objects, base data or result attributes.

Note: If you select data in the next step that cannot be displayed in a chart, the **Create Chart** option in the context menu will not be active and you cannot create a chart.

2. Select the desired data:

Data selection	Description
Click the column header	Selects the values of the attribute of all objects in the list
Press CTRL and click the headers of several columns	Selects the values of the attributes of all objects in the list that are located in the clicked columns
Click a row header	Selects the objects in the row
Press CTRL and click the headers of several rows	Selects the objects located in the clicked rows
Press CTRL and click several cells in a column	Selects only the value of the attribute of the object in the clicked cell
Press CTRL and click several cells in various columns	Selects only the values of the attributes of the objects in the clicked cells

3. Right-click on one of the selected cells.

4. Choose the desired entry from the context menu:

Context menu	Description
Create Chart	Only active when objects have been selected by row. Opens the **Create Chart** window The selected objects are copied and are displayed in the **Network objects** area on the right. In **Data selection**, select **Attributes** and then select the desired attributes on the right. You can select options for the dimensions in **Data selection** (see "Presenting data" on page 1113).
Create Chart > for this attribute	➤ If you have selected attribute values by column, the **Create Chart** window opens. The selected attributes and their objects are copied. You can create the chart or continue with configuration, such as selecting additional attributes or selecting options for the dimensions. ➤ If you have selected enough attribute values in list cells to create a chart, the chart is displayed.
Create Chart > generate for selected attributes over simulation runs	Displayed for every subattribute of the selected attributes that has a non-aggregated value for a simulation run. Opens the **Create Chart** window The subattribute is plotted on the x-axis and all instances are used.
Create Chart > generate for selected attributes over intervals	Displayed for every subattribute of the selected attributes that has a non-aggregated value for a time interval. Opens the **Create Chart** window The subattribute is plotted on the x-axis and all instances are used.
Create Chart > Generate for selected attributes over vehicle classes	Displayed for every subattribute of the selected attributes that has a non-aggregated value for a vehicle class. Opens the **Create Chart** window The subattribute is plotted on the x-axis and all instances are used.

*The **Chart – <Name Network object type>** window opens. The chart shows the data depending on the selected cells in the attributes list:*

Selected cells	Displayed data
In a column of an attribute	If only the values of an attribute in a column have been selected for objects in several rows or in all rows, each object for which a value was selected in its row is plotted on the X axis. For each of these objects, the attribute value is plotted in the chart on the y-axis above the associated object on the x-axis: - In a **line chart** the data points of the attribute values of the successive objects are joined by a line. - In a **bar chart** the attribute value is indicated by a bar above each object.
In different columns	If attribute values for objects in several cells or all cells in several columns have been selected, which means that values of several attributes have been selected, each object for which a value has been selected in its row is plotted on the X axis. For each of these objects, the attribute values are plotted in the chart on the Y axis above the associated object on the X axis: - In a **line chart** the data points of the attribute values of each attribute of the successive objects are joined by a line. Each column in which attribute values are selected is thus represented by a line. Each line that connects the values of a particular attribute has its own color. - In a **bar chart** each attribute value is indicated by a bar. Each column in which attribute values are selected is represented by a bar above the associated object on the X axis. The number of bars above an object on the X axis corresponds to the number of columns in which you have selected attribute values. All of the bars that represent the values of a particular attribute have the same color.

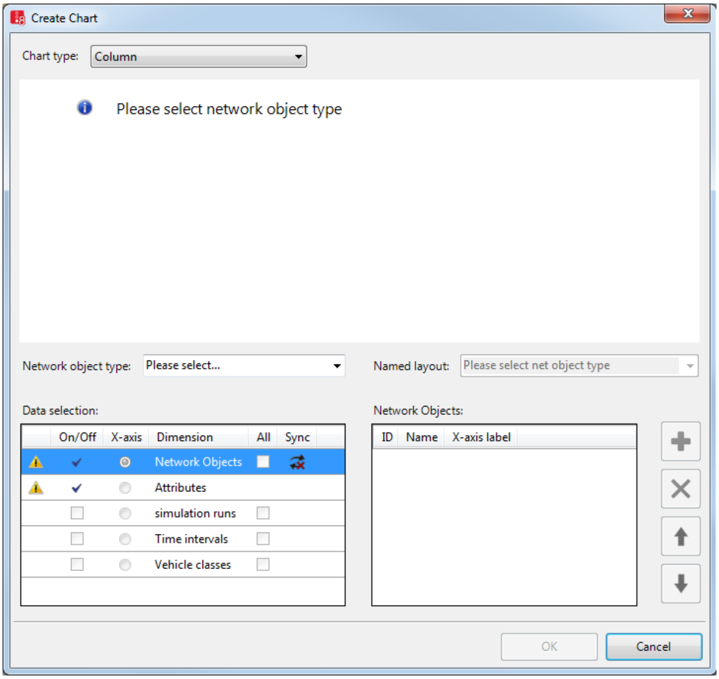
5. Edit the chart if you so wish (see "Configuring a created chart" on page 1126).
6. If desired, continue to use the chart in another program (see "Reusing a chart" on page 1132).

12.4.4 Creating a chart without preselection

You can manually select all the data for a chart.

1. From the **View** menu, choose **Create Chart**.

   *The **Create Chart** window opens.*



2. Select the desired entry in the **Chart type** list box:
   - **Bar chart**
   - **Line chart**
   - **Original**: Resets the chart display to the original data series graphic parameters. Only shown when data for different attributes is displayed in bar charts and line charts, for example after you have changed the **Data series graphic parameters** (see "Adjusting how the chart is displayed" on page 1127).
3. Select the desired network object type, base data type or result data type as the basis for further data selections in the **Network object type** list box.

   *In the next steps, in the left-hand **Data selection** area, configure the dimension, whose objects you can select in the right-hand area.*
4. In the **Data selection** area, please note the symbols:

- **No attribute references are selected yet. Please add attribute references.**: In the right-hand area, no objects have been selected yet for this dimension. You cannot close the window with **OK**. Select objects for the dimension or select another dimension.
- **This net object type does not have any attributes which depend on simulation runs**: The objects of the selected object type **Simulation runs**, **Time intervals**, **Vehicle classes** or **Pedestrian classes** do not have attributes which depend upon this dimension. You can only select simulation runs, time intervals, vehicle classes or pedestrian classes if the

selected network object type, base data type or result data type has attributes that depend on this dimension.

5. In the **Data selection** area, set the desired settings:

Column	Description
On/Off	Enable selection of objects for this dimension. If this option is selected, you can select objects for this dimension with the **(+)** button and add them to the area to the right. This option is always selected for **Network objects** and **Attributes** because you always have to select at least one network object and one attribute.
X-axis	Select a **dimension** whose selected objects will be plotted on the X axis. If the **This net object type does not have any attributes which depend on simulation runs** symbol is not displayed, you can select the **x-axis** option for **Simulation runs**, **Time Intervals**, **Vehicle types** or **Pedestrian types** and use the **(+)** button to select at least one object and add it to the area to the right. The selected dimension is shown above the area to the right as a title.
All	If this option is selected, all objects of the selected object type are selected. The objects are displayed in the area to the right. If you define new network objects in the Vissim network, they are automatically added to the chart. This option is not possible for attributes. • Regarding simulation runs, Vissim only shows the simulation runs for which data is available. • Regarding time intervals, Vissim only shows the MAX time intervals, if in the **Evaluation Configuration** window, in the **Result Attributes** tab, in the columns **Time from** and **Time to** no other values are selected (see "Configuring evaluations of the result attributes for lists" on page 1014). • Regarding vehicle classes or pedestrian classes, Vissim only shows the vehicle classes or pedestrian classes selected in the **Evaluation Configuration** window, in the **Result Attributes** tab, under **Additionally collect data for these classes** (see "Configuring evaluations of the result attributes for lists" on page 1014).
Sync	Synchronized object selection: If this option is selected and objects displayed in the chart are selected in a network editor or a list, the chart also shows this data as selected.

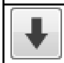
*The title of the area on the right shows the dimension chosen in the **Data Selection** area on the left.*

*In the next steps, in the section to the right, select the objects which you want to plot on the x-axis and for which you activated the option in the **Data Selection** area to the left in the **x-axis** column.*

- **Network Objects**: Select the network objects of the network object type which you selected in the **Network object type** list box. If you selected a network object type from which only objects are available, e.g. **Network performance**, you cannot select, sort or delete any individual objects in the the right-hand section.

- **Attribute**: Select the attribute of the selected network objects.
- **Simulation Runs**: If desired, select simulation runs to be used as sources for the attribute values.
- **Time Intervals**: If desired, select time intervals to be used as sources for the attribute values.
- **Vehicle Classes** or **Pedestrian Classes**: If desired, select classes to be used as sources for the attribute values.

6. Select in the right-hand area the objects you want:

Element	Description
	Open the **<Name Object type>: Select Object(s)** window, in which you can select the objects you want. - For attributes, you can select numeric attributes that are directly related to the selected object type. - For attributes, you can also select attributes that are linked to the object type by a relation. - For subattributes, all non-aggregated entries are merged in the window and the symbol * is shown as a placeholder instead of the individual entries. If you select an entry containing the * symbol and copy it to the list of selected attributes on the right, the merged results will be deleted and the individual entries will be displayed again. Attributes that are linked to the object types by relations are not merged and are not displayed with the * symbol. You can only select the entries one at a time.
	Move selected objects downwards in the right-hand list
	Move selected objects upwards in the right-hand list
	Delete selected objects in the list on the right

A chart preview is displayed when data has been selected in the area on the right and the data selection is valid. The selected objects are plotted on the X axis.

7. If you do not wish the x-axis to be labeled with the object name, in the **X-axis label** column, enter the name of your choice.
8. If you want to apply a named chart layout in the chart, select from the **Named layout** list box the entry desired (see "Using named chart layouts" on page 1130).
9. Confirm with **OK**.

*The **Create Chart** window closes The **Chart <title>** window opens in the Vissim GUI. The chart is displayed.*

12.5 Configuring a created chart

After you create a chart, you can edit it and show it again.

- Select the chart type and/or edit the data (see "Configuring the chart type and data" on page 1127).
- Use graphic parameters to adjust the chart view (see "Adjusting how the chart is displayed" on page 1127).
- Enlarge part of the chart in the chart itself or in the **Configure Chart** window (see "Showing a chart area enlarged" on page 1129)

12.5.1 Configuring the chart type and data

You can reconfigure and redisplay a created chart. You have the same settings available for this as for the creation of a new chart.

1. Click the **Configure Chart** icon on the chart toolbar.

   *The **Configure Chart** window opens.*

2. Make the desired changes (see "Creating a chart without preselection" on page 1123).

12.5.2 Adjusting how the chart is displayed

You can use graphic parameters to adjust how the chart is displayed.

1. Click the desired icon on the chart toolbar.
2. Make the desired changes:

Toolbar button	Name	Description
	Edit window graphic parameters	Choose the background color for the chart window
	Edit title graphic parameters	• **Chart title**: the chart title is shown above the chart. If you do not enter a chart title, Vissim generates a chart title based on the objects selected. You can change the title. • **Chart title font size**: font size in points • **Chart title visibility**: ☑ If this option is selected, the chart title is displayed above the chart.
	Edit legend graphic parameters	• **Legend visibility**: ☑ If this option is selected, the legend is displayed at the position selected in the **Legend position** attribute. • **Legend position**: Choose the desired position for the legend • **Legend font size**: Font size in points

Toolbar button	Name	Description

	Edit drawing area graphic parameters	- **Grid visibility**: ☑ If this option is selected, vertical and horizontal grid lines are shown in the chart.
		- **Activate automatic zoom**:
		  - ☑ Select this option to have Vissim automatically show the entire digram.
		  - ☐ If this option is not selected, you can zoom:
		    - Point the mouse pointer to the diagram and rotate the mouse wheel.
		    - Hold down the left mouse button and use the mouse pointer to drag open a frame over the section with the diagram that want to show enlarged. Release the mouse button.

	Edit x-axis graphic parameters	Show or hide elements on the X axis: - **Axis title**: Name of x-axis which is displayed under x-axis - **Axis title visibility**: ☑ Select this option to show the x-axis title below the chart. - **Axis title font size**: Font size in points - **Axis label visibility**: ☑ Select this option to show the x-axis label below the chart. - **Axis label font size**: Font size in points - **Axis label interval**: Show label of data series only for every n-th position. `1` = Label for each data series, `2` = Label for every second data series, etc. - **Axis unit visibility**: ☑ Select this option to show the units of the x-axis below the chart.
	Edit y-axis graphic parameters	Show or hide elements on the X axis: - **Axis title**: Name of y-axis displayed to the left of y-axis - **Axis title visibility**: ☑ Select this option to show the y-axis title to the left of the chart. - **Axis title font size**: Font size in points - **Axis label visibility**: ☑ Select this option to show the y-axis label next to the chart. - **Axis label font size**: Font size in points - **Axis label interval**: Show label of data series only for every n-th position. `1` = Label for each data series, `2` = Label for every second data series, etc. - **Axis unit visibility**: ☑ Select this option to show the units of the y-axis next to the chart.

Toolbar button	Name	Description

	**Editing data series graphic parameters**	Opens a list box for selection of the attribute whose graphic parameters you want to select. After you have selected the desired attribute in the list box, the graphic parameters list opens:
		- **Series type**: Chart type
		- **Name**: Data series name for legend. Default: Long name of the attribute.
		- **Line**: Line chart
		- **Column**: Bar chart
		- **Line style**:
		  - Solid line
		  - Dashed line
		  - No line
		- **Line color**:
		  - With a bar chart: color of the bar outline
		  - With a line chart: color of the line connecting the data points of the attribute values
		- **Fill style**:
		  - **Solid fill**: the **Fill color** attribute is used
		  - **No fill**
		- **Fill color**: Only with bar charts: fill color for the bars
		- **Marker style**: If you create a line chart, select the shape of the symbol that represents the attribute value in the chart.
		  - Disk
		  - Diamond
		  - Square
		  - None
		- **Marker size**: Symbol size in points

3. Confirm with **OK**.

The chart is shown with the changes.

12.5.3 Showing a chart area enlarged

You can show part of a chart at an enlarged scale. In the enlarged view, you can use the scroll bars to see other parts of the chart. You can also return the chart to its original scale.

These functions can be performed in the chart and in the **Create Chart** window.

1. With the mouse button pressed, drag a frame over the desired section of the chart.
2. Release the mouse button.

This area will be shown as enlarged, and the other parts of the chart in the drawing area will not be visible. In the enlarged view, scroll bars and the button are shown along the axes.

3. If you want to show the chart at its original size in the x-axis or y-axis direction, click the button on the respective axis.

12.6 Using named chart layouts

You can adjust how charts are displayed and assign a name to the current settings, which can be used to retrieve the chart layout later on. You can save these named chart layouts in the **.layx* file. In a chart you can choose a layout from all the named chart layouts to display the chart accordingly.

12.6.1 Generating a named chart layout

1. Open the desired chart.
2. Adjust the chart layout (see "Adjusting how the chart is displayed" on page 1127).
3. On the chart toolbar, enter a unique name in the **Chart layout selection** list box.
4. Confirm with ENTER.

*On the chart toolbar the new chart layout is shown in the **Chart layout selection** box and can be selected.*

12.6.2 Assigning a complete chart layout

You can assign the data selection and the graphic parameters saved in a named chart layout to a chart.

1. Open the desired chart.
2. On the Chart toolbar, in the **Chart layout selection** list, click the icon.
3. Select the desired chart layout.

The chart layout is assigned to the chart.

12.6.3 Assigning only the graphic parameters from a named chart layout

1. Open the desired chart.
2. Right-click on the **Chart layout selection** list box.
3. Position the mouse pointer on the **Apply only graphic parameters** entry in the context menu.

   The named chart layouts are displayed in the context menu.

4. Click the desired chart layout in the context menu.

Only the graphic parameters of the named chart layout are assigned to the chart. The data selection from the chart layout is not assigned.

12.6.4 Assigning only the data selection from a named chart layout

1. Open the desired chart.
2. Right-click on the **Chart layout selection** list box.
3. Position the mouse pointer on the **Apply only data selection** entry in the context menu.

   The named chart layouts are displayed in the context menu.
4. Click the desired chart layout in the context menu.

Only the data selection from the chart layout is assigned to the chart. The graphic parameters from the chart layout are not assigned.

12.6.5 Saving a named chart layout

1. In the menu, select **File** > **Save Layout As**.
2. Enter a unique name.
3. Confirm with **Save**.

*The entire current layout of the Vissim GUI, and thus all named chart layouts, is saved in the *.layx layout file.*

If in the User Preferences, the option **Auto-save layout when network file (inpx) is saved** is selected, the layout file is saved automatically under the name of the currently loaded network file, each time the network file is saved (see "Specifying automatic saving of the layout file *.layx" on page 154).

12.6.6 Reading saved named chart layouts additionally

You can import all chart layouts saved in a **.layx* layout file into your currently opened file. Then you can apply these chart layouts to a chart in the currently opened file.

1. From the **File** menu, choose > **Read Additionally** > **Named Chart Layouts**.
2. Select the desired **.layx* file.
3. Click the **Open** button.

*The chart layouts are imported. When you open a chart, you can select the added chart layouts on the toolbar of the chart window in the **Chart layout selection** list box.*

12.6.7 Deleting a named chart layout

1. On the chart toolbar, right-click on the **Chart layout selection** list box.
2. Position the mouse pointer on the **Delete** entry in the context menu.

   The named chart layouts are displayed in the context menu.
3. Click the desired chart layout in the context menu.

4. Confirm with **Yes**.

12.7 Reusing a chart

You can save a chart in a graphic file or copy it to the clipboard. This allows you to use the chart in other programs, for example for presentations.

12.7.1 Saving a chart in a graphic file

You can save a chart in a graphic file in one of the following formats:

- *bmp*
- *jpeg*
- *png*
- *gif*
- *tiff*

1. On the toolbar of the **Chart - <network object type>** window, click the **Export image (Screenshot)** button .
2. To use the chart in another program, copy the graphic file to the clipboard and paste it in the target program.

12.7.2 Copying a chart to the clipboard

You can save a chart to the clipboard as an image in the **.jpeg* file format.

1. On the toolbar of the **Chart - <network object type>** window, click the **Copy image to clipboard** button .
2. To use the chart in another program, paste it from the clipboard in the target program.

13 Scenario management

Scenario management allows you to manage related networks in a single project. The aim is to model comparable cases in these networks and then compare their simulation results. For each case within a project, you create a separate network that is saved as a scenario. You then change each scenario according to the requirements of the respective case. Then you configure the desired evaluations and perform simulation runs for the scenarios whose simulation results you want to compare.

Scenario Management is primarily based on the following elements:

- **Base network**: The original network from which you want to derive scenarios (see "Placing a network under scenario management" on page 1144). The base network may already contain a modeled network or be empty (see "Opening and editing the base network in the network editor" on page 1146). You cannot use the base network to perform simulation runs. Vissim automatically adopts changes made to the base network into all scenarios of the project, as these have been derived from the base network. Make sure that the entire base network has been modeled and simulation runs trouble-free before you derive a scenario. Do not change the base network once you have derived a scenario from it. If you need to change the base network, from the base network, derive a new scenario without modifications. Then export the scenario from the project structure into a new **.inpx* file and save it under another name. You can now edit this new base file, belonging to a new project ,until you derive a scenario from it.
- **Scenario**: A changed version of the base network or of another scenario within a project (see "Opening and editing scenarios in the network editor" on page 1147). A scenario contains the changes you made in the form of modifications. You can use a scenario to perform simulation runs.
- **Modification**: Changes made to a scenario, with the aim of creating different requirements for a simulation run than those in the scenarios whose simulation results you want to compare. Within a project, scenarios differ from the base network through one or multiple modifications. The scenarios of a project differ from each other through several modifications (see "Opening and editing modifications in the network editor" on page 1148). Within a project, you can assign a modification to different scenarios or deactivate the assignments made. Do not make changes to the base network. Edit the scenarios only.
- **Project** allows you to manage the following elements:
  - Base network
  - Scenarios based on the base network
  - Modifications that distinguish the base network from its scenarios as well as the scenarios from each other.

  You always create a project based on a network file **.inpx*. This network then becomes the base network of the project and the network file is managed under scenario management (see "Placing a network under scenario management" on page 1144). A project and its scenarios and modifications are displayed in the project explorer (see "Using the project explorer" on page 1136). In Scenario Management, the length of the path to the project directory is limited to 214 characters (see "Technical information and requirements" on page 36).

Example of use 1

In your Vissim network, you want to compare the impact of different volumes of an area of investigation during rush hour times in the morning and evening.

You create a new project based on your Vissim network. Vissim saves your network to the project as a base network and automatically creates a copy of the base network called scenario 1. In scenario 1, you define the traffic volumes for the morning. You then duplicate scenario 1 and rename it to scenario 2. In scenario 2, you define the traffic volumes for the evening. You configure the desired evaluations, perform simulation runs for both scenarios and compare the results.

After evaluating the simulation results, you can make changes to the base network and scenarios, duplicate or delete scenarios and perform simulation runs.

Example of use 2

For several PT lines, you want to compare the impact of changes to their route and their time distribution. To do so, you want to change base data, the location of network objects in the network and various attributes of network objects.

You create a new project based on your Vissim network that includes the initial situation of the PT lines. Vissim saves your network to the project as a base network and automatically creates a copy of the base network called scenario 1. In scenario 1, you change the requirements of your use case, namely the route, distribution times, network objects and base data. You then duplicate scenario 1 for all other use cases. You can rename these duplicates. In each duplicate, you change the requirements according to the respective use case, i.e. the route, additional network objects and base data. You then configure the desired evaluations, perform simulation runs for the individual scenarios and compare the simulation results.

After evaluating the simulation results, you can make changes to the base network and scenarios, duplicate or delete scenarios and perform simulation runs.

Managing the project and editing attributes and elements

You manage the base network, scenarios and modifications in the project explorer and the project structure (see "Using the project explorer" on page 1136), (see "Project explorer toolbar" on page 1138), (see "Editing the project structure" on page 1139). The project structure also allows you to edit attributes of the elements. You cannot undo a function executed in scenario management. By default, an undo function is available when you model Vissim networks.

Editing elements in the network editor

In the network editor, you can open and edit all scenarios and modifications (see "Opening and editing the base network in the network editor" on page 1146), (see "Opening and editing scenarios in the network editor" on page 1147), (see "Opening and editing modifications in the network editor" on page 1148). In the **File** menu, in the list of files last opened, you can select the base networks and scenarios last opened in the network editor (if the file path has not been changed).

If in several scenarios, you subsequently create or edit network objects of a network object type and these network objects have the same number in the different scenarios, this may affect the simulation as well as the comparability of the simulation results.

Tip: Alternatively, in the project explorer, right-click **Base network**, Scenario or Modification, and from the shortcut menu, choose **Open**.

Numbering network objects

In various scenarios of project, the values of the **Number** attribute of network objects of a network object type may be identical. This allows you to easily identify the same network objects in different scenarios and compare simulation results referring to these network objects. When you define a new network object in a scenario, by default Vissim suggests a value for the **Number** attribute that is not used in any other scenario for a network object of the same network object type. You can accept the value suggested or enter a different value.

If you add new network objects of the same network object type to different modifications of a scenario, Vissim avoids identical numbers and proposes an individual number for each one. This ensures that you can use the modifications in a scenario without causing conflicts due to identical numbers of network objects of the same network object type.

Comparing scenarios

You can compare the attributes and attribute values of network objects of an opened scenario with those of other scenarios (see "Comparing scenarios" on page 1148).

Calculate multiple scenarios automatically in a row

You can select multiple scenarios and have them automatically calculated one after the other (see "Project explorer toolbar" on page 1138). You can do the same for all scenarios.

13.1 Quick start scenario management

This quick start info describes a simple use case, illustrating the most important steps of how to place a Vissim network under scenario management: A project with a base network and a scenario is created. From this scenario, another scenario is derived. The scenario is edited and saved. You can then start a simulation run for both scenarios to compare their simulation results.

1. Save the **.inpx* network file that contains the original network and shall serve as the base file.
2. Create a project (see "Placing a network under scenario management" on page 1144).

   *Vissim generates the project and automatically creates the base network and scenario 1 based on the *.inpx network file currently open and saved.*
3. Open scenario 1, edit the network and then save scenario 1 (see "Opening and editing scenarios in the network editor" on page 1147).

Vissim automatically creates one or several modifications for scenario 1 based on the changes made.

4. Duplicate scenario 1 (see "Project explorer toolbar" on page 1138).
5. In the network editor, open the duplicate, edit the network and then save the duplicate (see "Opening and editing scenarios in the network editor" on page 1147).
6. Configure the evaluations you want to perform for the simulation runs of scenario 1 and the duplicate (see "Performing evaluations" on page 1001).
7. Start a simulation run for scenario 1 (see "Running a simulation" on page 840).
8. Start a simulation run for the duplicate.
9. Compare the results of the two simulation runs.
10. If desired, compare the scenarios (see "Comparing scenarios" on page 1148).

13.2 Using the project explorer

In the project explorer, a treeview structures the base network, scenarios and modifications of the current project. The functions on the toolbar and in the shortcut menu of the Project explorer allow you to manage scenarios and modifications and open the base network, a scenario or modification in the Network editor (see "Project explorer toolbar" on page 1138). You may also select multiple entries and edit them using the toolbar buttons and the shortcut menu.

- Open the project file of your choice. This may be:
  - an **.inpx* file placed under scenario management
  - a base network (see "Opening and editing the base network in the network editor" on page 1146)
  - a scenario (see "Opening and editing scenarios in the network editor" on page 1147)

  In the network editor, the network is displayed. The project explorer opens. In the project explorer, a treeview of the following elements is displayed:

Element	Description
<Project name>	Folder and name of the project under which the base network and first scenario are displayed. Vissim adopts the name from the **Project name** box, in the **Place Under Scenario Management** window.
Base network	The original network placed under scenario management. In the base network, you can open and edit the base network (see "Opening and editing the base network in the network editor" on page 1146).
Scenarios	**Scenarios** folder under which the scenarios of the projects are displayed.

Element	Description
1 Scenario 1	Number and name of the project scenarios. You cannot change the number. For a scenario, you can execute the following functions: - Open and edit a scenario in the network editor (see "Opening and editing scenarios in the network editor" on page 1147) - Duplicate or delete a scenario (see "Project explorer toolbar" on page 1138) - Copy a scenario into one or several other scenarios (see "Project explorer toolbar" on page 1138) - Rename the scenario (see "Editing the project structure" on page 1139)
Modifications	**Modifications** folder under which all modifications of a project are saved that were performed in scenarios. In the project explorer, under **Modifications**, modifications are only shown after you have changed and saved a scenario. Each modification is saved to a **.trax* model transfer file. In the project structure, you can show the name of the model transfer file of a modification (see "Editing the project structure" on page 1139).
<Name of modification>	Name of a modification. A modification may be listed under the following elements: - under a scenario in which it was created (see "Opening and editing scenarios in the network editor" on page 1147) - under a scenario it was saved to from another scenario or the list of modifications (see "Project explorer toolbar" on page 1138) - under the **Modifications** folder, as soon as you change and save a scenario (see "Opening and editing scenarios in the network editor" on page 1147) - under the **Modifications** folder, when you create a new modification under it (see "Creating a new modification" on page 1146)
	For a modification, you can execute the following functions: - Open and edit the modification in the network editor (see "Opening and editing modifications in the network editor" on page 1148) - Duplicate or delete the modification (see "Project explorer toolbar" on page 1138) - Copy the modification into one or several scenarios (see "Project explorer toolbar" on page 1138) - Rename the modification (see "Editing the project structure" on page 1139)

13.3 Project explorer toolbar

You can use the project explorer toolbar functions to manage the base network, scenarios and modifications of your project:

Element	Name	Description
	Project Structure	Opens the **Project Structure** window (see "Editing the project structure" on page 1139).
	Open	In the network editor, opens the base network, scenario or modification selected in the project explorer (see "Opening and editing the base network in the network editor" on page 1146), (see "Opening and editing scenarios in the network editor" on page 1147), (see "Opening and editing modifications in the network editor" on page 1148).
	Save in highlighted scenarios	For a selected modification: Open the **Select scenario** window. Select at least the current scenario and other scenarios. Then assign the modification to the scenarios selected. Vissim adds the modification under **Scenarios**, under the selected scenarios.
	Add	Creates a new scenario or modification: ▸ If **Scenarios** is selected, Vissim adds a new scenario that is based on the base network under the previous ones. ▸ If **Modifications** is selected, Vissim adds a new modification that is based on the base network under the previous ones.
	Duplicate	Copies the selected scenario or modification. ▸ Vissim adds a copied scenario under **Scenarios**. The number is incremented to the next available number. The name is adopted and extended with the extension *- Copy*. ▸ Vissim adds a copied modification under **Modifications**. The number is incremented to the next available number. The name is adopted and extended with the extension *- Copy*.
	Delete	Deletes the selected scenario or modification.

<table>
<tr><th>Element</th><th>Name</th><th>Description</th></tr>
<tr><td></td><td>Calculate selected scenarios</td><td>The scenarios selected are simulated in succession. In the Project explorer, next to each scenario, the current status is displayed:
<table>
<tr><td></td><td>The simulation run has been started for the scenario.</td></tr>
<tr><td></td><td>The scenario is selected for a simulation run. The simulation run will be started as soon as the previously performed simulation run is completed.</td></tr>
<tr><td></td><td>The simulation run for the scenario has been completed.</td></tr>
<tr><td></td><td>The simulation run was canceled.</td></tr>
</table>
Once a simulation run for a scenario has been started, in the Project explorer, you will not be able to carry out certain commands, for example Open, Save, Add, Duplicate, Delete, or start another simulation run.</td></tr>
<tr><td></td><td>Calculate all scenarios</td><td>All scenarios listed the Project explorer are simulated in succession. In the Project explorer, next to the scenarios, their current status is displayed.</td></tr>
<tr><td></td><td>Cancel scenario simulation</td><td>The simulation run started is stopped. The simulation run for the respective scenario is not completed. No other scenarios are simulated.</td></tr>
</table>

Tip: Alternatively, you can call this function in the shortcut menu of the project explorer. To do so, in the project explorer, right-click the element of your choice.

13.4 Editing the project structure

In the project structure, you may edit the project's basic settings, including the properties of scenarios and modifications.

13.4.1 Editing basic settings

In the basic settings, you may edit, create and delete log entries manually. Log entries that were created automatically by Vissim contain data on the scenarios and modifications created. You may add a log entry to the list and thus document a process step manually.

1. From the **File** menu, select > **Scenario Management** > **Project Structure**.

   *The **Project Structure** window opens.*

2. Select the **Basic settings** tab.

   Data on the creation of scenarios and modifications as well as manual log entries (if applicable) will be displayed.

The tab contains the following attributes:

Element	Description
Project name	Name of the project
Log	Overview of log entries. Vissim creates a log entry whenever a scenario or modification is created. The list contains the following attributes: • **Number**: Total number of log entries • **No**: Consecutive numbers created by Vissim based on the order in which the log entries were created by Vissim or added manually. • **Date**: Date and time of creation • **Entry**: Description which includes the number and name of the created scenario or modification, for example.

3. If desired, edit the project name, date, time and description in the **Entry** column.

   Use the buttons shown to the right of the list to execute the following functions:

Element	Name	Description
	Add log entry	Document an individual process step: adds a new row to the **Log** list and enters the current date and time. Enter a description in the **Entry** column.
	Delete selected log entries	Selecting and deleting one or more rows

13.4.2 Editing scenario properties

1. From the **File** menu, select > **Scenario Management** > **Project Structure**.

   *The **Project Structure** window opens.*

2. Select the **Scenarios** tab.

   All project-related scenarios are displayed, including their attributes.

The list contains the following attributes:

Element	Description
Count	Total number of project-related scenarios
No	Consecutive numbers created by Vissim based on the order in which the scenarios were added.
Name	Name of the scenario. You may rename the scenario.
Description	Optional description of the scenario
Directory	Path that the file of the scenario is stored to

Element	Description
Modifications	Number of modifications allocated to the scenario. If you want to allocate or undo a modification, go to the **Modifications** column and click the button. In the list, click the modification for the scenario of your choice.
Concatenate:ModifComplete\No	**Concatenate:Modifications (complete)\Number**: Consecutive numbers created by Vissim based on the order in which the modifications were added.
Concatenate:ModifCompleteByLoadOrder\No	**Concatenate:Modifications (complete) by load order\Number**: Consecutive numbers created by Vissim based on the order in which the modifications were added.
ScenToComp	Numbers and names of scenarios whose attribute values you want to compare with those of the current scenario. To select or deselect a scenario, in the **ScenToComp** column, click the button. Then in the list, click the scenarios you want to load into the background (see "Comparing scenarios" on page 1148), (see "Selecting scenarios for comparison" on page 1148).

Use the buttons shown to the right of the list to execute the following functions:

Element	Name	Description
	**Add new scenario**	A new row is added to the **Scenarios** list and the current date and time are entered. ➤ In the **Name** column, enter the name of the new scenario. ➤ In the **Description** column, enter the description of the scenario.
	**Duplicate selected scenarios**	A duplicate of the selected scenario is created in the **Scenarios** list. Rename the new scenario in **Name** column, if desired. The name is adopted for the duplicate and extended with the extension `- Copy`.
	**Delete selected scenarios**	Selecting and deleting one or more rows

Element	Name	Description
	**Export selected scenarios**	A window opens. Here you can select the desired directory path where you can store the scenario as **.inpx* file. This **.inpx* file is not part of the project and is not listed under the scenario management.
	**Select attributes to display in the grid**	Columns with attributes are shown and hidden in the **Scenarios** list

13.4.3 Editing modification properties

1. From the **File** menu, select > **Scenario Management** > **Project Structure**.

   The ***Project Structure*** *window opens.*

2. Select the **Modifications** tab.

   All project-related modifications are displayed together with their attributes.

The list contains the following attributes:

Element	Description
Count	Total number of modifications within the project
No	Consecutive numbers created by Vissim based on the order in which the modifications were added.
LoadIndex	**Load index**: Consecutive numbers created by Vissim based on the order in which the modifications were added.
Name	Name of the modification. You can rename the default entry **<generated automatically>** or an entered name.
Description	Optional description of the modification
Group	Name of a group of related modifications. It is used to identify modifications in a list. To do so, you can sort the (see "Sorting lists" on page 106) column.
TraFilename	**Model transfer file name**: Name of the **.trax* file storing the changes that are part of the modification.
Exclusions	Modifications which exclude each other cannot be allocated to the same scenario.
DependOn	**Dependent on**: Once a modification is created, this modification is related to the modifications which have already been allocated to the base network.

Use the buttons shown to the right of the list to execute the following functions:

Element	Name	Description
	**Add new modi-fication**	A new model transfer file is created. A new row is added to the **Modifications** list. Edit the fields in the **Name**, **Description**, **Exclusions** and **DependOn** columns.

Element	Name	Description
	**Duplicate selected modifications**	A duplicate of the selected modification is created in the **Modifications** list. Edit the fields in the **Name**, **Description**, **Exclusions** and **DependOn** columns.
	**Delete selected modifications**	Selecting and deleting one or more rows
	**Move selected modifications upwards in loading sequence**	Moving selected modifications upwards in loading sequence and adjusting the number in the **LoadIndex** column.
	**Move selected modifications downwards in loading sequence**	Moving selected modifications downwards in loading sequence and adjusting the number in the **LoadIndex** column.
	**Check combinability of selected modifications**	Vissim checks whether a batch of selected modifications can be uploaded to a scenario or whether this is excluded in terms of functionality. A message is displayed and shows the results of the check made.
	**Check independence of selected modifications**	Vissim checks whether the selected modifications result in networks that are modeled in different ways when changing their loading sequence. ➤ If changing the loading sequence does not result in networks that are modeled in different ways, the modifications do not relate to each other. This means entries in the **DependOn** column have no effect. A message is opened. You can choose to delete the relations entered in the **DependOn** column. ➤ If changing the loading sequence result in networks that are modeled in different ways, the modifications relate to each other. A message is displayed and shows the results of the check made. You may reselect the relations between the modifications. The matrix file is shown in the **DependOn** column.
	**Selecting attributes to be displayed in the grid**	Columns with attributes are shown and hidden in the **Modifications** list.

13.5 Placing a network under scenario management

Warnings:

- Do not change the names or structure of the files and directories generated by Vissim.
- Do not save the files of a project to the directory of another project.

In both cases, scenario management cannot work properly, which may lead to loss of data!

Place your network as a base network under scenario management to derive other networks from it, save these as scenarios and make further changes to them. The base network may already contain a modeled network or be empty. You cannot use the base network to perform simulation runs.

1. Ensure that the network of your choice has been modeled according to your requirements and has been saved as a **.inpx* network file.
2. Ensure that the network file is not saved to a folder that already contains a project or part of a project.
3. From the **File** menu, choose > **Scenario Management** > **Place Under Scenario Management**.

   *The **Place Under Scenario Management** window opens.*
4. Enter the desired data.

Element	Description
Project name	Name of the project to which the base network and first scenario are saved.
First scenario	Name of the first scenario saved as a copy of the base network to the project.

5. Confirm with **OK**.

   In the network editor, the first scenario is displayed. The title bars and frames of non-selected elements of the program interface, e.g. of the network editor, project explorer or quick view are no longer displayed in blue (default color), but in a different color, now green by default, as a scenario has been opened.

   *The title bar of Vissim displays the name of the project and the first scenario. The project explorer opens (see "Using the project explorer" on page 1136). In the directory that contains the *.inpx network file, Vissim creates the following directories and files for scenario management.*

Warning: Avoid deleting, moving, editing or other changes to these directories and files. Otherwise, scenario management cannot work properly, which may lead to loss of data!

- *Backups*: used internally by Vissim
- *Modifications*: contains **.trax* model transfer files of the modifications
- *Scenarios*: When a scenario is simulated and direct output is selected for evaluations, Vissim saves the direct output files to subfolders of this directory.
- *Temp*: contains backup files

If you have generated files using Direct Output of Evaluations, these files remain saved in your directory and are not automatically moved before you place the network file under Scenario Management.

If when editing the network file, you have saved or generated the following files within Vissim before placing the network file under Scenario Management, these files are not automatically moved by Vissim when you place the network file under Scenario Management:

- SC control files
- Fixed time control files **.sig*
- Animation files **.ani*
- Video files **.avi*

*When you place a network file that contains evaluation results, path files *.way, trip chain files *.fct or cost files *.bew under scenario management, these are assigned to an automatically created Scenario 1 and are saved to its folder under the* ***Scenarios*** *directory.*

If when editing the network file, you have created a results database, this database is automatically moved by Vissim to the ***Scenarios*** *directory before you place the network file under Scenario management.*

13.6 Creating a new scenario

You have the following options for creating a new scenario:

- In the base network: You adopt all settings, network objects and base data from the base network into the new scenario.
- In the project structure: Here you can edit the attributes of the scenario and select the modifications of your choice (see "Editing the project structure" on page 1139).
- As a duplicate of a scenario in the project explorer or project structure (see "Project explorer toolbar" on page 1138), (see "Editing the project structure" on page 1139)

13.6.1 Creating a new scenario in the base network

1. In the project explorer, open the project of your choice.
2. In the project explorer, right-click **Scenarios**.
3. From the shortcut menu, choose **Add**.

The new scenario is shown in the project explorer, in the ***Scenarios*** *folder (see "Using the project explorer" on page 1136). In the network editor, you can open and edit the scenario (see "Opening and editing scenarios in the network editor" on page 1147). You can rename* *the scenario, enter a description for it and assign it modifications (see "Editing the project structure" on page 1139).*

13.7 Creating a new modification

You have the following options for creating a new modification:

- In the base network: Adopt all settings, network objects and base data from the base network into the new modification.
- In the project structure: Vissim creates a model transfer file (**.trax*) for changes to the modification and you can edit the attributes of scenarios (see "Editing the project structure" on page 1139).
- As a duplicate of a modification in the project explorer or project structure (see "Project explorer toolbar" on page 1138), (see "Editing the project structure" on page 1139)

13.7.1 Creating a new modification in the base network

1. In the project explorer, open the project of your choice.
2. In the project explorer, right-click **Modifications**.
3. From the shortcut menu, choose **Add**.

*The new modification is shown in the project explorer, in the **Modifications** folder (see "Using the project explorer" on page 1136). You can open and edit the modification in the network editor and copy it into one or multiple scenarios (see "Opening and editing scenarios in the network editor" on page 1147), (see "Using the project explorer" on page 1136). You can rename the modification, enter a description and make additional settings (see "Editing the project structure" on page 1139).*

Tip: Alternatively, you can create a new modification in the project structure (see "Editing the project structure" on page 1139).

13.8 Opening and editing the base network in the network editor

In the network editor, you can open and edit the base network of a project. In the base network you cannot perform simulation runs. Vissim automatically adopts changes made to the base network as modifications into all scenarios of the project, as these have been derived from the base network. In the project explorer, under Scenarios and under **Modifications**, the modifications are displayed.

1. From the **File** menu, choose > **Scenario Management** > **Open Base Network**.

Tip: Alternatively, in the project explorer, right-click **Base network**. Then click **Open**. If under the **File** menu, in the list of files last opened, the file is shown and the path has not been changed, you can select the file there and show it in the network editor.

In the network editor, the base network is opened. The title bars and frames of non-selected elements of the program interface, e.g. of the network editor, project explorer or quick view are no longer displayed in blue (default color), but in olive green by default.

2. Edit the base network.
3. Use one of the following options to save the base network:
   - Press CTRL+S.
   - On the **File** toolbar, click **Save**.

Your changes are adopted in all scenarios of the project.

13.9 Opening and editing scenarios in the network editor

In the network editor, you can open and edit scenarios of a project. In a scenario, you can perform simulation runs. If multiple scenarios have been saved under a project and you edit and save one of them, Vissim will add it as a modification under the edited scenario and under **Modifications**.

1. Make sure no other simulation diagram is open.
2. From the **File** menu, choose > **Scenario Management** > **Open Scenario**.

Tips: Alternatively, in the Project Explorer, open a scenario as follows:
- Double-click the desired scenario.
- Right-click the desired scenario and from the shortcut menu, choose **Open**.
- If under the **File** menu, in the list of files last opened, the file is shown and the path has not been changed, you can select the file there and show it in the network editor.

   *The **Open Scenario** window opens.*

3. Click on the desired entry.

4. Confirm with **OK**.

   *If the **Messages** window opens, check the messages and correct the errors reported. Messages regarding modifications are listed under Modification.*

   In the network editor, the scenario is opened. The title bars and frames of non-selected elements of the program interface, e.g. of the network editor, project explorer or quick view are no longer displayed in blue (default color), but in green by default.

5. Edit the scenario.
6. Use one of the following options to save the scenario:
   - Press CTRL+S.
   - On the **File** toolbar, click **Save**.

*Changes are saved to the scenario and displayed in the project explorer as a modification under Scenarios and under **Modifications**.*

13.10 Opening and editing modifications in the network editor

In the network editor, you can open and edit modifications of a project. When you edit and save a modification, all changes become effective in the scenarios that access the edited modification.

1. From the **File** menu, choose > **Scenario Management** > **Open Modification**.

Tip: In the project explorer, right-click the scenario of your choice to open the Vissim network in the network editor.

*The **Open Modification** window opens.*

2. Click on the desired entry.
3. Confirm with **OK**.

In the network editor, the modification is opened. The title bars and frames of non-selected elements of the program interface, e.g. of the network editor, project explorer or quick view are no longer displayed in blue (default color), but in maroon by default.

4. Edit the modification.
5. Use one of the following options to save the modification:
   - Press CTRL+S.
   - On the **File** toolbar, click **Save**.

The changes are saved to the modification. The modification then becomes effective in all scenarios that reference the modification.

13.11 Comparing scenarios

You can compare the attributes and attribute values of network objects of an opened scenario with those of one or multiple other scenarios (comparison scenarios) (see "Selecting scenarios for comparison" on page 1148).

In the currently opened scenario, in the attribute list of each network object type, you can show the scenarios in which the network objects of this network object type were used (see "Selecting attributes and subattributes for columns of a list" on page 112). To do so, select at least one scenario for comparison with the opened scenario (see "Selecting scenarios for comparison" on page 1148).

13.11.1 Selecting scenarios for comparison

In the project structure, for each scenario, you can select one or multiple scenarios for comparison from the same scenario management project. This way you have one base scenario that is assigned to one or multiple comparison scenarios.

The network of the comparison scenario selected loads Vissim into the background. Based on the network of the base scenario, relations to networks loaded into the background are generated. When you open the base scenario, in the Attribute selection window, you can select attributes and attribute values of network objects or comparison scenarios and display them in the attribute lists of network objects of the base scenario to compare them (see "Selecting attributes for scenario comparison" on page 1149).

1. From the **File** menu, select > **Scenario Management** > **Project Structure**.
2. Select the **Scenarios** tab.
3. Click into the row of the scenario you want to compare with other scenarios. Then in the **ScenToComp** column, click the button.
4. In the list, click the scenarios you want to use for comparison.
5. Click the **Close** button.

   The scenarios selected (comparison scenarios) are loaded into the background.
6. Open the base scenario.
7. Open the attribute list of the network object type whose attributes you want to compare (see "Opening lists" on page 95).
8. Select the attributes you want to compare (see "Selecting attributes for scenario comparison" on page 1149).

13.11.2 Selecting attributes for scenario comparison

For network objects that exist in the current as well as in comparison scenarios you can: a) in the Attribute selection window, select attributes of the network objects of scenarios b) in the attribute lists of the network objects of the current scenario, show and easily compare them.

You have the following options to select subattributes:

- select and apply individual subattributes from one or multiple scenarios
- selecting individual subattributes from one or multiple scenarios and apply them from other scenario comparisons

13.11.2.1 Select individual subattributes and apply them in the attributes list

1. Open the base scenario you want to compare with another scenario (see "Opening and editing scenarios in the network editor" on page 1147).
2. For the base scenario opened, select the desired comparison scenarios (see "Selecting scenarios for comparison" on page 1148).
3. Open the attribute list of the network object type whose attributes you want to compare (see "Opening lists" on page 95).
4. In the attribute list, click the **Attribute selection** symbol .

   The ***<Network object type>: Select Attributes*** *window opens. In the section on the left, the attributes of the network object type are displayed. The scenarios are marked with the symbol and sorted by their number.*

5. In the section on the left, in front of the desired entry **Scenario comparison <Name of base scenario> <Name of comparison scenario>**, click the **+** symbol.

   The following attributes are displayed under scenario comparison. The attributes are highlighted with a red circle. You must select at least one subattribute. They cannot be edited.

Attribute	Description
Value	Attribute value of network object in comparison scenario
<Name of base scenario> minus <Name of comparison scenario>	Difference of attribute values between base scenario and comparison scenario
<Name of comparison scenario> minus <Name of base scenario>	Difference of attribute values between comparison scenario and base scenario
<Name of base scenario> minus <Name of comparison scenario> %	Difference of attribute values between base scenario and comparison scenario in percent
<Name of comparison scenario> minus <Name of base scenario> %	Difference of attribute values between comparison scenario and base scenario in percent
Minimum	Smallest attribute value of network object in comparison scenario
Maximum	Largest attribute value of network object in comparison scenario

6. Under scenario comparison, in front of the attribute of your choice, click the **+** symbol.

   All attributes of the network object type are displayed.

7. Select the desired attributes whose values you want to compare with the values of the same attributes in the base scenario.

   *If under an attribute, you select the subattribute **Current run**, **Current run** will show the following the simulation run:*

   - In a comparison scenario, the last simulation run completed
   - in the currently loaded base scenario, the active, current simulation run
   - If no simulation is run in the base scenario, the last simulation run completed.

   *If simulations were run in the comparison scenarios, you can select subattributes from the desired simulation runs for the comparison. The subattributes begin with the number of the respective simulation run, e.g. **4 x current x All types**.*

8. Click the symbol.

   The attributes are displayed on the right in additional rows.

9. Confirm with **OK**.

   The attributes are displayed in the attribute list of the network object type.

13.11.2.2 Also applying subattributes selected from all scenario comparisons

1. Select the desired subattributes as described above.
2. Click the **Add from all scenario comparisons** button.

13.11.2.3 Showing comparison scenarios in which the selected network object is used

In an opened scenario, in the attribute list of a network object type, you can show the scenarios assigned for comparison that use network objects of the same network object type. To do so, in the Attribute selection window, on the left, select the desired comparison scenario and add it to the right side (see "Selecting attributes and subattributes for columns of a list" on page 112). This is also possible in the result lists **Simulation Runs**, **Vehicles In Network**, **Pedestrians In Network**, and for dynamic assignment, in the **Paths** list.

1. Opens the attribute list of the network object type in which you want to show the columns with scenarios that include the network object type (see "Opening lists" on page 95).
2. In the attribute list, click the **Attribute selection** symbol.

   *The **<Network object type>: Select Attributes** window opens. In the section on the left, the attributes of the network object type are displayed.*
3. In the section to the left of **Exists in scenario**, click the **+** symbol.

   *All comparison scenarios with a network object of the network object type are displayed. The scenarios to compare are marked with the symbol and sorted by number: **Scenario comparison <Name Scenario>**.*
4. Select the desired comparison scenario.
5. Click the symbol.

   The selected scenarios to compare are listed on the right in an additional row. You cannot edit hatched cells or the attribute name.
6. Confirm with **OK**.

*In the attribute list, an **ExistsInScen,<Number>** column is displayed for each comparison scenario selected. In this column, in each row and for each network object that exists in the respective comparison scenario, the option is selected.*

13.12 Comparing and transferring networks

You can save the differences between two networks.to a model transfer file (**.trax*). Generally, one network models the base, while the other network models the target. One of the networks can be the one currently opened.

You apply the model transfer file (**.trax*) to a network file. The network file must be opened in the Network editor. When applying the model transfer file (**.trax*), you transfer its network data (**.trax*) to the network opened in the Network editor.

Application example for Scenario management

You can transfer two network variants to Scenario management: The network with the base is placed under Scenario management. You then apply the model transfer file to this network. This is how the second network selected is created:

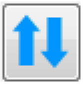
1. Generate the model transfer file (**.trax*) based on network **A** and **B** (see "Creating model transfer files" on page 1152).
2. Then you place network **A** under Scenario management as scenario 1 (see "Placing a network under scenario management" on page 1144)..
3. Copy Scenario 1.
4. Rename the copied scenario to Scenario 2.
5. Open scenario 2 in the Network editor.
6. Apply the model transfer file to scenario 2 (see "Applying model transfer files" on page 1153).

13.12.1 Creating model transfer files

1. Open the network file you want to use as base file to create the model transfer file.
2. From the **File** menu, choose > **Compare and Transfer Networks** > **Create Model Transfer File**.

   *The **Create Model Transfer File** window opens.*
3. Make the desired changes:

Element	Description
Define network comparison	**Base**: ➤ **Use currently loaded network**: ☑ Select this option to use the network currently opened as the basis for the model transfer file. ➤ **Read from file**: Click the 🗁 symbol to choose another file as the basis for the model transfer file you want to create. **Target**: ➤ **Use currently loaded network**: ☑ Select this option if the currently opened network is the desired target you want to create with the help of the model transfer file to be created. Both files must use the same coordinate system. ➤ **Read from file**: Click the 🗁 symbol to select the file of your choice. The model transfer file created is used to transfer the base network into the selected network.

	**Swap base and target**: The entries made and options selected in the **Base** and in the **Target** are swapped
Result of the com-parison	**Write model transfer file to**: Click the  symbol, enter the filename of the model transfer file (**.trax*), select the folder of your choice and save the model transfer file to it (**.trax*).

4. Click the **Create Model Transfer File** button.

13.12.2 Applying model transfer files

You can apply a saved model transfer file (**.trax*) to the network file currently opened. In doing so, you transfer the network data to the network opened in the Network editor.

1. Open the network file to which you want to apply the model transfer file.
2. From the **File** menu, choose > **Compare and Transfer Networks** > **Apply Model Transfer File**.

   *The **Choose model transfer file** window opens.*
3. Select the desired file.
4. Click the **Open** button.
5. Check any messages displayed by Vissim.

14 Testing logics without traffic flow simulation

You can test the response of a signal control logic to several detector type constellations, without actually modeling traffic flows.

You have the following options to generate detector types:

- Add types to the detectors.
- Run recorded or created macros.

The **Test** function is useful, if you want to check for bugs in newly developed signal control logics. This is particularly true when they contain functions that are used only sporadically.

The following detector types are distinguished:

Detector type	Description
Single actuation	Increasing impulse (front end of vehicle) and decreasing impulse (rear end of vehicle) within one second
Repeated actuation	Increasing and decreasing impulse every second, equivalent to a single actuation every time step.
Continuous actuation	Single impulse increase, impulse decrease only after explicit termination of actuation.

14.1 Setting detector types interactively during a test run

1. On the Network objects toolbar, click **Vehicles In Network**. Then click the **Edit graphic parameters** icon.

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

2. Select **Object visibility**.
3. Ensure that neither the **Use color scheme** option nor a classification are selected.
4. On the Network objects toolbar, click **Detectors**. Then click the **Graphic parameters** icon.

   The list with the graphic parameters of the network object type opens (see "List of graphic parameters for network objects" on page 161).

5. Select **Object visibility**.

Tips:

- You can save a Signal Control Detector Record as a file or display it in a window (see "Evaluating SC detector records" on page 1070).
- You may record an event for comparison with another control logic (see "Using macros for test runs" on page 1155).

---

6. From the **Test** menu, choose > **Continuous** or **Single Step**.

The test run is started. During the test run, you can switch back to the ***Test run single step*** *mode, e.g. the simulation second during which startup of the SC program ends.*

Symbol	Name	Description
▶	**Test run continuous**	Starts continuous test run or switches from **Test run single step** mode to continuous mode.
▶❙	**Test run single step**	Starts simulation in **Test run single step** mode or switches from Test run continuous to **Test run single step** mode or executes the next single step.
■	**Stop test run**	Quit test run

Tip: You can set whether you want to activate detectors in test mode with a single or a double click (see "Defining click behavior for the activation of detectors in test mode" on page 154).

7. In the Single Step mode, in the Network Editor, double-click a detector type.

   With each click, you switch to the next requirement:

   - No actuation: no fill color
   - Single actuation: blue
   - Repeated actuation: turquoise
   - Continuous occupancy: pink

   If you double-click on Continuous occupancy, the detector call switches back to ***No actuation****.*

Tip: Alternatively, right-click the detector and from the shortcut menu, choose **Edit Actuation**. Then click the state of your choice.

14.2 Using macros for test runs

You do not need to manually and interactively set each individual detector call in the Network Editor in each test run (see "Setting detector types interactively during a test run" on page 1154). You can perform test runs with macros for which you use different parameter settings of logic in each case. The desired detector types are saved to **.m_i* macro files.

14.2.1 Recording a macro

You can record a macro file for identical test runs of several control scenarios.

1. From the **Test** menu, choose > **Macro Recording**.

   The entry is marked with a checkmark the next time you open the menu.

2. From the **Test** menu, choose > **Continuous** or **Single Step**.

3. Set the desired detector types in the Network Editor (see "Setting detector types interactively during a test run" on page 1154).
4. From the **Test** menu, choose > **Stop**.

   The test run is completed. A macro file <name of network file>.M_I is saved.
5. Change the parameters of the control logic for the next recording.
6. From the **Test** menu, choose > **Run Macro**.

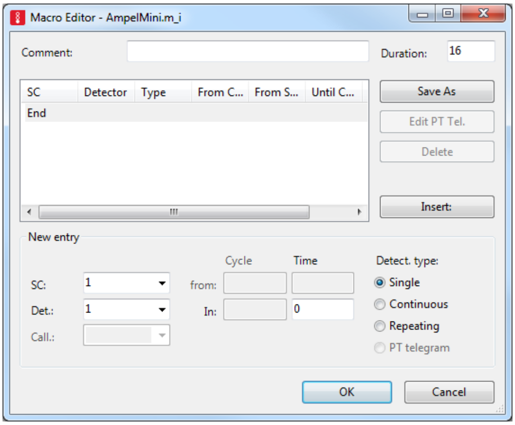
   The repetition of the same calls starts.

14.2.2 Editing a macro

To evaluate a control logic with similar test runs, you can create macro files in the Macro Editor. If you want to edit an existing call, for example, change the time interval, you must delete the call and add a new call.

1. From the **Test** menu, choose > **Edit Macro**.
2. Select the macro file **.m_i*.

   The Macro Editor opens. The existing calls are shown in the table.



3. Make the desired changes:

Element	Description
SC	Signal controller of the call
Det.	Detector of the call
Call.	PT calling pt: only records PT vehicles that send PT telegrams.
Cycle from, to	Number of cycle

Element	Description
Time from, to	Time interval
Detect. type	- **Single**: single calls work only within a second. You can thus only specify the time **in**. - **Continuous**: permanent occupancy - **Repeating**: one crossing per second

4. If you have defined a new entry, click **Insert**.

   The new call is added before the currently marked call. Automatic sorting does not take place.

5. Enter another file name for the new test case.
6. Confirm with **OK**.

Note: If the current control supports PT calling points, PT telegrams can be recorded in the macro as special calls for detectors of type **PT telegram**.

14.2.3 Run Macro

Perform the following steps for each test run.

1. Record the macro file.
2. If desired, edit the macro file.
3. Adjust the parameters of the control logic for the test case.
4. If desired, activate evaluations (see "Evaluating SC detector records" on page 1070).
5. From the **Test** menu, choose > **Run Macro**.

   *The **Run Macro File** window opens.*

6. If desired, into the **To second** box, enter the simulation second up to which you want to macro to run.
7. If you have enabled evaluations, save and analyze the evaluations.

Notes:

- Vissim does not automatically name the files of different test cases. This gives you flexibility when naming. Give the files useful and unique file names, for example, include the test case number.
- If different test cases are examined when testing VS-PLUS logics, an SC detector record is created in a file **.ldp* for each test case. Give this file the name of the **.m_i* macro file which was used to generate the **.ldp* file. This improves the clarity and makes it easier to find the files.

15 Creating simulation presentations

You can record simulations for presentations in the following data formats and save them in files:

- 3D video files **.avi* (see "Recording a 3D simulation and saving it as an AVI file" on page 1158)
- animation files **.ani* (see "Recording a simulation and saving it as an ANI file" on page 1166)

15.1 Recording a 3D simulation and saving it as an AVI file

Vissim can be used to save a 3D simulation as a video file in the **.avi* format. You need to specify camera positions in order to assign them to keyframes in a storyboard (see "Saving camera positions" on page 1158), (see "Using storyboards and keyframes" on page 1160).

You can also save the 3D perspective of a driver or a pedestrian as camera position (see "Showing 3D perspective of a driver or a pedestrian" on page 196).

You can also start the recording in the Quick Mode (see "Using the Quick Mode" on page 89). The Quick Mode is ignored during the recording.

Note: Video files require a lot of memory. Make sure that depending on the recording planned, sufficient memory space is available.

15.1.1 Saving camera positions

To record a simulation as an AVI file, you need to save the camera positions with the network view of your choice. You then assign keyframes to the camera positions.

You can also save a camera position during a simulation run. When doing so, you can save the 3D perspective of a driver or a pedestrian as camera position as well (see "Showing 3D perspective of a driver or a pedestrian" on page 196).

1. Make sure you have selected the 3D mode.

   The symbol is selected. The network is displayed in 3D. You do not need to perform the next step if you want to save the camera position during a simulation run, e.g., to save a 3D perspective of a driver or a pedestrian as camera positions.

2. To set the desired camera position, click the **Rotate mode (3D)** symbol or use the mouse or keyboard.
3. On the Network editor toolbar, in the **Selection of camera position** list, enter the name of your choice.
4. Confirm with ENTER.

Tip: You can edit the attributes in the **Camera Positions** list (see "Attributes of camera positions" on page 1159).

15.1.2 Attributes of camera positions

1. From the **Presentation** menu, choose **Camera Positions**.

   *The **Camera Positions** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. A network object may have the following attributes: These can be shown in the attributes list.

Element	Description
No	Number of camera position
Name	Name of camera position. Changing the name of a camera position automatically changes the name of the keyframes assigned to it, if the keyframes have the same name as the camera position.
CoordX	**Coordinate (x)**: horizontal camera position on the x-axis
CoordY	**Coordinate (y)**: horizontal camera position on the y-axis
CoordZ	**Coordinate (z)**: vertical camera position on the z-axis. Showing 3D perspective of a driver or a pedestrian, this is at eye level. The eye level depends on the model: ▶ Vehicles, with the exception of HGVs, buses and trains: default 1.5 m ▶ HGVs, buses and trains: default 2.5 m ▶ Vehicle types, **man**, **woman**: default 1.75 m ▶ Pedestrians: 92.5 % of height of 3D model
CoordPt	**Coordinates**: Coordinates of the camera position **Coordinate (x)**, **Coordinate (y)** and **Coordinate (z)**. Showing the 3D perspective of a driver or a pedestrian: 0, 0, eye level.
YawAngle	**Yaw angle**: Angle of rotation around z axis. Showing 3D perspective of a driver or a pedestrian: 0°
PitchAngle	**Pitch angle**: Angle of rotation around y axis. Showing 3D perspective of a driver or a pedestrian: 10°
RollAngle	**Roll angle**: Angle of rotation around x axis. Showing 3D perspective of a driver or a pedestrian: 0°
FOV	**Field of View**: Aperture angle of the camera in degrees from viewer's position
FocLen	**Focal length** in mm
VehNo	**Vehicle number**: Number of vehicle selected for camera position
PedNo	**Pedestrian number**: Number of pedestrian selected for camera position

3. If during editing you have changed the display of the Vissim network and want to show the entire Vissim network again, on the Network Editor toolbar, click the **Show entire network** button.

15.1.3 Using storyboards and keyframes

A storyboard allows you to define basic settings for the recording of a simulation. These, e.g., include the **resolution**, **framerate** and the option of saving the recording to an **.avi* file. You assign at least one keyframe to your storyboard. In a storyboard, you can combine multiple keyframes that then elapse in sequence. You assign a camera position to each keyframe. This allows you to specify the order of the camera positions used during simulation. The camera dwells on the keyframes for the **dwell time** specified. It moves between the keyframes on a linear guide, with a constant or gliding motion, for the **transition time** specified.

You are basically using keyframes as a “script" for the AVI recording. During the AVI recording, the keyframes elapse in the sequence specified in the **Start time** attribute. The recording begins with the first start time. It does not have to be the start time of the simulation. As soon as the AVI recording is started, the view switches to the camera position of the first keyframe. To record **.avi* files, you must define at least one keyframe with a **camera position** and a **dwell time** and assign it to a storyboard.

During a simulation run you can:

- Editing attributes of keyframes (see "Defining a storyboard with keyframes" on page 1160)
- add new camera positions (see "Saving camera positions" on page 1158)
- Editing attributes of camera positions (see "Attributes of camera positions" on page 1159)
- Showing a preview of the movie in a window in simulation speed (see "Showing a preview of camera movement" on page 1164)

Tip: You can additionally read Storyboards like other network objects and base data from other Vissim **.inpx* network files (see "Reading a network additionally" on page 361).

15.1.3.1 Defining a storyboard with keyframes

1. Make sure you have selected the 3D mode.

   The symbol is selected. The network is displayed in 3D.

2. Set the desired camera position, for example, using the icon **Rotate mode (3D)** and the mouse.
3. From the **Presentation** menu, choose **Storyboards**.

   *The **Storyboards** list opens. In the **Relations** list box, the entry **<Single List>** is shown.*

   By default, you can edit the list *(see "Using lists" on page 93).*

4. In the list, on the toolbar, click the **Add** button .

A new row with default data is inserted.

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

5. Make the desired changes:

Element	Description
Resolution	Resolution in pixels **Resolution (x)** x **Resolution (y)**, e.g. 1,280 x 780
ResX	Horizontal **resolution (x)**, e.g. 1,280
ResY	Vertical **resolution (y)**, e,g, 780
RecAVI	**Record AVI**: ☑ Select this option to create an AVI recording for the story-board.
Framerate	**Frame rate (fps)**, default 20 images per second
Filename	Path and file name of AVI file
RealTmFact	**Real time factor** = $Framerate / Simulation\ resolution$ (see "Defining simulation parameters" on page 840). Default 2.0
NetLayout	Select the named network editor layout for the beginning of the story-board (see "Using named Network editor layouts" on page 86)
Name	Name of storyboard
No	Storyboard number
ShowPrev	**Show preview**: While recording a simulation, you can show a preview of the film in a window.
PrevZoomFact	**Preview zoom factor** for film preview (default value 1)

You can define keyframes for the storyboard.

6. In the **Relations** list box, click **Keyframes**.

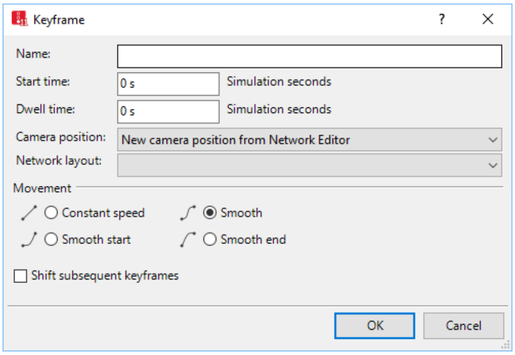
   The right-hand list is shown. If there is no assignment, only the column titles are shown.

7. In the list on the left, click the storyboard for which you want to define keyframes.

8. In the list on right, on the toolbar, click the **Add** button ✚ .

   A new row with default data is inserted.

   *The **Keyframe** window opens.*



9. Make the desired changes:

Element	Description
Name	Name of keyframe. ▶ When you define a keyframe without specifying a name for it and then select a camera position, the name of the camera position is entered as the keyframe name in the **Keyframes** list. ▶ If the name of the keyframe and the name of the assigned camera position are the same and you decide to change the name of the camera position, the name of the respective keyframe is automatically changed in the **Keyframes** list. ▶ If no name is entered for the keyframe and you change the name of the assigned camera position, the new camera position name is automatically entered as the keyframe name in the **Keyframes** list. ▶ Changing the name of a camera position automatically changes the name of the keyframes assigned to it, if the keyframes have the same name as the camera position.
StartTime	Simulation second during which the keyframe is started **Start time** $= 0$: Keyframe starts when from the **Presentation** menu, you choose > **AVI Recording**. If **Start time** $= 0$ is not defined for any keyframe, but instead start times > 0 are defined for all keyframes, the AVI recording starts with the keyframe for which the shortest start time has been defined. Changing the **StartTime** also allows you to move the order of the keyframe in the list.
DwellTime	Time period during which simulation is viewed from this keyframe position.
	Note: Vissim uses the **StartTime** and **DwellTime** to check whether the current keyframe fits into the existing keyframe list. You must not enter a **StartTime** or **DwellTime** for a keyframe that overlaps with the **DwellTime** of another keyframe. To move any of the following keyframes, select ☑ **Shift subsequent keyframes**.
CamPos	Camera position (see "Attributes of camera positions" on page 1159) ▶ Adopt new camera position from the active Network editor ▶ Select saved camera position
NetLayout	Select named network editor layout for the beginning of the keyframe (see

Element	Description
	"Using named Network editor layouts" on page 86)
TransTime	**Transition time** between two keyframes, which is automatically calculated as the difference of **StartTime** and **DwellTime** of the current keyframe and the **StartTime** of the next keyframe. Overlapping keyframes cannot be added to the list.
TransType	Transition types, in the **Movement** section, define the motion between the current and next camera position of the keyframe. - **Constant speed**: Position change at constant speed - **Smooth**: Motion close to keyframe positions is slower, but becomes faster between them. - **Smooth start**: Keyframe position starts with increasing speed and continues with constant speed toward the next keyframe. - **Smooth end**: Keyframe position starts with a constant speed and gradually slows down towards the next keyframe position. You can use the latter two options to define a keyframe without **DwellTime** as "intermediate point" between a movement of keyframes, e.g. in order to create an effect on the distance.
	Example transition types **Smooth start**, **Smooth end**: If keyframe 2 is an intermediate keyframe with a **DwellTime** of 0 s, then the movements could be defined as follows: Keyframe 1: Smooth start Keyframe 2: Smooth Keyframe 3: Smooth end
Shift subsequent Keyframes	☑ If this option is selected, the start times of all subsequent keyframes according to the current settings made for the keyframe selected are moved: - When you insert a new keyframe between two defined keyframes, Vissim checks whether the start time of the new keyframe lies after the **DwellTime** of the preceding value. If so, then Vissim changes the start times of all subsequent keyframes: $\Delta t = \textit{start time of new KF} + \textit{DwellTime of new KF} + \textit{movement time of previous KF} - \textit{start time of next KF}$ By adjusting the start time of the next keyframe accordingly, Vissim ensures that the (calculated) movement time is retained until the keyframe following the added keyframe is reached. It thus remains the same as before the new keyframe was inserted. - When you edit an existing keyframe, all subsequent keyframes are moved according to the new start time and/or **DwellTime**. The start times of all subsequent keyframes are moved by:

Element	Description
	Δt = *new start time – old start time + new DwellTime – old DwellTime* Note: ☑ If you select this option, a defined keyframe cannot be moved before any of its previous keyframes.

10. On the **File** toolbar, click **Save**.

15.1.3.2 Showing a preview of camera movement

You can show a preview of the camera movement at the AVI default rate of 20 frames per second or at maximum speed.

1. Open the **Storyboards** list.
2. Make sure that **Keyframes** is selected in the **Relations** list box.
3. In the list on the right, right-click the keyframe of your choice.
4. From the context menu, choose **Preview with AVI speed** or **Preview with simulation speed**.

   *The movement shown when recording the *.avi file is simulated using the speed currently selected.*
5. To cancel and close the preview, in the top right corner of the preview, click the **X**.

   At the end of the AVI recording, the preview is closed.

15.1.4 Recording settings

Vissim generates **.avi* video files that are played at a default rate of 20 frames per second. As each simulation time step generates exactly one picture, the playback speed depends on the number of time steps per simulation second: With a **simulation resolution** of 10 time steps, which is recommended, the playback speed is twice as fast as real time. If you choose to use only one time step per simulation second, the playback speed is 20 times faster than real time.

Note: Please note that changing the simulation resolution has an impact on the behavior of vehicles and pedestrians and thus might lead to different simulation results.

Besides standard recording, you have the following options:

- **Anti-aliasing** (see "Selecting the 3D mode and 3D recording settings" on page 151)
- **Double sided lighting** (see "Selecting the 3D mode and 3D recording settings" on page 151)

15.1.5 Starting AVI recording

1. Ensure that the following requirements are met:
   - You have saved at least one camera position (see "Saving camera positions" on page 1158)

- You have saved at least one storyboard (see "Using storyboards and keyframes" on page 1160).
- The storyboard has been assigned at least one keyframe.
- In the keyframe, the attributes **Start time**, **Dwell time**, **Transition type** and **Transition time** are defined and have been assigned a **camera position**.

2. In the **Storyboards** list, click the storyboards of your choice and select ☑ **RecAVI** (**Record AVI**).
3. From the **Presentation** menu, choose > **Record AVIs**.

   *In the **Presentation** menu, **Record AVIs** is marked with* ☑*.*
4. Start the simulation (see "Running a simulation" on page 840).

   *If you have not started AVI recording for the current network files yet, the window **Video Compression** opens. An *.avi file requires approx. 20 MB/s. On your computer there may be several compression programs installed by default (see "Selecting a compression program" on page 151).*

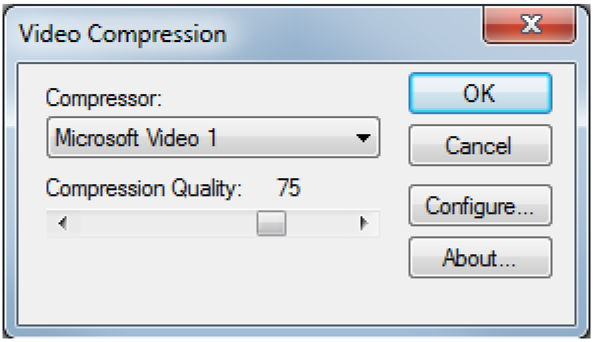
   Codec x264 provides an excellent quality for smaller files x264vfw.sourceforge.net. For further information, please visit our webpages at: PTV Vissim FAQs (#91).

Notes:

- Not all compression programs displayed might be suitable for recording an **.avi* file. This depends on the system configuration. Test the compression program you want to use with a short Vissim example.
- The compression program used for the original video recording must be available on the computer you are using to show the **.avi* file.

---

5. For a minimum file size, in the **Compressor** list, click a compression program.
6. In the **Compression Quality** box, enter a value of your choice.



7. If the compression program can be configured, the **Configure** button is activated and you can make additional settings.
8. Confirm with **OK**.

Tip: You may specify a default compression program (see "Selecting a compression program" on page 151).

*The simulation starts. If the **Preview** option has been selected for the storyboard, a window opens in which the movie is shown. The *.avi file is recorded during the simulation run. At the end of the simulation, the *.avi file is saved in the same folder as the *.inpx network file.*

9. To stop AVI recording before the end of the simulation run, from the **Presentation** menu, choose > **Record AVIs**.

   *In the **Presentation** menu, **Record to AVI** is deactivated. The attribute **RecToAVI** (**AVI recording**) of the storyboard is deactivated.*

10. To stop the simulation, on the **Simulation** toolbar, click the **Stop** button .

Note: The recording of an **.avi* file may take considerably longer than a simulation in 3D mode, particularly if from the **Presentation** menu, you chose > **3D-Anti-Aliasing**. This option corresponds to the **Anti-aliasing** option under User Preferences (see "Selecting a compression program" on page 151).

15.2 Recording a simulation and saving it as an ANI file

You can save simulation runs for vehicles and pedestrians in animation files (**.ani*). After this you can use the animation files for presentations in Vissim. Since an animation file only contains the graphical representation of the simulated vehicles or pedestrians, the animation runs considerably faster than the actual simulation. Evaluations are not available in the animations.

When defining sections, you can chose whether you want to record the simulation for the sections defined or for the entire network.

You can also record simulation runs with the following settings:

- Pedestrian simulations that use the LOS schema (see "Using LOS schemes for showing aggregated pedestrian values" on page 186).
- The wireframe view can be enabled or disabled (see "Network editor toolbar" on page 75)
- Links may be hidden if their display type is set to **Invisible** (see "Defining display types" on page 320).
- Elevators, elevator doors and cab

You can enter the desired simulation time interval for the recording.

Playback is possible at any speed supported by Vissim and your hardware.

Unlike simulation mode, you can play animations both forward and backward for the selected time sequences and network sections. This allows you to quickly reach the desired network status in an animation.

Animations record attribute values of network objects that are required for animation display:

- Vehicles: position, turn signal, color, 3D model, state. The animation recording accounts for the graphic parameter setting **Color by vehicle state** under **Vehicles In Network**.
- Pedestrians: position,3D status, color, 3D model
- Signal head: state

You can edit these attribute values using script files. Attribute values of other network objects are not recorded and cannot be edited via script files.

Notes:

- If the script files change attributes in the simulation while an animation is recorded, the changed values are saved to the animations.
- When you run animations, the changes become effective through the script files.
- When you run animations, the script files are not called again.
- Animation files require a lot of memory. Make sure that depending on the recording planned, sufficient memory space is available.

15.2.1 Defining an animation recording

1. From the **Presentation** menu, choose > **Animation Recordings**.

   *The **Animation Recordings** list opens.*

   By default, you can edit the list (see "Using lists" on page 93).

Note: In lists, you can use the **Attribute selection** icon to show and hide attribute values (see "Selecting attributes and subattributes for columns of a list" on page 112).

---

2. In the list, on the toolbar, click the **Add** button .

   A new row with default data is inserted.

3. Make the desired changes:

Element	Description
RecAnim	**Record animation**: If this option is selected, the animation is recorded when you select **Record Animation** in the **Presentation** menu.
Sections	Number of sections to be recorded. No number: The entire network is recorded.
Filename	Name of the **.ani* or **.ani.txt* file to which the simulation is saved as an animation. Depending on the selection available in the **Type** attribute, make sure that after the filename, the extension of your choice is displayed, **.ani* or **.ani.txt*.

Element	Description
End	Simulation second when recording ends If you define multiple animation recordings, avoid creating overlaps between the time intervals.
No	Unique number
Name	Animation name
Start	Simulation second when recording starts
Type	- **For Export (*.ani.txt)**: Saves the animation to a text file. Simulation time, various vehicle data and/or pedestrian data, vehicle position and ped-estrian position as well as the statuses of signal heads and detectors are saved to the text file in rows The text file can be imported into Autodesk 3DS MAX. For further information, refer to the *API\3dsMaxExport\HowTo* folder in your Vissim installation directory. - **For Replay (*.ani)**: Saves the animation to a ANI file that you can replay.

Showing and editing dependent objects as relation

The attribute and attribute values of this network object type are shown in the list on the left, which consists of two coupled lists.

1. In the list on the left, click the desired entry.

The list on the right contains attributes and attribute values of sections allocated to the network object selected in the list on the left (see "Using coupled lists" on page 119).

2. On the list toolbar, in the **Relations** list box, click > **Sections**.
3. Enter the desired data.

The data is allocated.

15.2.2 Recording an animation

1. Ensure that the following requirements are met:
   - At least one animation recording is defined (see "Defining an animation recording" on page 1167)
   - To record parts of the network, you need to define sections that contain these parts (see "Modeling sections" on page 677). The desired sections must be selected in the **Sections** attribute of the animation recording (see "Defining an animation recording" on page 1167).
2. From the **Presentation** menu, choose > **Record Animations**.

   *In the **Presentation** menu, **Record Animations** is marked with* ☑.
3. Start the simulation (see "Running a simulation" on page 840).

*The simulation starts and the data is recorded in the *.ani or *.ani.txt file while the simulation is running. The *.ani or *.ani.txt file is saved at the end of a simulation to the directory and under the name defined in the **Filename** attribute of the animation recording.*

4. To stop the animation recording, from the **Presentation** menu, choose > **Record Animations**.

   *In the **Presentation** menu, **Record Animations** is marked with .*

5. To stop the simulation, click the **Stop** button.

15.2.3 Running the animation

You can run an animation of the simulation with or without an animation file (**.ani*).

When playing back an animation file (**.ani*) in Vissim, you first need to open the corresponding Vissim network.

15.2.3.1 Running the animation with an animation file (**.ani*)

1. From the **Presentation** menu, choose **Animation with ANI file**.
2. From the **Presentation** menu, choose the desired entry:

Toolbar button	Name	Description
	Continuous	When no animation run is started: Opens a window in which you can select the **.ani* file of your choice. Then starts the continuous animation run. When the **Animation single step** mode is started: Let animation run continuously.
	Single Step	When no animation run is started: Opens a window in which you can select the **.ani* file of your choice. Then starts the animation run and shows the step of the first simulation second. When the **Animation continuous** mode is started: Stops animation run and show next single step.
	Stop	Finish animation run
	Single Step Reverse	Stops running animation and shows last step before the current simulation second.
	Continuous Reverse	Stops running animation and plays animation in continuous reverse order.

During the recording of the simulation, attribute values are stored. During playback of the animation, you can show these attribute values in lists. For pedestrians in the network, they include attribute values of the pedestrian routes, such as the static routing decision number, the static route number and the current destination number.

15.2.3.2 Running the animation without an animation file (**.ani*)

Aggregated result attributes are used for visualization, e.g. for the color of link segments or link bars.

1. In the **Presentation** menu, deselect **Animation with ANI file**.
2. Then, from the **Presentation** menu, choose the desired command.

The animation of the simulation is run without an animation file.

15.2.4 Displaying values during an animation run

While running an animation, you can show time-based aggregated values.

1. Before you run the animation, make sure the following requirements are met:
2. Under Evaluation Configuration, specify that you want to collect result attributes for links and/or areas (see "Configuring evaluations of the result attributes for lists" on page 1014).
3. For vehicles on links, click in the network objects toolbar next to **Links** on **Edit graphic parameters**; for pedestrians in areas, click in the network objects toolbar next to **Areas** on **Edit graphic parameters**.
4. Select the option **Use color scheme**.

   *The **Edit color scheme for <network object type>** window opens.*

   If the window does not open because the option has already been deactivated once since program start, click the icon at the end of row to open the window.
5. For vehicles, select the classification of your choice **Lanes** or **Segments**; for pedestrians, select the classification of your choice **Areas** or **Pedestrian Grid Cell Data**.
6. Click the **Attribute** button .
7. Select the desired result attribute.
8. Click the **Filter** button.

   *The **Preselection Filter** window opens.*
9. If desired, under **Simulation Runs**, select the option of your choice.
10. Under **Time Intervals**, select **Show only selected time intervals**.
11. Then select **Last completed**.
12. If under Evaluation Configuration you selected the desired vehicle classes, under **Vehicle Classes** you can now select the desired vehicle class.
13. If under Evaluation Configuration you selected Pedestrian Classes, under **Pedestrian Classes** you can now select the desired pedestrian class.
14. Confirm with **OK**.

15. Click the icon .
16. In the list, click the color scheme of your choice.
17. Confirm with **OK**.
18. Start the animation (see "Running the animation" on page 1169).

The time interval results produced during the selected simulation run are used for the color scheme of the animation run.

16 Using event based script files

You can define scripts in a network file. Using the scripts, you manage script files and define the times at which you want to run the script files during the simulation. This allows you to simulate systems that respond to different simulation states and/or influence the simulation.

16.1 Use cases for event-based script files

- You are modeling a traffic management system that depending on a certain amount of traffic, at the beginning of a time step, uses a script file to change the volumes for partial routes and to allow the use of emergency lanes.
- Opening and closing barriers
- Changing the display state of variable message signs
- Changing the color of vehicles once they have reached their destination parking lot

16.2 Impact on network files

When script files change attributes and you then save the network file, the latter will also contain the changed data.

16.3 Impact on animations

- When script files change attributes during a simulation and animation recording, these changes also become effective when the animation is replayed.
- When you replay animations, the script files are not called again.

16.4 Impact on evaluations

- Script files allow you to access evaluations, e.g. in order to activate evaluations before initialization.
- If the script files change attributes that are used in evaluations, the changed values are saved to the results of the evaluations.

16.5 Defining scripts

1. From the **Scripts** menu, choose > **Event-Based Scripts**.

   *The **Scripts** table is opened.*

2. Make the desired changes:

Short name	Long name	Description
No	**Number**	Unique **number**
Name	**Name**	Name of script

Short name	Long name	Description
RunType	**Type of execution**	**Run type**: Time at which the script is to be called and executed. If you define several scripts to be run at the same time, the scripts are executed in the sequence of their numbering. - **Manually**: The script is not automatically called during the simulation. You call the script at the time of your choice (see "Starting a script file manually" on page 1173). - **Before simulation start** - **After simulation start** - **At time step start** - **At time step end** - **Before simulation end** - **After simulation end**
FromTime	**From time**	Selecting a time step of simulation: Time at which the script is executed for the first time
ToTime	**To time**	Selecting a time step of simulation: Time at which the script is executed for the last time
Period	**Period**	Selecting the length of time intervals (number of time steps) for simulation
Scope	**Scope**	Time period during which the script remains active and is not automatically deactivated: - **Simulation run**: Leave script open during the entire simulation run and in it, call user-defined functions, depending on the **Function name** attribute - **Single call**: Always run the entire script
ScriptFile	**Script file**	Selecting a script file for the script from a directory
FctName	**Function name**	Name of the function or subprogram executed. This option is enabled when for the **Scope** attribute, the time range **Simulation run** is selected.

*If you have selected the run type **Manually**, run the script at the time of your choice (see "Starting a script file manually" on page 1173). If you have defined other run types, the scripts are run automatically.*

16.6 Starting a script file manually

You may run script files manually that have been selected in the attributes of scripts. This also applies to scripts that run script files automatically and have the following attribute values:

- **Before simulation start**
- **After simulation start**
- **At time step start**

- **At time step end**
- **Before simulation end**
- **After simulation end**

Scripts with the **Run type** attribute **Manually** can only be started manually, not automatically.

1. Ensure that the following requirements are met:
   - The Vissim network model is complete, so that a simulation can be started.
   - The simulation parameters have been defined (see "Defining simulation parameters" on page 840).
   - The script file of your choice has been defined and saved to the desired directory.
   - At least one script has been defined for the desired script file (see "Defining scripts" on page 1172).
   - In the script, in the **Script file** attribute, the script file is selected.
2. From the **Scripts** menu, choose > **Event-Based Scripts**.

   *The **Scripts** table is opened.*

   *You can either start the script immediately or at a chose time during the simulation. To run the script file at the time of your choice, perform the simulation in **Simulation single step** mode.*
3. If desired, start the simulation (see "Selecting the number of simulation runs and starting simulation" on page 845).
4. In the **Scripts** table, right-click the entry of your choice.
5. From the shortcut menu, choose **Run script**.
6. If you want to stop the script file, from the **Scripts** menu, choose > **Stop running script**.

17 Runtime messages and troubleshooting

Runtime messages, such as process messages, error messages, and warnings are shown in the **Messages** window. Messages regarding test runs are also displayed in the **Messages** window (see "Showing messages and warnings" on page 1178).

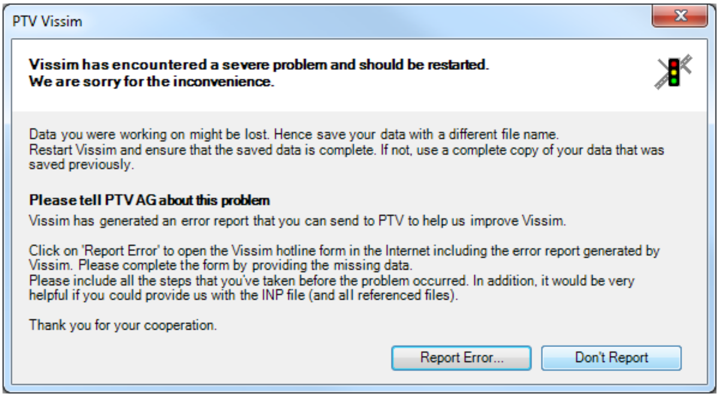
In addition, Vissim opens separate windows for error messages about unexpected program states.

Vissim saves errors to error files (*.err*) (see " Checking the runtime warnings in the file *.err" on page 1176).

You can use the *VDiagGUI.exe* program to perform an error diagnosis (see "Performing an error diagnosis with VDiagGUI.exe" on page 1182).

17.1 Editing error messages for an unexpected program state

An error message opens in case of an unexpected program state:



1. Follow the instructions in the error message.

   This prevents loss of data and consequential errors.

2. Please forward the errors to us, so that we can correct them:
3. Click **Report Error**.

   The PTV Vision Support contact form opens. The necessary data, including the error report, are automatically copied into the form.

   *If you call up the contact form with the menu **Help** > **Technical Support** or the Internet page Technical Support PTV Vissim or Technical Support PTV Viswalk, the error report is not entered automatically.*

4. Save the network file **.inpx* under a different name.
5. Close Vissim.
6. Open Vissim.

*Vissim always copies the network file and saves the copy in the file format *.inp0.*

7. Compare the **.inp0* file with the network file **.inpx* which was saved under a different name.
8. Make sure that the data is complete.
9. If the data in the network file **.inpx* are not complete, copy the file **.inp0*.
10. Name the copy of the **.inp0* file into a **.inpx* file.
11. Continue work with this version.

17.2 Checking the runtime warnings in the file **.err*

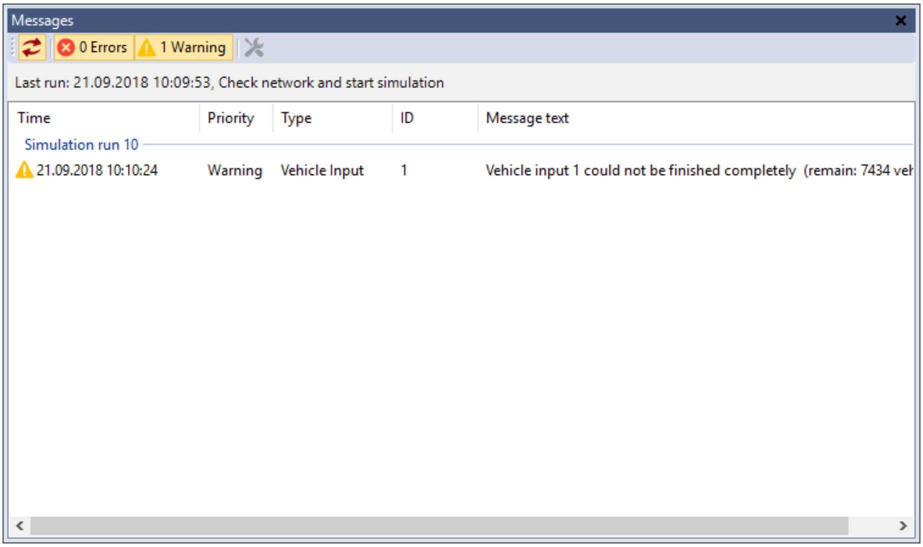
1. Vissim shows errors and messages in the **Messages** window (see "Showing messages and warnings" on page 1178). In addition, by default, Vissim saves errors to the **.err* files and the *vissim_msgs.txt* file (see "Using the vissim_msgs.txt log file." on page 1181). These files are saved in the folder in which the network file **.inpx* is saved.

If in the simulation parameters, only one run is defined, the simulation is started, errors occur and an error file (**.err*) is saved, the name of the error file will not include the simulation run number. If then another simulation run is started, the error file (**.err*) from the previous simulation run will be overwritten.

17.2.1 Runtime warnings during a simulation

If Vissim detects problematic situations during a simulation run which do not prevent the continuation of the simulation, runtime warnings are displayed in the **Messages** window and written to an **.err* file. Except for the file extension **.err*, the name of this file corresponds to the name of the network file.

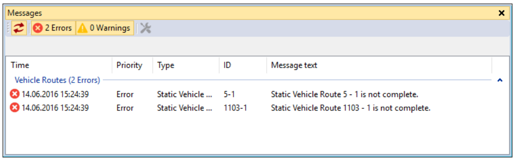
If runtime warnings have occurred, the **Messages** window opens after the simulation run.



- Correct the causes of warnings and errors.

17.2.2 Runtime warnings before a simulation

If the Vissimnetwork contains errors and you start the simulation, runtime warnings will be displayed before the simulation is started.



All errors are saved to the **.err* file. For example, this may be due to the following errors:

- Private Transport: Too little distance between a routing decision and the first connector of the route.
- Private Transport: A vehicle traverses a routing decision and within the same time step exits the link on which the routing decision is located.

  Public Transport: Too little distance between an entry link and the first connector of the route. Because of this, the Private Transport vehicle or the Public Transport vehicle have left their route. Either the vehicle did not stop in time in order to wait for an opportunity for a lane change, or within the same time step, the vehicle has traversed the routing decision and another connector.
- Too little distance between the desired speed decision and the first connector. This message is displayed if, within a time step, a vehicle with the highest speed which is possible within a defined desired speed distribution could traverse both the desired speed decision as well as the start of a connector. For example, this would be possible at 205 km/h and 10 time steps per simulation second, at a maximum distance of 5.70 m. If this actually occurs during the simulation run, the relevant vehicle ignores this desired speed decision.
- An input volume which has not been completely processed, for example due to insufficient capacity of the entry link, because of which there is congestion outside of the network at the end of a defined time interval.
- A vehicle, which has been removed from the network after the maximum lane change waiting time has elapsed (default value 60 sec)
- For some signal controls: Infringements of the minimum green period and interim time infringements during the simulation (see "Detecting inconsistent planning" on page 667).
- For pedestrian simulation: too high density within area per simulation second

17.2.3 Runtime warnings during multiple simulation runs

If the error file (**.err*) is written during multiple simulation runs, Vissim enters the number of the respective simulation run into the error file (**.err*). The number of simulation runs is defined in the simulation parameters (see "Defining simulation parameters" on page 840).

17.3 Showing messages and warnings

The **Messages** window displays process messages, error messages and warnings, e.g. after you have chosen the **Check network** command (see "Checking the network" on page 857), but during scenario management, simulation and reading and writing of model transfer files.

Related messages are combined in a group. The name of each group and the number of errors are listed in blue font above the group.

In Scenario management the name of a group can include different information, for example the name and number of the scenario, the name of the calculation performed and the number of the simulation run. The information included in the group name is separated by colons. Example: `Scenario 1: Calculate Scenario 1': Load scenario: Read in network`

Vissim opens different significant messages in a separate window that allow you to open the **Messages** window and view all messages. For example, Vissim opens the **Check network** window, showing errors, after you have selected the **Check network** command. Click the **Show** button to open the **Messages** window.

Before a pedestrian simulation is started, Vissim also automatically checks the network for consistency and displays error messages and warnings in the **Messages** window.

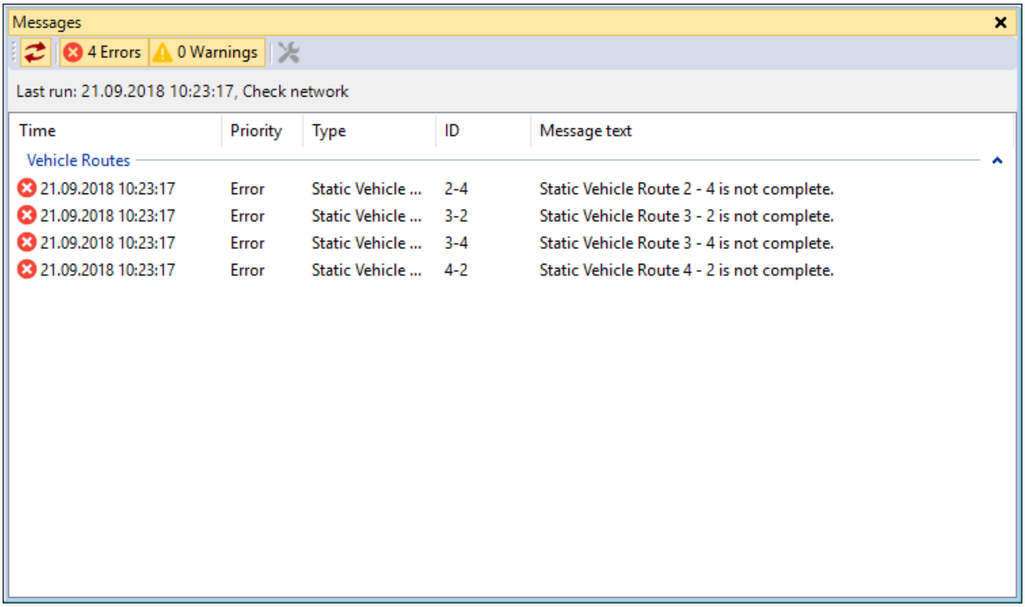
The **Messages** window is not updated and does not display messages when the Quick mode is activated and the simulation is running. This allows for a maximum simulation speed.

The **Messages** window can be freely positioned or docked (see "Arranging or freely positioning program elements in PTV Vissim" on page 91). The **Messages** window cannot be anchored in a section together with a list or a Network editor, but only across the entire width or height of the user interface.

17.3.1 Opening the Messages window

- In the **View** menu, choose > **Messages**.

  *The **Messages** window opens.*



Element	Description
Time	Time at which the message was generated
Priority	Debug messages, error messages or warnings
Type	Network object type, e.g. **Nodes**
ID	Number of the network objects
Message text	Description of error message or warning
Syn-chronization	Zoom on network object: Network objects of the rows selected are selected in the Network editor. This may also be a number of network objects of the same network object type.
	Network objects of the rows selected are not selected in the Network editor.
<Number> Errors	Show all errors found

Element	Description
<Number> Warnings	Show all warnings found
Fix	Only for errors of the category **Vehicle routes** and 2D/3D model segments: Start interactive repairs. If, e.g., a vehicle route is interrupted because a connector was deleted, you can identify this error in the **Messages** window, define the missing connector in the Network editor and then click the **Fix** button . Vissim will complete the vehicle route. Then select **Check network**. Error messages are no longer displayed for the repaired objects.

17.3.2 Editing messages

Sorting messages

- To sort a column, click the column header.

  An arrow in the column header indicates whether the entries are sorted in ascending or descending order.

Expanding or collapsing groups

To close a group, double-click the row with the group name highlighted in blue.

Editing individual messages or groups

- To edit an individual entry, right-click the row of your choice.
- To edit a group, right-click the row with the group name highlighted in blue.

  If ***Synchronization*** *is selected, objects or groups selected in the* ***Messages*** *window are also selected in the Network editor.*

  The context menu opens.

1. Choose the desired entry from the context menu.

Element	Description
Delete message	Delete individual rows
Fix	The **Fix** command is described further above.

Element	Description
Fix group	Repair all errors or messages of the current category. The program continues to display the errors. Then select **Check network**. Error messages are no longer displayed for the repaired objects.
Delete all messages in this group	Delete all errors or messages of the current category.
Discard sorting	Only active if at least one column is sorted: Resets column sorting to default sorting.

Toast notification shows number of messages

When the **Messages** window is open, a simulation ends or is terminated and Vissim shows messages in the **Messages** window, a toast message opens at the bottom right border of the screen, showing the number of messages displayed.

17.4 Using the *vissim_msgs.txt* log file.

The *vissim_msgs.txt* log file is automatically saved to:

C:\Users\<username>\AppData\Local\Temp\VISSIM\vissim_msgs.txt

- From the **File** menu, choose > **Show Log File**.

17.4.0.1 Naming the log files of multiple instances

Each time an instance of Vissim is opened, a log file is saved. These log files are named consecutively according to the following convention: *vissim_msgs_0.txt*, *vissim_msgs_1.txt*, etc.

17.4.0.2 Structure of the log file

The log file consists of an upper and lower section. Upper section: System entries which were generated at the start of Vissim. Example:

```
5/11/2018  9:43:18 System set VISSIM localization part I...
5/11/2018  9:43:18 Initialize system licensing...
5/11/2018  9:43:20 System certified date: 2017-05-11 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System certified time period: 25 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System system date: 2017-05-11 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System certified date: 2017-05-11 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System certified time period: 25 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System system date: 2017-05-11 (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box: x-xxxxxxx)
5/11/2018  9:43:20 System set VISSIM localization part II...
```

```
5/11/2018  9:43:21 Start system licensing 2017-05-11 09:43:19.
5/11/2018  9:43:21 System certified time successfully updated (server: ptv-dxx-
xxxxx.ptvag.ptv.de, Box-Serial Number: x-xxxxxxx)
5/11/2018  9:43:21 System licensing completed 2017-05-11 9:43:20.
5/11/2018  9:43:21 System initialization of core components completed...
2018-05-11  9:43:21 System version 10.00 - 00* [65780] , filename C:\Program
Files\PTV Vision\PTV Vissim 10\Exe\VISSIM100.exe
5/11/2018  9:43:21 System initializing Vissim...
5/11/2018  9:43:21 System starting Vissim...
5/11/2018 9:43:21 System read settings...
5/11/2018  9:43:21 System initialization completed.
...
```

Lower section: Detailed messages about the functions of Vissim, e.g. warnings or messages, which were shown during the ANM import and the creation of the Vissim network:

```
...
5/11/2018  9:56:57 System [ANM] import started (initial).
5/11/2018  9:56:57 System [ANM] import network data.
5/11/2018  9:56:57 Warning ANM link 4A: SPEED: (from 20 to 40) This attribute is 0
and is set to the default value.
5/11/2018  9:56:57 Warning ANM link 4B: SPEED: (from 40 to 20) This attribute is 0
and is set to the default value.
5/11/2018  9:56:57 System This ANM file does not provide a projection definition.
Coordinates are assumed to be cartesian values.
5/11/2018  9:56:57 Warning ANM node 10: Multi-leg node has additionally ANM zones
attached. Dummy link stubs are generated ignoring node geometry.
5/11/2018  9:56:57 Warning ANM node 40: Multi-leg node has additionally ANM zones
attached. Dummy link stubs are generated ignoring node geometry.
5/11/2018  9:56:57 Warning ANM node 20: This node contains lanes that do not have any
movements defined to or from.
5/11/2018  9:56:57 Warning ANM node 40: This node contains lanes that do not have any
movements defined to or from.
5/11/2018  9:56:57 Copy system [ANM] file C:\Test\PTV Vissim\ANM_Import\ANM_Import_
Test.anm to C:\Test\PTV Vissim\ANM_Import\ANM_Import_net3.panm.
5/11/2018  9:56:57 System [ANM] Remove obsolete objects.
5/11/2018  9:56:57 System [ANM] import end.
...
```

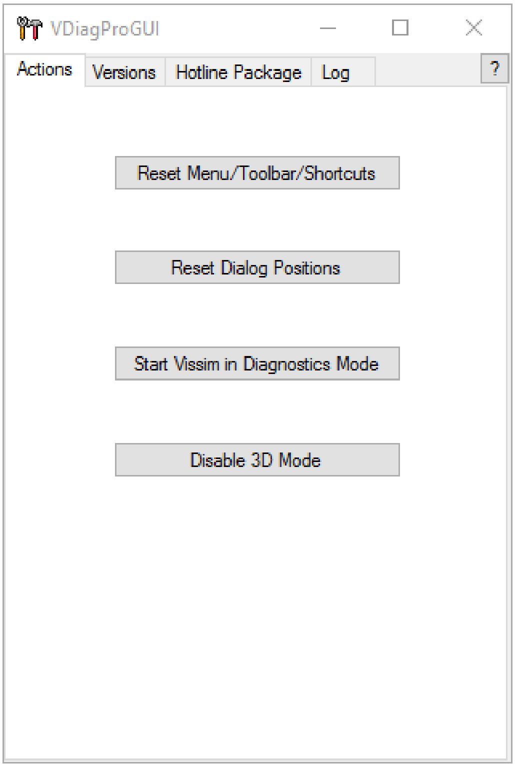
17.5 Performing an error diagnosis with VDiagGUI.exe

In certain cases PTV Vision Support can ask you to run the file *VDiagGUI.exe*.

1. Start Windows Explorer.
2. Open the Vissim installation directory.
3. Open the *Exe* directory.
4. Double-click the **VDiagGUI.exe** file.

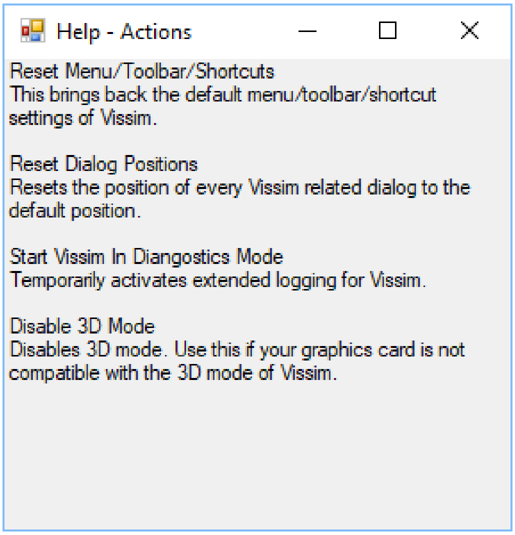
   *The **VDiagProGUI** window opens. The **Actions** tab is shown by default.*
5. Follow the instructions from PTV Vision Support.
6. Open the desired tab.

Actions tab



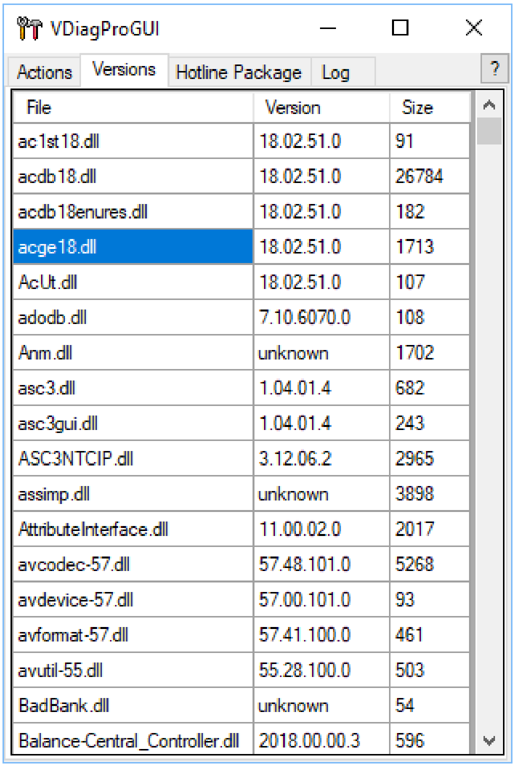
Button	Description
Reset Menu/Toolbar/Shortcuts	Reset toolbars and menus Note: You can also reset toolbars and menus in Vissim (see "Resetting menus, toolbars, shortcuts, and dialog positions" on page 155).
Reset Dialog Positions	Reset window positions Note: You can also reset window positions in Vissim (see "Restoring the display of windows" on page 93).
Start Vissim in Diagnostics Mode	Only click the button if instructed to do so by PTV Vision Support and follow the instructions from PTV Vision Support.
Disable 3D Mode	Disable 3D mode. This may be necessary if Vissim cannot be started in 3D mode. Further information can be found in the installation instructions *Vissim <Version> - Installation Manual.pdf* in the *Doc* directory of your Vissim installation.

- If you would like to see information on the tab, please click the **?** button.

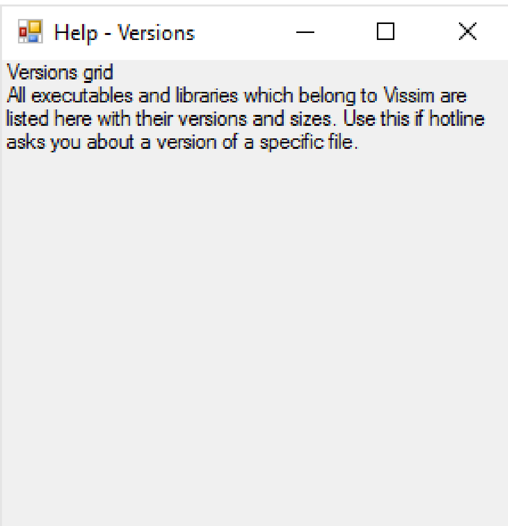


Versions tab

PTV Vision Support may ask you to check in this tab whether certain files are present.

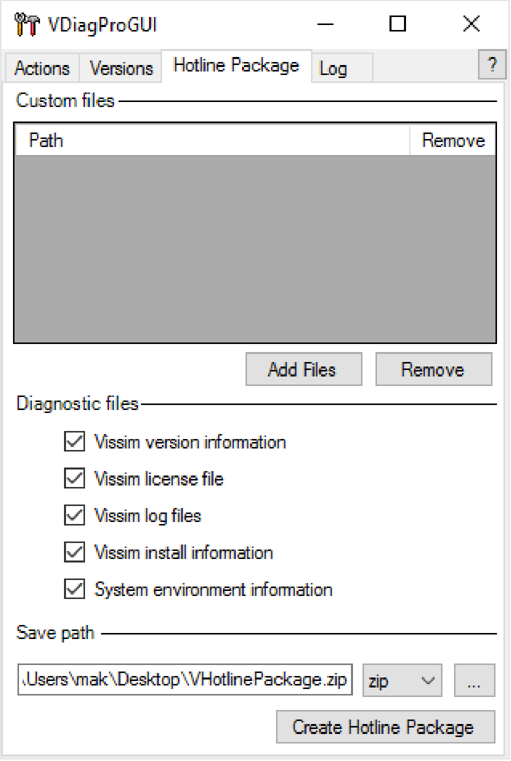


- If you would like to see information on the tab, please click the **?** button.



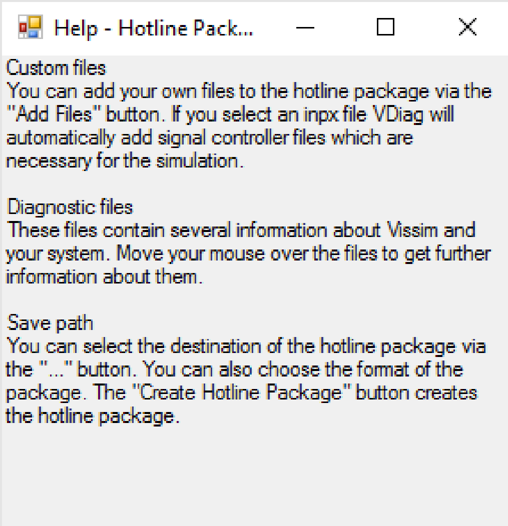
Hotline Package tab

In the **Hotline Package** tab you can combine all desired data for PTV Vision Support, pack into a file and send to PTV Vision Support.



Section	Description
Custom files	➤ **Add files**: Add files to the package ➤ **Remove**: Remove selected files
Diagnostic files	Check the desired options. For more information about the data provided by the respective options, move the mouse pointer over the option text.
Save path	Select the path and file name. In the list box, click the desired file format.
Create Hot-line Pack-age	Create the file for PTV Vision Support.

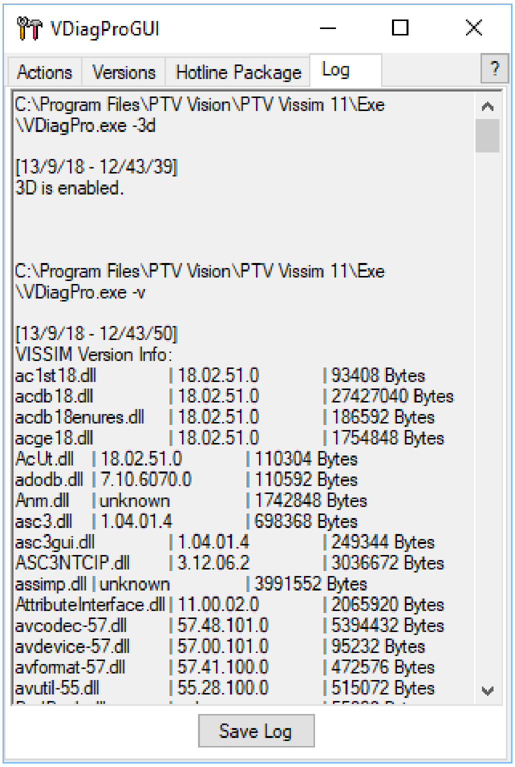
▸ If you would like to see information on the tab, please click the **?** button.



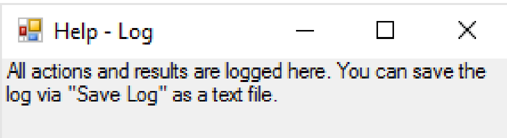
Log tab

PTV Vision Support may ask you to create a log file in this tab.

- Click the **Save Log** button.



- If you would like to see information on the tab, please click the **?** button.



17.6 Saving network file after losing connection to dongle

If there are problems with the connection between Vissim and the dongle, a message is displayed.

- Click **Cancel** to open another message and save the ***.inpx** network file, if you made changes to the Vissim network since you last saved it.
- If you then click **OK**, Vissim will wait until the connection to the dongle has been re-established.

18 Add-on modules programming interfaces (API)

Vissim provides various APIs as an add-on module. With this, you can integrate your own applications into Vissim. Not all add-on modules are by default part of your Vissim license.

Add-on modules and their documentation are saved in the folder *..\Vissim<Version>\API*.

18.1 Using the COM Interface

Note: The following Vissim versions do not have a COM interface:

- Demo versions of Vissim
- Vissim Viewer

You can use the add-on COM interface module (Component Object Model) for:

- Preparation and postprocessing of data
- Efficiently controlling the sequence for the examination of scenarios
- Including control algorithms which you have defined
- Access to all network object attributes

Via the COM interface you can start Vissim from other applications or scripts or you can access Vissim objects.

You can use various programming environments, e.g. VisualBasic for Applications in Microsoft Excel™, Visual C++ or Visual J++ and use simple script languages, for example, VB Script or Python.

Script files from the COM interface can be executed in Vissim (see "Selecting and executing a script file" on page 1190).

COM Help

Information about the functions of the COM interface and associated commands can be found in the COM Help.

- From the **Help** menu, choose > **COM Help**.

You can find examples in the following default directory:

- *..\Users\Public\Public documents\PTV Vision\PTV Vissim <Version number>\Examples Training\COM*

18.1.1 Accessing attributes via the COM interface

Using the **AttValue** method, you have read access to all network object attributes. Write access is not available for all attributes. In the COM Help, the **Editable** row of each attribute specifies a **True** or **False** value, indicating whether or not the attribute can be write-accessed.

18.1.2 Selecting and executing a script file

In Vissim you can select and execute external script files. The following scripting languages are supported:

- Visual Basic **.vbs*
- Javascript **.js*
- Python **.py*, **.pys* (see "Using Python as the script language" on page 1191)

Tip: You may also manage scripts, which are executed during a simulation at a chosen time, to the network file (see "Using event based script files" on page 1172).

The global variable Vissim can be used without initialization in the script and always refers to the running Vissim, from which the script has been started. Additionally, you can use the variable **CurrentScript** to create a reference to the current script, e.g. in order to access user-defined attributes there.

You need not create a Vissim object, using *CreateObject*, as Vissim has already been started. If additional instances of Vissim are required in the script, you can use *CreateObject*.

Scripts may include any command of the scripting language, e.g. loop, branch, entry, and output commands.

Scripts must not contain any global declarations.

18.1.2.1 Executing script files

1. From the **Scripts** menu, choose > **Run Script File**.

   A window opens.

2. Select the folder to which the script file has been saved.
3. Select the desired script file.
4. Click the **Open** button.

The script file is executed in Vissim.

18.1.2.2 Examples of script files

The following directory contains a Vissim network that uses script files: *..\Examples Training\COM\Drop-off Zone*. This example is available for Visual Basic Script (VBS) and Python.

The following scripts are used to halve the volume of the first time interval for each input.

Example 1 of a **.vbs* script file

```
For each flow in all_flows
For i = 1 To all_flows.Count
flow.AttValue("Volume(1)") = 0.5 * flow.AttValue("Volume(1)")
Next
```

Example 2 of a **.vbs* script file

```
all_flows = Vissim.Net.VehicleInputs.GetAll
For i = 0 To UBound (all_flows)
all_flows(i).AttValue("Volume(1)") = 0.5 * all_flows(i).AttValue("Volume(1)")
Next
```

18.1.3 Using Python as the script language

A distinguishing feature of Python is its clear and understandable syntax. With this you can conveniently set up a prompt which is tailored to your project.

Python and all of the additional libraries are open source programs without usage restrictions.

Notes:

- You must install **Python 2.7** or a higher version and the appropriate extension **PythonWin**, which provides Python with the COM functionality.
- **Python 2.7** and the appropriate extension **PythonWin** must correspond to your Vissim version, e.g. **python-2.7.6.amd64.msi** and **pywin32-218.win-amd64-py2.7.exe** for the 64-bit edition of Vissim.
- wxPython is not supported.

Example of a **.pys* script file

```
all_flows = Vissim.Net.VehicleInputs.GetAll()
for i in range(len(all_flows)):
all_flows[i].SetAttValue("Volume(1)", 0.5 * all_flows[i].AttValue("Volume(1)"))
```

18.2 Activating the external SC control procedures

You can use external, user-defined signal control procedures in Vissim (see "Add-on module external signal control SC" on page 615). For this, you must provide an external program DLL for the control and a specific dialog DLL. The DLL files must be programmed in C or C++. The files are integrated with the interfaces *SignalControl.DLL* and *SignalGUI.DLL*.

18.3 Activating the external driver model with DriverModel.dll

You can replace the driver model with an external, user-defined driver model using Vissim. The external driver model can be used for all or part of the vehicles in a simulation.

During the simulation, the DLL file is called up in each time step for each vehicle for which an external driver model is available. The DLL file then controls the behavior of the vehicle. Vissim transfers data to the DLL file:

- the current state of the vehicle
- the current state of the vehicle environment, e.g. the number and state of priority rule (1 = blocked, 3 = free), if closer to current vehicle than the next signal head downstream.
- World coordinate of the front edge and rear edge of neighboring vehicles

- Spline of the middle line of the lane the Ego vehicle is using on its route or path within the line of sight
- Values of user-defined vehicle attributes can be transferred to the DLL and changed by it.

The acceleration, deceleration and lane changing behavior are calculated in the DLL file. The result is returned to Vissim.

You activate the external driver model for the desired vehicle type (see "Using vehicle types" on page 267).

External driver model DLL-files can be used in simulation runs with multiple cores. For this purpose, each driver model DLL-file must confirm that it supports multithreading. This does not apply in the following cases, in which the DLL file must be programmed accordingly:

- There is only one externally controlled vehicle.
- All externally controlled vehicles are on the same link.

The external driver model must be implemented as a DLL file in the programming language C or C++. The structure is described in the documentation.

Documentation in English

- *..\Program Files\PTV Vision\PTV Vissim <Version number>\API\DriverModel_DLL\Interface_Description.pdf*

Example file

- *..\Program Files\PTV Vision\PTV Vissim <Version number>\API\DriverModel_DLL\DriverModel.cpp*

18.4 Accessing EmissionModel.dll for the calculation of emissions

For the calculation of emissions, you can access the file *EmissionModel.dll*. You must provide the *EmissionModel.dll*. It is not part of Vissim.

You activate the external emission model for the desired vehicle type (see "Activating emission calculation and emission model for a vehicle type" on page 274). In each time step Vissim calls up the functions in the file *EmissionModel.dll* for the calculation of the emission values. The emission values can be output into the vehicle log and the link evaluations.

If no *EmissionModel.dll* is available, you can use the EnViVer Pro add-on module or EnViVer Enterprise to calculate the emissions for individual vehicle types.

Documentation in English

- *..\Program Files\PTV Vision\PTV Vissim <Version number>\API\EmissionModel_DLL\EmissionModel.txt*

Files

- *..\Program Files\PTV Vision\PTV Vissim <Version number>\API\EmissionModel_DLL*

18.5 Activating the external pedestrian model with PedestrianModel.dll

Instead of the default file *PedestrianModel.dll* you can also use external pedestrian DLL files (see "Requirements for pedestrian simulation" on page 866).

With an external pedestrian DLL file, you can optionally also generate pedestrian inputs.

Please contact us if you wish to create your own pedestrian DLL files.

Directory of *PedestrianModel.dll* file

- *..\Program Files\PTV Vision\PTV Vissim <Version number>\\Exe\PedestrianModel.dll*

19 Overview of PTV Vissim files

The following tables describe the files which are used in Vissim.

19.1 Files with results of traffic flow simulation

Extension	Name	Description
***.att**	Attribute file	Output file with static attributes and/or result attributes (see "List toolbar" on page 97), (see "Performing evaluations" on page 1001).
***.err**	Runtime warnings	Error messages and warnings from the last simulation run. If several simulation runs have been performed, multiple files whose names each contain a simulation run number (see " Checking the runtime warnings in the file *.err" on page 1176).
***.fhz**	Vehicles	List of all vehicles with information on when and where they were used at what speed in the network (see "Saving vehicle input data to a file" on page 1110)
***.fzp**	Vehicle record	Evaluation of vehicle data (see "Saving vehicle record to a file or database" on page 1031)
***.knr**	Node evaluation (raw data)	Output of raw data of node evaluation (see "Evaluating nodes" on page 1057)
***.ldp**	SC detector record	Evaluation file for a VS-PLUS SC (see "Evaluating SC detector records" on page 1070)
***.lsa**	Signal changes	Chronologically sorted file of the signal state changes of all SCs (see "Evaluating signal changes" on page 1081)
***.lzv**	SC time distributions	Release durations and closure periods of signal groups of all SCs (see "Saving SC green time distribution to a file" on page 1078)
***.mdb**	Database out-put	Database outputs (see "Saving evaluations in databases" on page 1021)
***.mer**	Data collection (raw data)	Raw data of data collections (see "Evaluating data collection measurements" on page 1093)
***.merP**	Area measurements (raw data)	Raw data of area measurements of pedestrian simulation (see "Evaluating pedestrian areas with area measurements" on page 1041)
***.mle**	Managed lanes evaluation	Vehicles on managed lane route and general purpose route, with aggregated data (see "Saving managed lane data to a file" on page 1084)
***.ovw**	Public transport waiting times	Waiting times of PT vehicles not due to own boarding and alighting, for example, waiting on SC or behind other PT vehicles (see "Saving PT waiting time data to a file" on page 1092)

Extension	Name	Description
***.pp**	Pedestrian record	Result attributes of pedestrian movements (see "Saving pedestrian record to a file or database" on page 1053)
***.rsr**	Travel times (raw data)	Record of completed travel time measurements in chronological order (see "Evaluating vehicle travel time measurements" on page 1096)
***.rsrP**	Pedestrian travel time (raw data)	Record of completed travel time measurements in chronological order from the pedestrian simulation (see "Evaluating pedestrian travel time measurements" on page 1046)
***.spw**	Lane changes	Record lane change location and time (see "Saving lane change data to a file" on page 1028)
***.trc**	Trace	Trace outputs which were created by VAP logic (see "Add-on module Traffic-dependent VAP Programming" on page 626)

19.2 Files for test mode

Extension	Name	Description
***.m_i**	Macro input	Manually set or edited detector types. Input file for macro test runs (see "Using macros for test runs" on page 1155).
***.m_o**	Macro output	Manually set detector types. The file is saved when the test run is selected and subsequently renamed in *m_i*.

19.3 Files of dynamic assignment

Extension	Name	Description
***.bew**	Cost file	Up-to-date list of costs for current paths, edge times and volumes of two previous simulation runs (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771). The first list `Edges` does not contain edges that traverse managed lane routes and include aggregated values. The second list `Edges` only contain edges that traverse managed lane routes and include aggregated values. Refer to the **Example of a cost file *.bew** below this table.
***.cva**	Evaluation of convergence	Volumes and travel times for the current and prior dynamic assignment (see "Saving data about the convergence of the dynamic assignment to a file" on page 1067)
***.weg**	Path file	List of edges and paths currently found in network that were part of the last simulation run (see "Attributes for the trip chain file, matrices, path file and cost file" on page 771), (see "Path search and path selection" on page 738): Refer to **Structure of the path file *.weg** and **Example of a path file *.weg** below this table.
***.fkt**	Trip chain file	Traffic demand for dynamic assignment in a trip chain file (see "Modeling traffic demand with trip chain files" on page 730)

19.3.1 Example of a cost file **.bew*

```
$VISION
* File: C:\Users\Public\Documents\PTV Vision\PTV Vissim 11\
Examples Training\Dynamic Assignment\3 Paths\3 Paths.inpx
* Comment:
* PTV Vissim:   11.00 [01*]
*
* Table: Dynamic assignment
*
$DYNAMICASSIGNMENT:EVALINT
600
*
* Table: Edges
*
$EDGE:NO;FROMNODE;TONODE;LINKSEQ
1;1;2;1
2;2;3;1
3;2;3;10000,2,10003
4;2;3;10001,3,10002
5;3;4;1
6;2;2;1,10001
7;2;2;1
8;2;2;1,10000
9;3;3;10002,1
10;3;3;1
11;3;3;10003,1
*
* Table: Edges
*
$EDGE:NO;TRAVTMNEW(1,ALL);TRAVTMOLD(1,ALL);VOLNEW(1,ALL);TRAVTMNEW(2,ALL);
TRAVTMOLD(2,ALL);VOLNEW(2,ALL);TRAVTMNEW(3,ALL);TRAVTMOLD(3,ALL);VOLNEW(3,ALL)
1;6.363;6.125;76;6.447;6.226;82;6.428;6.199;84
2;17.846;17.877;29;17.908;17.975;36;17.926;17.974;45
3;19.366;19.370;46;19.382;19.391;46;19.359;19.370;38
4;20.177;20.177;0;20.204;20.204;0;20.114;20.114;0
5;6.311;5.375;73;6.304;5.386;82;6.287;5.409;84
6;1.005;1.005;0;1.015;1.015;0;1.008;1.008;0
7;1.087;1.074;29;1.079;1.079;37;1.054;1.062;45
8;1.143;1.157;47;1.156;1.167;45;1.139;1.155;39
9;0.978;0.978;0;1.000;1.000;0;0.962;0.962;0
10;1.083;1.078;29;1.079;1.092;36;1.102;1.088;45
11;1.245;1.246;46;1.299;1.281;46;1.239;1.233;38
*
* Table: Edges
*
$EDGE:NO;TRAVTMNEW(1,ALL);TRAVTMOLD(1,ALL);VOLNEW(1,ALL);TRAVTMNEW(2,ALL);
TRAVTMOLD(2,ALL);VOLNEW(2,ALL);TRAVTMNEW(3,ALL);TRAVTMOLD(3,ALL);VOLNEW(3,ALL)
2;18.042;17.873;75;18.057;17.922;82;18.011;17.899;83
7;1.099;1.097;76;1.095;1.094;82;1.050;1.050;84
10;1.113;1.094;75;1.124;1.100;82;1.119;1.098;83
```

19.3.2 Structure of path file *.weg

- `Table: Dynamic Assignment`:

```
<Evaluation interval:>;<Number of current simulation run>;<Number of converged sim-
ulation runs>
```

- `Table: Edges`:

  One row for each edge. Each row contains the numbers of the Vissim objects:

```
<Edge number>;<FromNode number>;<ToNode number>; comma-separated:
&lt;Segment number 1&gt; to &lt;Segment number n&gt;
```

- `Table: Paths`: One row for each path from parking lot to parking lot. The row contains numbers of the objects and attribute values in Vissim:

```
<Number of path>;<Number of From parking lot>;<Number of To parking lot>;comma-sep-
arated: <Number of path 1> to <Number of path n>;
<Volume (new)> in time interval 1> to <Volume (new) in time interval n>;
<Last travel time in time interval 1> to <Last travel time in time interval n>;
<Demand target (relative) in time interval 1> to <Demand target (relative)> in time
interval n>;
```

The other `Paths` tables contain object numbers and attribute values, each in a separate row, for the following objects:

- Paths from a dynamic routing decision to a parking lot
- Representative data for paths from a parking lot to a parking lot
- Representative data for paths from a dynamic routing decision to a parking lot

19.3.3 Example of a path file *.weg

```
$VISION
* File: ..\Examples Training\Dynamic Assignment\3 Paths\3 Paths.inpx
* Comment:
* PTV Vissim: 11.00 [01*]
*
* Table: Dynamic assignment
*
$DYNAMICASSIGNMENT:EVALINT;CURITERIDX;NUMCONVSIMRUNS
600;1;0
*
* Table: Edges
*
$EDGE:NO;FROMNODE;TONODE;LINKSEQ
1;1;2;1
2;2;3;1
3;2;3;10000,2,10003
4;2;3;10001,3,10002
5;3;4;1
```

```
6;2;2;1,10001
7;2;2;1
8;2;2;1,10000
9;3;3;10002,1
10;3;3;1
11;3;3;10003,1
*
* Table: Paths
*
$PATH:NO;FROMPARKLOT;TOPARKLOT;EDGESEQ;VOLNEW(1,ALL);VOLNEW(2,ALL);VOLNEW(3,ALL);
VOLNEW(4,ALL);VOLNEW(5,ALL);VOLNEW(6,ALL);PATHTRAVTMNEW(1,ALL);PATHTRAVTMNEW(2,ALL);
PATHTRAVTMNEW(3,ALL);PATHTRAVTMNEW(4,ALL);PATHTRAVTMNEW(5,ALL);PATHTRAVTMNEW(6,ALL);
DEMTARGREL(1,ALL);DEMTARGREL(2,ALL);DEMTARGREL(3,ALL);DEMTARGREL(4,ALL);
DEMTARGREL(5,ALL);DEMTARGREL(6,ALL)
1;1;2;1,7,2,10,5;33;33;31;33;28;35;20.8;20.9;20.9;21.0;21.0;20.8;;;;;;
2;1;2;1,6,4,9,5;25;27;16;20;27;29;22.8;22.9;23.1;23.0;23.0;22.9;;;;;;
3;1;2;1,8,3,11,5;26;32;24;20;35;26;22.4;22.7;22.5;22.8;22.6;22.7;;;;;;
*
* Table: Paths
*
$PATH:NO;FROMDYNVEHROUTDEC;TOPARKLOT;EDGESEQ;VOLNEW(1,ALL);VOLNEW(2,ALL);
VOLNEW(3,ALL);PATHTRAVTMNEW(1,ALL);PATHTRAVTMNEW(2,ALL);PATHTRAVTMNEW(3,ALL);
DEMTARGREL(1,ALL);DEMTARGREL(2,ALL);DEMTARGREL(3,ALL)
*
* Table: Paths
*
$PATH:NO;FROMPARKLOT;TOPARKLOT;EDGESEQ;VOLNEW(1,ALL);VOLNEW(2,ALL);VOLNEW(3,ALL);
PATHTRAVTMNEW(1,ALL);PATHTRAVTMNEW(2,ALL);PATHTRAVTMNEW(3,ALL);DEMTARGREL(1,ALL);
DEMTARGREL(2,ALL);DEMTARGREL(3,ALL)
1;1;2;1,7,2,10,5;77;82;85;37.6;37.6;37.4;;;
*
* Table: Paths
*
$PATH:NO;FROMDYNVEHROUTDEC;TOPARKLOT;EDGESEQ;VOLNEW(1,ALL);VOLNEW(2,ALL);
VOLNEW(3,ALL);PATHTRAVTMNEW(1,ALL);PATHTRAVTMNEW(2,ALL);PATHTRAVTMNEW(3,ALL);
DEMTARGREL(1,ALL);DEMTARGREL(2,ALL);DEMTARGREL(3,ALL)
```

19.4 Files of the ANM import

Extension	Name	Description
***.anm**	Abstract network model	Input file in **.xml* format. Network export from Visum (see "Importing ANM data" on page 366)
***.anmroutes**	ANM route file	Input file in **.xml* format. Export of paths and path volumes from Visum (see "Importing ANM data" on page 366)
***.panm**		Backup copy of an **.anm* imported file (see "Importing ANM data" on page 366)
***.panmroutes**		Backup copy of an **.anmroutes* imported file (see "Importing ANM data" on page 366)

Extension	Name	Description
***.inpx**	Input: Network file (input file)	The output of ANM import may be a Vissim network file with the extension **.inpx*.
***.weg**	Path file	Output file of ANM import for dynamic assignment (see "Files of dynamic assignment" on page 1195)

19.5 Other files

Extension	Name	Description
***.layx**	Layout settings file	Initializes the Vissim session with position and size of the screen windows and settings of the display options (see "Saving and importing a layout of the user interface" on page 146)
***.inpx**	Input: Network file (input file)	Description of the entire Vissim traffic network with all elements which belong to it
***.inp**	Input: Network file (input file)	File format of network file until Vissim 5.40. **.inp* network files saved in Vissim 5.40 can also be opened in Vissim from version 6 and above. Network files from older versions cannot be opened in Vissim version 6 or above.
***.inp0**	Network backup copy	Automatically generated copy of the input file
***.pua**	Interstages (ASCII)	Output file of Vissig/CROSSIG/P2, input file for a VAP control logic with stages and interstages (see "Exporting data from the SC Editor" on page 669), (see "Add-on module Traffic-dependent VAP Programming" on page 626)
***.pw1**	Parameter file	VS-PLUS parameter settings (see "Add-on module VS-Plus" on page 628)
***rcf**	Route coordinates file	Input file in **.xml* format. Visum file for Vissim route export (coordinates) (see "Data stored in the *.rcf file" on page 381). The file contains the following route data: - Numbers of the zones in which routes begin and end - Coordinates of the link polygons - Volume - Vehicle types - Projection information
.*.sig	Fixed time controller file	Supply file of the fixed time controllers in XML format (see "Opening and using the SC Editor" on page 631), (see "Using SC type Fixed time" on page 607)
***.stg**	TRENDS supply	ASCII supply file for the TRENDS control procedure (see "Using add-on module TRENDS" on page 624)

Extension	Name	Description
***.trax**	Model transfer file	File to which the modification changes are saved (see "Editing the project structure" on page 1139)
***.v3d**	Vissim 3D file	3D model file for vehicles, pedestrians or static objects (see " Using 2D/3D models" on page 219), (see "Using static 3D models" on page 674) A **.v3d* file may include movement statuses that are relevant for animation of the model, e.g. for the opening and closing of doors or the pedaling movement of a cyclist.
***.vap**	VAP logic	File with signal control program logic of a VAP controller (see "Add-on module Traffic-dependent VAP Programming" on page 626)
***.vce**	VS-PLUS C format	VS-PLUS parameter settings in C format (see "Add-on module VS-Plus" on page 628)
***.wtt**	Value type table	Internal interface description of data types and their formats transferred by an external control procedure in the Signal Times Table window and/or the SC detector record (see "Defining SC and signal control procedures" on page 602), (see "Using SC type Fixed time" on page 607)
***.vxb**	TRENDS supply	Binary supply file for the TRENDS control procedure (see "Using add-on module TRENDS" on page 624)
***.vissimpdb**	Vissim project file	Binary project file for scenario management (see "Scenario management" on page 1133)

20 References

References for traffic planning and engineering

Böttger, R.: Moderne Steuerungsverfahren mit dezentraler Wirkungsweise. Grünlicht, Ausgabe 27, pages 5-11, Siemens AG, München, 1989 (State-of-the art control procedures)

Fellendorf, M.: VISSIM: Ein Instrument zur Beurteilung verkehrsabhängiger Steuerungen. In: Tagungsband zum Kolloquium „Verkehrsabhängige Steuerung am Knotenpunkt", Forschungsgesellschaft für Strassen- und Verkehrswesen, Köln, 1994 (VISSIM: A tool for assessment of traffic-actuated controls)

Forschungsgesellschaft für Straßen- und Verkehrswesen: Richtlinien für Lichtsignalanlagen - RiLSA, Ausgaben 1992, 2010, Köln (Guidelines for signal control)

Hubschneider, H.: Mikroskopisches Simulationssystem für Individualverkehr und Öffentlichen Personennahverkehr. Schriftenreihe des Instituts für Verkehrswesen, Heft 26, Universität (TH),Karlsruhe, 1983 (Microscopic simulation systems for private and public transportation)

Hoefs, D.H.: Untersuchung des Fahrverhaltens in Fahrzeugkolonnen. Straßenbau und Straßenverkehrstechnik Heft 140, Bonn, 1972 (Analysis of driving behavior in groups of vehicles)

Hurrle, R.: Das ÖPNV-System Freiburg. DER Nahverkehr, Heft 7/8, pages 56-60, 1993

Kaul, H.: VS-PLUS: Ein neuer Weg zur Realisierung verkehrsabhängiger Steuerungen. In: Tagungsband zum Kolloquium „Verkehrsabhängige Steuerung am Knotenpunkt“, Forschungsgesellschaft für Strassen- und Verkehrswesen, Köln, 1994 (A new way of implementing traffic-actuated signal controls)

Kohoutek, P.; Nagel, C.; Fellendorf, M.; Vortisch, P.; Brosthaus, J.: ADVANCE - integrierte Simulation von Verkehrsnachfrage, Verkehrsfluß und Kfz-Emissionen. In: Tagungsbericht zur HEUREKA ´99, Forschungsgesellschaft für Straßen- und Verkehrswesen, Köln, pages 359-368, 1999 (ADVANCE - integrated simulation of transport demand, traffic flow and car emissions)

Mahut, M.: Discrete flow model for dynamic network loading, PhD Thesis, Département d'informatique et de recherche opérationelle, Université de Montréal 2001

Wiedemann, R.: Simulation des Verkehrsflusses. Schriftenreihe des Instituts für Verkehrswesen, Heft 8, Universität (TH) Karlsruhe (seit 2009 KIT – Karlsruher Institut für Technologie), 1974 (Traffic flow simulation)

Wulffius, H.; Bormet, J.: Projektierung verkehrsabhängiger Lichtsignalsteuerungen. Verkehr und Technik, Heft 2, pages 49-53, 1994 (Project planning for traffic-actuated signal controls)

References for pedestrian simulation and the social force model

For a an excellent overview of the social force model: **Helbing, D.; Johansson, A.**: Pedestrian, Crowd and Evacuation Dynamics. In: R.A. Meyers (Editor). Encyclopedia of Complexity and System Science. Volume 16, p. 6476. Springer, Berlin Heidelberg New York, 2009

Galea E.R. (Editor): Pedestrian and Evacuation Dynamics: 2nd International Conference, Old Royal Naval College, University of Greenwich, London, CMS Press, 2003

Helbing, D.; Molnár, P.: Social force model for pedestrian dynamics. In: Physical Review E 51 (5), pages 4282-4286, 1995

Helbing, D.; Farkas, I.; Vicsek, T.: Simulating dynamical features of escape panic. In: Nature, 407, pages 487–490, 2000

Helbing, D.; Farkas, I.; Molnár, P.; Vicsek, T.: Simulation of Pedestrian Crowds in Normal and Evacuation Situations. In: Schreckenberg and Sharma (Editors) Pedestrian and Evacuation Dynamics, Duisburg 2001. Springer Berlin Heidelberg, 2002

Hoogendoorn S.P. ; Bovy P.H.L. ; W. Daamen: Microscopic Pedestrian Wayfinding and Dynamics Modelling. In: Schreckenberg and Sharma, pages 123-154, 2002

Johansson, A.; Helbing, D.; Shukla, P.K.: Specification of the Social Force Pedestrian Model by Evolutionary Adjustment to Video Tracking Data. In: Advances in Complex Systems 10(4), pages 271–288, 2007

Schreckenberg M.; Sharma S. D. (Editors): Pedestrian and Evacuation Dynamics, Duisburg, Januar 2002. Springer Berlin Heidelberg, 2002

Schadschneider, A.; Klingsch, W.; Klüpfel, H.; Kretz, T.; Rogsch, C.; Seyfried, A.: Evacuation Dynamics: Empirical Results, Modeling and Applications. In: R.A. Meyers (Editor), Encyclopedia of Complexity and System Science. Part 5, p. 3142. Springer, Berlin Heidelberg New York, 2009

Werner; Helbing, D.: The Social Force Pedestrian Model Applied to Real Life Scenarios. In: Galea (Editor) Proceedings of the 2nd Conference on Pedestrian and Evacuation Dynamics, Greenwich. CMS Press Greenwich, 2003

21 Index

PTV VISSIM

the mind of movement

PTV GROUP
Haid-und-Neu-Str. 15
76131 Karlsruhe
Germany
Phone +49 (0) 721 96 51- 300
E-Mail info@vision.ptvgroup.com
www.ptvgroup.com
www.vision-traffic.ptvgroup.com