Contents

Preface

About *SAPPORO*

The *SAPPORO* integrated traffic control system is a flexible traffic simulation environment. The control structure to manipulate the road network database and the corresponding density calculus database as well as the simulation process are described in this user's manual. A detailed description of *SAPPORO*'s goals, specifications and implementation you will find in

***SAPPORO* Annual Report Phase 2, March 1992**
Intelligent Traffic Control for Urban Networks
by Bernd Wild, Bernd Schuetze, Olaf Dalchow, Axel Saffran
and Ron Zohar

About this book

This user manual gives you a short introduction and overview of the *SAPPORO* system user interface. If you are already familiar with the *SAPPORO* system, you can use this manual as a reference guide if any further problems or questions about *SAPPORO*'s behaviour occur during a *SAPPORO* session. When you are a *SAPPORO* beginner, this manual will guide

you through a typical *SAPPORO* session, so that you can learn the functionality of *SAPPORO* by studying this book from chapter 1 to chapter 6.

In chapter 1 the screen layout of the *SAPPORO* surface will be described. You can learn how to extract extensive information of your data objects from the screen. Chapter 2 is concerned with the operations on the network topology and the density calculus database, such as loading, saving and creating new road networks and density calculi and how to get informations about them from the screen. Chapter 3 tells you everything about the simulation of traffic on a loaded road network with a specific density calculus while chapter 4 describes the postprocessing of simulated data, as for example the replay of the simulation run or the examination of the generated objects event lists. Chapter 5 offers you some more, auxilliary commands that are helpful for an extensive *SAPPORO* session. Chapter 6 presents the integrated intersection designer tool IDAR. In the Appendix you will find an additional short, graphical description of the most important window screens and a complete *SAPPORO* command index.

In this manual, for each *SAPPORO* command the aim and the effects are described for short, followed by the dialog instructions which show and describe all appearing dialog windows.

Conventions

When you move your mouse pointer on the *SAPPORO* window, some graphical or textual objects will be highlighted. This tells you that there is the possibility to get more information on these objects by pressing the left mouse button on these objects while they are highligthed. This process we call '*clicking* a screen object'. Some objects provide additional commands when you press the right mouse button on the highlighted object. These commands will be explicit described in the following chapters.

While working with the *SAPPORO* system two types of windows will appear starting a user dialog. When *SAPPORO* presents a menu of selectable items, a not movable and not scrollable window appears, from which you shall select one item by clicking on it or simply click Abort **to get rid of this window. In all other cases, where you can change or arrange the windows contents, a movable and scrollable window is presented. These windows will show you two buttons to close the window. If you press the** Abort **button (click on it with the mouse) the window disappears without any action and without any effect on your data objects. If you press the** Done **button the selected command for this window will be executed, us-**

ing the actual contents of this window and the window will disappear too.

Don't get worried, when nothing happens immediately after you have pressed a mouse button or entered a command. The creation of CLIM-windows is a time-consuming procedure, just wait a few seconds before calling your system operator.

Getting Started

For getting started with your *SAPPORO* application be sure, that you have the execution bits set on sapporo **and** cl_nih **in your current *SAPPORO* directory.**

execute the sapporo command

From now on we assume that the current directory in which sapporo**,** cl_nih **and** SAPPORO-DATA **reside is called** <current-dir> **and that the host on which the display should appear is called** <display-host>**. Note that you need X running on the** <display-host> **but not necessarily on the host who execute the sapporo image.**

At the cl> **prompt input the following lines**

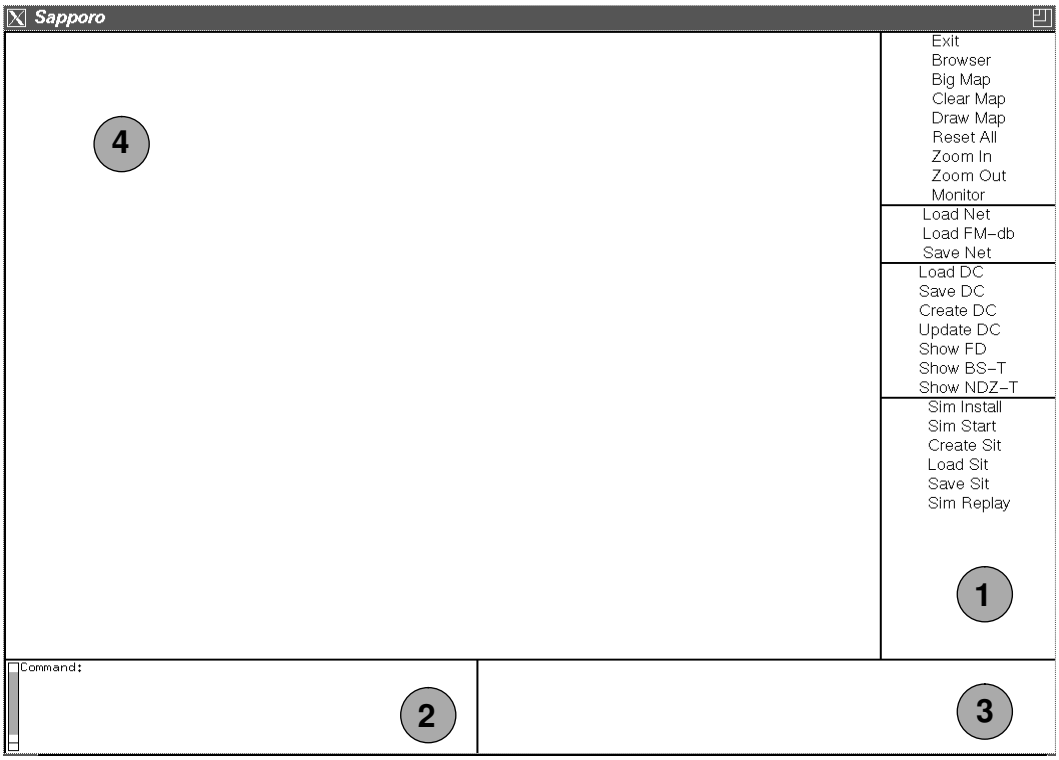
```
cl> (excl::generate-library-pathnames "<current-dir>/")

cl> (setf sa::*default-database-directory* "<current-dir>/SAP-
PORO-DATA/databases/")

cl> (setf sa::*default-filemaker-database-directory* "<current-
dir>/SAPPORO-DATA/FM/")

cl> (sa::reset-sapporo "<display-host>")
```

After these commands *SAPPORO* starts to bring up its window on <display-host>**.**

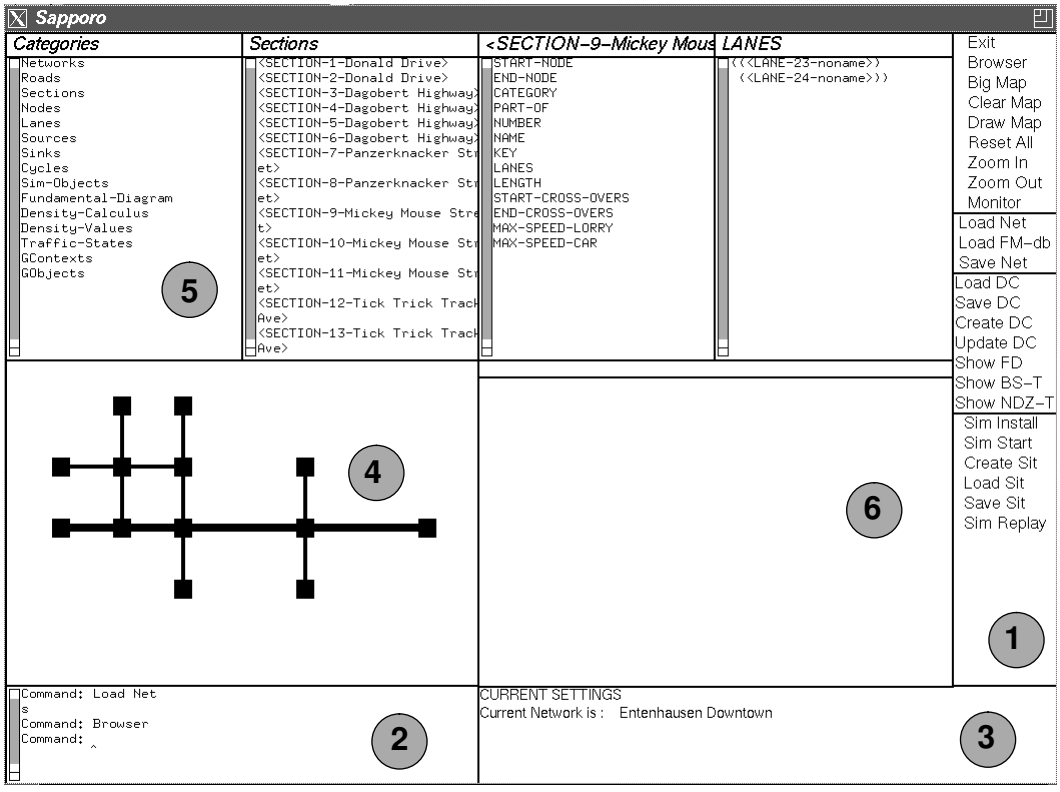
1 Screen layout

1.1 About the Screen layout

When you start your *SAPPORO* application you will get a pop up window on your Screen showing one of the two following *SAPPORO* layouts, called the *'Big-Map'* layout and the *'Browser'* layout. During your *SAPPORO* session you will be able to switch between these two layouts at every timepoint. The *'Big-Map'* layout shows you a bigger Map of the current loaded road network, while the *'Browser'* layout gives you the possibility to get actual informations of all dynamic *SAPPORO* objects. The *'Browser'* layout offers an object browser, an animation pane for simulation monitoring and a smaller pane for displaying the road map.



The *'Big-Map'* layout



The *'Browser'* layout

All numbered subwindows in this two layouts are called *panes* and have a specific function for the system-user-interaction.

The Menu pane

In this column you find all possible *SAPPORO* control commands. They are grouped in the four categories *control, network access, density calculus access* and *simulation*. All shown commands commands are mouse-sensitive.

The Interaction pane

In this pane you can enter your *SAPPORO* commands 'by hand'. If you enter the unambiguous beginning of a *SAPPORO* command and press the Space **key, your command will automatically be completed. Enter your written command by pressing the** Return **key.**

The Settings pane

All necessary informations about your current working environment, such as the current road network, the current density calculus (if loaded), the number of simulation objects installed and the name of the current simulation start situation are displayed in this pane.

The Map pane

Your current road network is graphically presented in this pane. All node-objects are displayed as mouse-sensitive squares and all section-objects are displayed as mouse-sensitive rectangles. The network is scaled to fit in this Map pane.

The Browser pane

In this four-column-pane allows you to examine very quickly the contents of all dynamic system objects. The first column shows a list of all object categories. If you select one of these categories by clicking on it, you will get a list of all existing instances of this object category in the second column of this pane. clicking on one of these instances displays all of its slot-names in column three. clicking on a slot-name shows you its contents in column four. All columns are verticall scrollable to access all of the list entries.

The Simulation pane

If you have chosen to monitor a simulation run or Replay a former simulation run, a graphical animation of the roads traffic is shown in this pane.

1.2 Switching to the Big-Map layout

At any time you want to have a bigger view of the current road network, you can switch to the "Big-Map" layout by the Big Map command. This command does not have any effects on the current settings and your current object instances. Only the simulation pane and the Browser panes are cleared.

INSTRUCTIONS

To switch to the Big-Map layout:

1. **Click in the main menu command** Big Map **or type in the command** Big Map **in the instruction pane.**

   The window layout will change to the Big-Map layout.

1.3 Switching to the Browser layout

At any time you want to have more informations about your *SAPPORO* objects ,you can switch to the "Browser" layout by the Browser command. This command does not have any effects on the current settings and your current object instances. Only the simulation pane and the Browser panes are cleared.

INSTRUCTIONS

To switch to the Browser layout:

1. **Click on the main menu command** Browser **or type in the command** Browser **in the instruction pane.**

1.4 Exiting the system

After a *SAPPORO* session you have to exit the *SAPPORO* application by a main menu command. The *SAPPORO* main window will be closed and you will find yourself back on the point where you started the *SAPPORO* application.

INSTRUCTIONS

To exit the *SAPPORO* application:

1. **Click on the main menu command** Exit **or type in the command** Exit **in the instruction pane.**

   Be sure you have not left any unsaved data objects left, that will be lost when you quit *SAPPORO*. You will not be asked for any confirmation on exiting !

2 Database and Inspection Commands

The database of the system *SAPPORO* contains the description of all the network objects, which contain the topological description of the network and the description of the traffic flow entering that network, and the description of the objects of the "density calculus". Those objects are required to generate the generate the simulation network objects and to start a simulation run.

2.1 Loading the textual description of the network topology

At first, you have to load the objects describing the network topology. Those objects are stored as textual files (created using Macintosh FileMaker II**) or stored in the system *SAPPORO* using its own data**

structures.

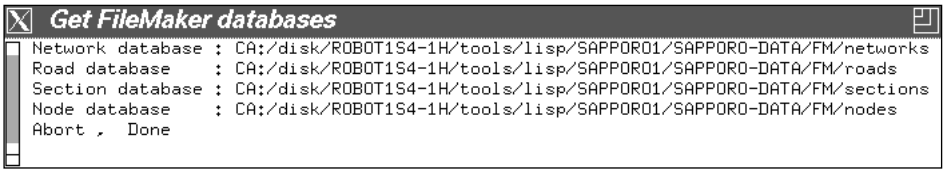
INSTRUCTIONS

To load a network from the FileMaker Database:

1. **If you have used another network in the system *SAPPORO* before, then click in the main menu command** Reset All **by the arrow to remove the old network.**

2. **Click in the main menu command** Load FM-db **by the arrow.**

3. **The pop up menu** Get FileMaker databases **appears:**



   It shows the path names to access those files, that contain the textual representation of the network ("<current-dir>/SAPPORO-DATA/FM/networks"**), of the roads (**"<current-dir>/SAPPORO-DATA/FM/roads"**), of the sections (**"<current-dir>/SAPPORO-DATA/FM/sections"**) and of the nodes (**"<current-dir>/SAPPORO-DATA/FM/nodes"**).**

   Click Done **to use the given file names.**

   If you want to use different file names, click on one of the names. Now, you can edit that name. Type a new file name and press Return**. Finally, click** Done **to use the new file names.**

***SAPPORO* loads those files and creates the network objects. The nodes and sections of the created road network are displayed in the Map pane.**

2.2 Saving the current network

All the generated network objects can be stored in the *SAPPORO* database. By doing so, you have not always to load the textual (FileMaker) **database and create all network objects next time.**

INSTRUCTIONS

To save the network objects:

1. **Click the main menu command** Save Net **by the arrow.**

Using the name of the network, in the directory "<current-dir>/SAPPORO-DATA/databases/<network-name>" **a subdirectory is created containing files with the information about the network objects.**

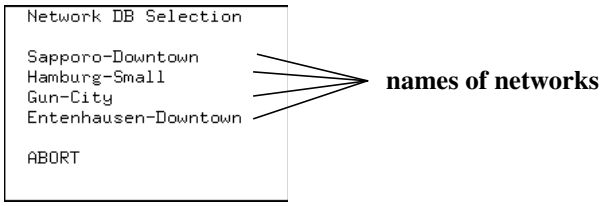
2.3 Loading the network from the *SAPPORO* database

If you have stored a network in the *SAPPORO* database, you can load the information about the network objects directly. Thus, all the calculations have to be redone, which are necessary to generate the network objects, if you load the textual representation of the network (see Chapter 2.1).

INSTRUCTIONS

To load a network from the *SAPPORO* database:

1. **Click in the main menu command** Reset All **by the arrow to remove an old network, which might have been used in the system *SAPPORO* before.**
2. **Click in the main menu command** Load Net **by the arrow.**
3. **The pop up menu** Network DB Selection **appears:**



To load the network objects, click on one of the names in the given catalog (for example on "Entenhausen-Downtown"). The files are loaded and the network objects are generated.

Click ABORT **to prevent loading of files from the *SAPPORO* database (for example if no networks are listed).**

***SAPPORO* loads those files and creates the network objects.**

If you have modified the network objects, you can save the network objects by clicking the main menu command Save Net**. Using the name of the network, in the directory** "<current-dir>/SAPPORO-DATA/databases/<network-name>" **a subdirectory is overwritten. The files contain the modified information about the network objects.**

2.4 Inspection of the network elements

After loading the road network you can inspect and edit the network elements in the *Object Browser*. Additonally, you get the object information about a network object directly from the nodes displayed in the Map pane of the 'Big Map' layout.

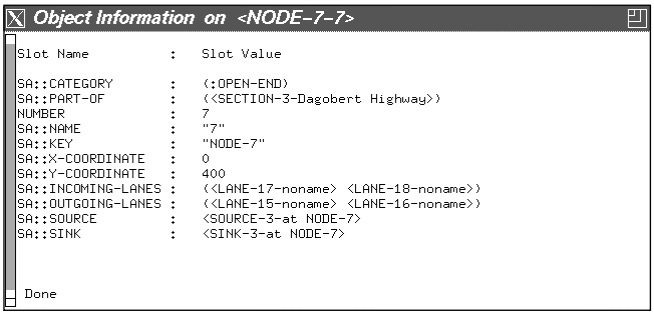
INSTRUCTIONS

To get the information about a node directly in the Map pane:

1. **Move the mouse pointer over the node in the Map pane until it is highlighted and click on it using the right mouse key.**
2. **Then the following pop up menu appears:**

Click on Get Info <NODE>**.**

3. **The pop up menu** Object information on <NODE> **appears:**



Click Done **to close this window.**

If you are in the 'Browser' layout, you can click on the node to select the node. Thus, the object information about that node is displayed in the object browser.

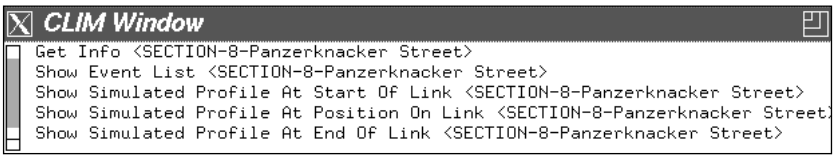
You can get the object information about the sections displayed in the Map pane of the 'Big Map' layout.

INSTRUCTIONS

To get the information about a section directly in the Map pane:

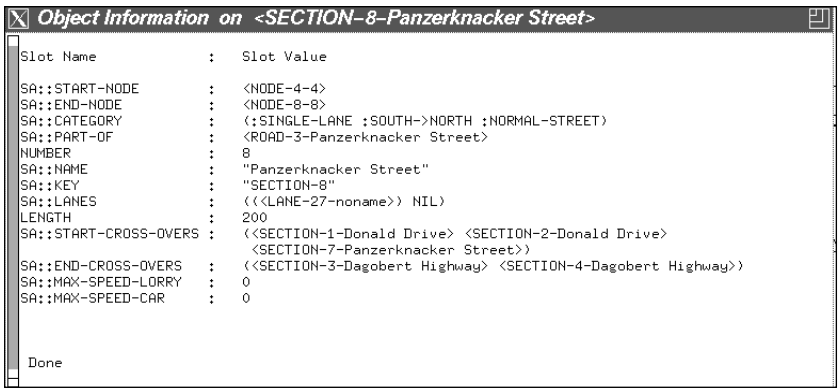
1. **Select the section in the map.**

2. **If the arrow is in a road map window, click on the section using the right mouse button. Then the following pop up menu appears:**



Click on Get Info <SECTION>**.**

3. **The pop up menu** Object information on <SECTION> **appears:**



Click Done **to close this window.**

If you are in the 'Browser' layout, you can click on the node to select the node. Thus, the object information about that node is displayed in the object browser.

2.5 Creation of a "density calculus"

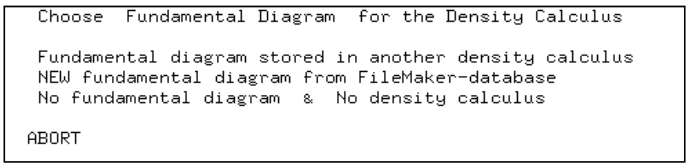
After loading the network, you can create a "density calculus" using the information given by a fundamental diagram. You can also load a "density calculus", which has already been created for the loaded network or for an other network stored in the SAPPORO database.

INSTRUCTIONS

To create a new "density calculus":

1. **Click in the main menu command** Create DC **by the arrow.**
2. **The pop up menu** Choose Fundamental Diagram for the Density Calculus

appears:



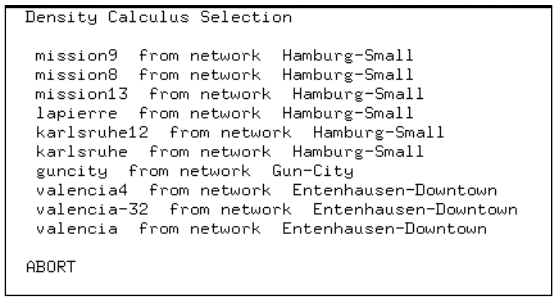
It shows the possibilities of creating a "density calculus".

If you click Fundamental Diagram Stored in another Density Calculus**, then the system looks for the fundamental diagrams of all the "density calculi" already created.**

If you click New Fundamental Diagram from FileMaker-database**, then the system looks for the fundamental diagrams stored in the FileMaker database (path name** "<current-dir>/SAPPORO-DATA/FM/"**).**

You must click No Fundamental Diagram & No Density Density Calculus **or just** ABORT **to prevent loading of files and creation of a "density calculus".**

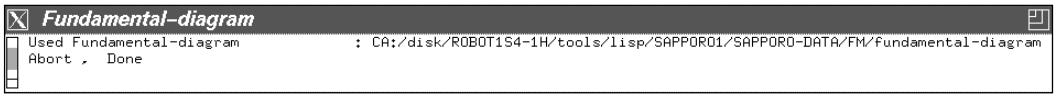
3. **If you have clicked** Fundamental Diagram Stored in another Density Calculus**, then the pop up menu** Density Calculus Selection **appears:**



It shows the list of all existing "density calculi" used for the simulation in a road network. A click on the descriptor of the density calculus causes loading of the fundamental diagram used for the creation of that density calculus.

Click ABORT **to prevent loading of fundamental diagram.**

If you have clicked New Fundamental Diagram from FileMaker-database**, then the pop up menu** Fundamental-Diagram **appears:**

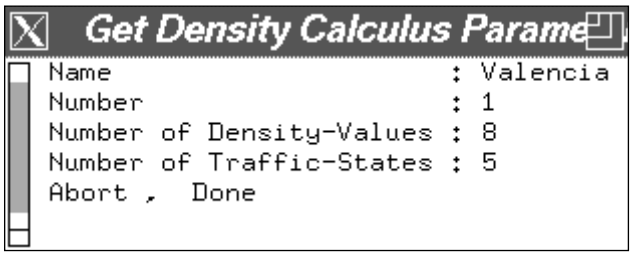


It shows the default name of the file, in which the information about a fundamental diagram is stored in the FileMaker database prevent loading of files and the creation of a "density calculus".

Click Done **to use the given file name.**

If you want to use different file name, click on the name. Now, you can edit that name. Type a new file name and press Return**. Finally, click** Done **to use the new file name.**

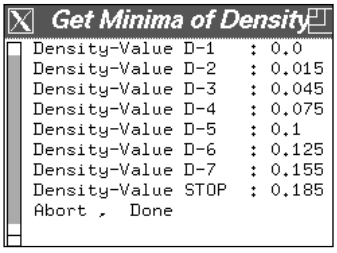
4. **Now the pop up menu** Get Density Calculus Parameters **appears:**



In that menu you can set the name of the "density calculus" and a number for the internal management. You can also set the number of density values and traffic states.

To edit one of those values, click on it. Type a new name or number and press Return**. Finally, click** Done **to use all the new values.**

5. **The pop up menu** Get Minima Of Density Values **appears:**

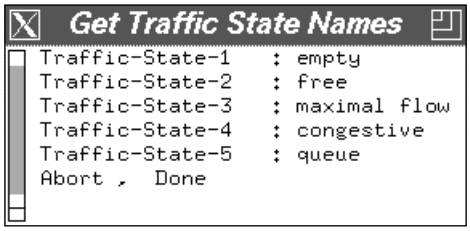


In that menu you can set the lower limits of the density intervals, which should be assigned to the qualitative density values.

The lower border of the first density interval is always 0.0 . The lower border of the last density interval (attached to "STOP") must be smaller than the maximal density value given by the fundamental diagram.

To edit one of those values, click on it. Type a new number and press Return**. Finally, click** Done **to use all the new values.**

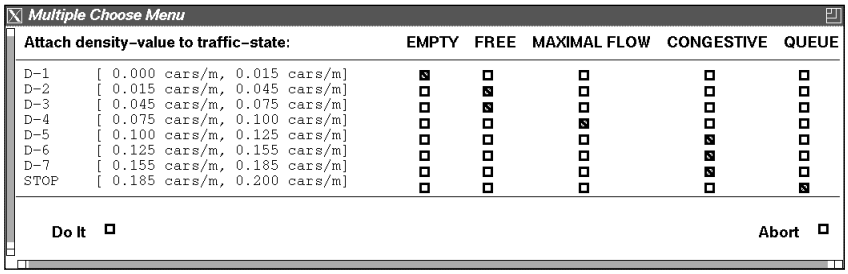
6. **The pop up menu** Get Traffic State Names **appears:**



In that menu you can set the name of the traffic states, which should describe the density values.

To edit one of those values, click on it. Type a new name and press Return**. Finally, click** Done **to use all the new names.**

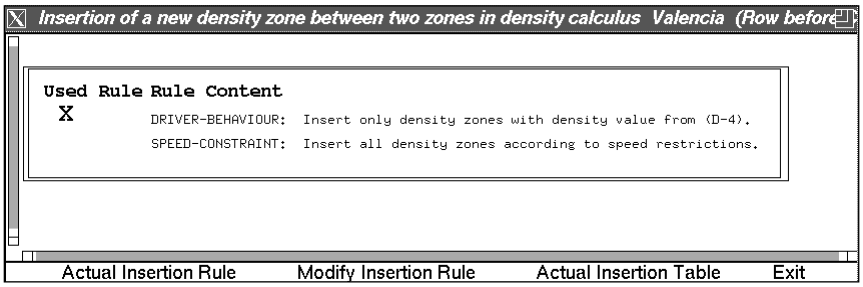
7. **The pop up menu** Attach Density Values to Traffic State **appears:**



In that menu you can assign the qualitative density values to one of the traffic states. To make an assignment, click on one of the boxes in a row.

Click DoIt **to use the assignments.**

8. **Finally, the output window** Insertion of a new density zone **appears:**



That window shows the rule used to insert new density zones, if you click on Actual Insertion Rule**. To see the created new density zone table, you have to click on** Actual Insertion Table **in the command line. To modify the insertion rule, click on** Modify Insertion Rule **in the command line. Then a anew new density zone table will be created.**

Click Exit **to close this output window.**

You can inspect and edit the objects of the "density calculus" in the *Object Browser*. All the generated "density calculus" objects can be stored in the *SAPPORO* database.

2.6 Saving of a “density calculus”

After creating a new “density calculus” you have to save it in the database for later access

INSTRUCTIONS

To save the density objects:

1. **Click the main menu command** Save DC **by the arrow.**

   Using the name of the road network, in the directory “<current-dir>/SAPPORO-DATA/databases/<network-name>” **a subdirectory** “<density-calculus-name>” **is created containing files with the information about the “density calculus” objects.**

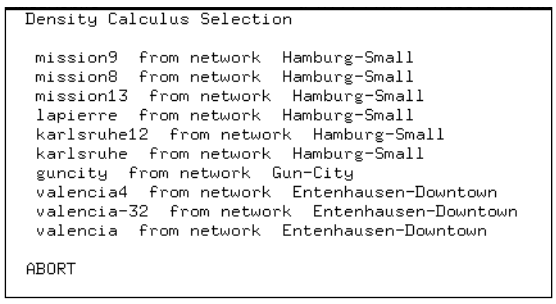
2.7 Loading of a “density calculus”

After loading the network, you can load a “density calculus”, which has already been created for the loaded network or for an other network stored in the *SAPPORO* database.

INSTRUCTIONS

To load a “density calculus”:

1. **Click in the main menu command** Load DC **by the arrow.**
2. **The pop up menu** Density Calculus Selection **appears:**



   It shows the list of all existing “density calculi” used for the simulation in a road network. A click on the descriptor of the density calculus causes loading of that density calculus. *SAPPORO* creates all objects of the “density calculus”.

   Click ABORT **to prevent loading a “density calculus”.**

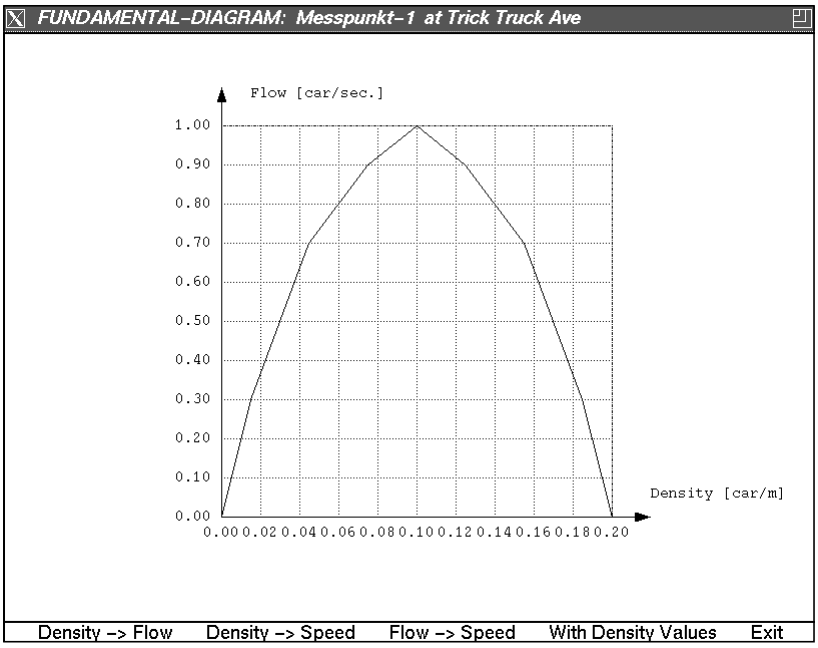
2.8 Inspection of the fundamental diagram

After creating or loading a "density calculus" you can inspect and edit all the "density calculus" objects using the Object Browser. You can have a closer look at the fundamental diagram used for the creation of the "density calculus".

INSTRUCTIONS

To look at the fundamental diagram of the "density calculus":

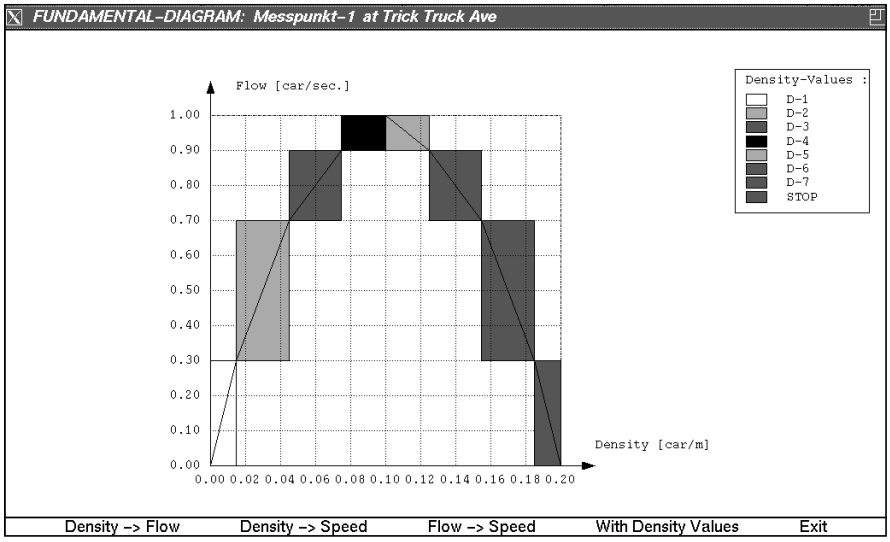
1. **Click in the main menu command** Show FD **by the arrow.**
2. **Then the output window** Fundamental-Diagram **appears. To see the classical fundamentaldiagram click on** Density -> Flow **from the command line:**



If you click on Density -> Speed **or** Flow -> Speed **from the command line, the corresponding density-speed-relation or flow-speed relation is displayed.**

3. **To see those diagrams and in addition to that the intervals assigned to the qualitative density values, click on** With Density Values **from the**

command line:



According to the displayed graph, the intervals of the traffic values (density, speed or flow) attached to qualitative density values are displayed.

Click Exit **to close this output window.**

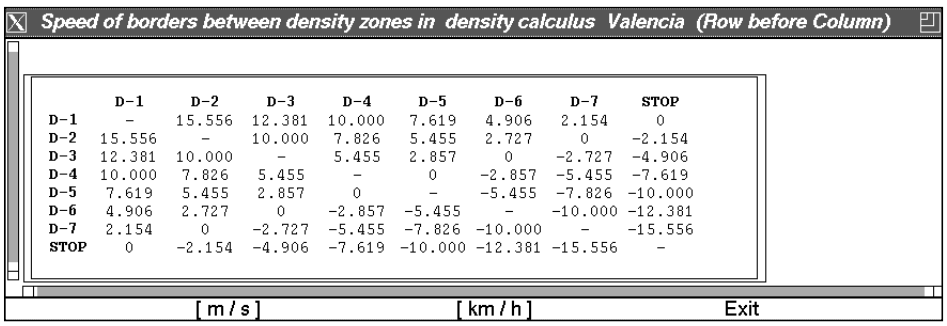
2.9 Inspection of the border speed table of the “density calculus”

After creating or loading a “density calculus” yon can inspect the created matrix on the border speeds between density zones.

INSTRUCTIONS

To display the border speed table:

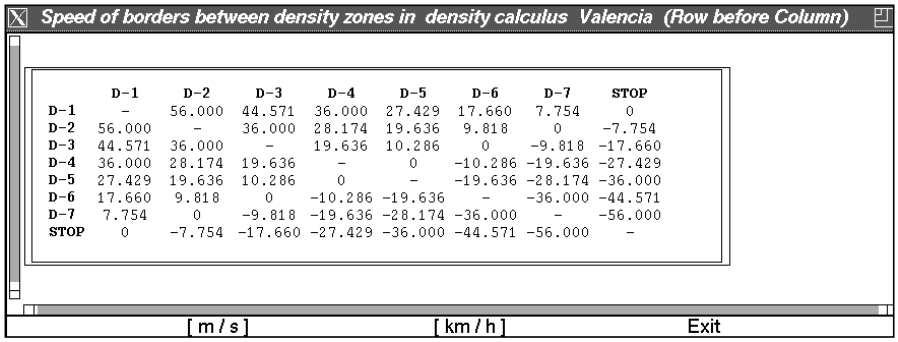
1. **Click in the main menu command** Show BS-T **by the arrow.**
2. **Then the output window** Speed of Borders between density zones **appears. To see the matrix, click on** [m/s] **from the command line:**



	D-1	D-2	D-3	D-4	D-5	D-6	D-7	STOP
D-1	–	15.556	12.381	10.000	7.619	4.906	2.154	0
D-2	15.556	–	10.000	7.826	5.455	2.727	0	-2.154
D-3	12.381	10.000	–	5.455	2.857	0	-2.727	-4.906
D-4	10.000	7.826	5.455	–	0	-2.857	-5.455	-7.619
D-5	7.619	5.455	2.857	0	–	-5.455	-7.826	-10.000
D-6	4.906	2.727	0	-2.857	-5.455	–	-10.000	-12.381
D-7	2.154	0	-2.727	-5.455	-7.826	-10.000	–	-15.556
STOP	0	-2.154	-4.906	-7.619	-10.000	-12.381	-15.556	–

It shows the table on the border speeds in [m/s]. This table is used for internal calculations.

3. **If you click on** [km/h] **from the command line, the corresponding table on the border speeds in [km/h] is displayed:**



	D-1	D-2	D-3	D-4	D-5	D-6	D-7	STOP
D-1	-	56.000	44.571	36.000	27.429	17.660	7.754	0
D-2	56.000	-	36.000	28.174	19.636	9.818	0	-7.754
D-3	44.571	36.000	-	19.636	10.286	0	-9.818	-17.660
D-4	36.000	28.174	19.636	-	0	-10.286	-19.636	-27.429
D-5	27.429	19.636	10.286	0	-	-19.636	-28.174	-36.000
D-6	17.660	9.818	0	-10.286	-19.636	-	-36.000	-44.571
D-7	7.754	0	-9.818	-19.636	-28.174	-36.000	-	-56.000
STOP	0	-7.754	-17.660	-27.429	-36.000	-44.571	-56.000	-

Click Exit **to close this output window.**

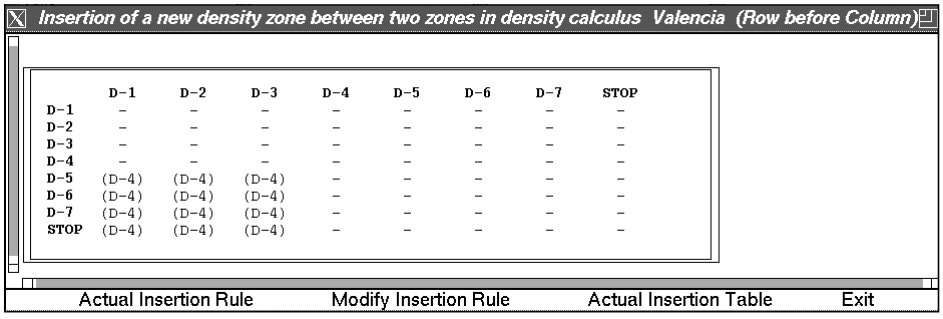
2.10 Inspection of the new density zone table

After creating or loading a "density calculus" yon can inspect the matrix on insertable density values, too. It's calculated using the border speed table of the "density calculus" and a general insertion rule.

INSTRUCTIONS

To display the new density zone table:

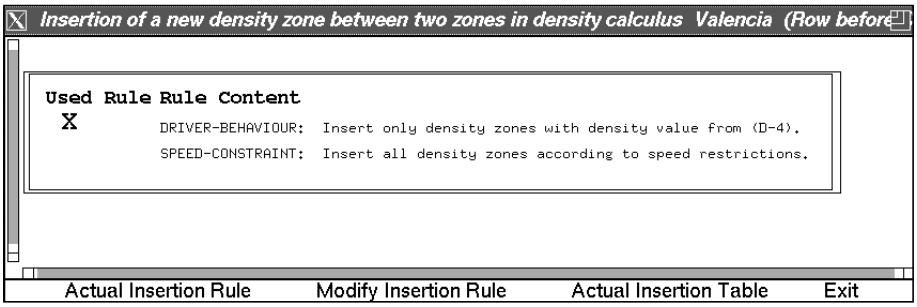
1. **Click in the main menu command** Show NDZ-T **by the arrow.**
2. **Then the output window** Insertion of a new density zone two zones **appears. To view the matrix created by the density calculus, click on** Actual Insertion Table **from the command line:**



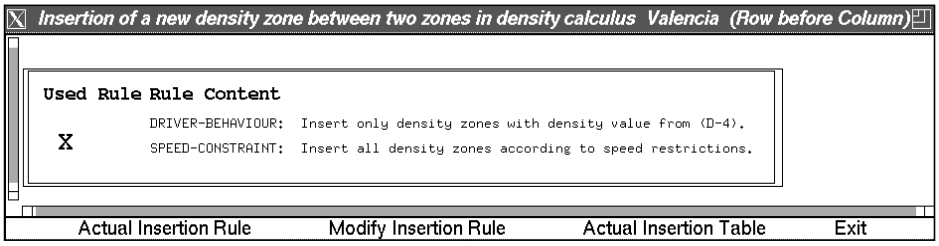
	D-1	D-2	D-3	D-4	D-5	D-6	D-7	STOP
D-1	-	-	-	-	-	-	-	-
D-2	-	-	-	-	-	-	-	-
D-3	-	-	-	-	-	-	-	-
D-4	-	-	-	-	-	-	-	-
D-5	(D-4)	(D-4)	(D-4)	-	-	-	-	-
D-6	(D-4)	(D-4)	(D-4)	-	-	-	-	-
D-7	(D-4)	(D-4)	(D-4)	-	-	-	-	-
STOP	(D-4)	(D-4)	(D-4)	-	-	-	-	-

It shows the table on the lists of density zones insertable between two density zones according to the actually used insertion rule. This table is used for internal calculations during a simulation run.

3. **To view the insertion rule used to create that new density zone matrix, click on** Actual Insertion Rule **from the command line:**

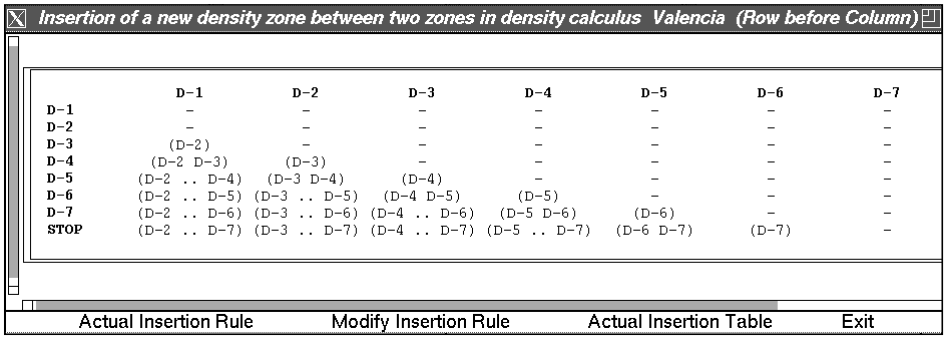


4. **To modify the insertion used to create that new density zone matrix, click on** Modify Insertion Rule **from the command line::**



By doing that, the new density zone table is modified according to the selected rule. All the lists in that table are recalculated.

5. **To view the matrix created according to the modified insertion rule, click on** Actual Insertion Table **from the command line again:**



Click Exit **to close this output window.**

2.11 Update of the “density calculus”

You can inspect and edit all the objects of the “density calculus” in the *Object Browser*. If you changed the traffic density intervals attached to the qualitative density values or the assignment of density values to

traffic states you have to update your “density calculus”

INSTRUCTIONS

To update the density calculus:

1. **Click on the main menu command** Update DC **to update the border speed table and the new density zone table.**

To inspect the result of Update DC**, use the commands described above (**Show FD**,** Show BS-T **and** Show NDZ-T**)**

3 Simulation

For the simulation of traffic in the *SAPPORO* system be sure that you have loaded a current road network and a current density calculus.

The first step to do is to install the simulation environment by creating all necessary simulation objects. Before you start a simulation run with a specific time horizon, you can choose to monitor the simulation in the simulation pane. You can fix every simulated traffic situation on the net using the situation management.

3.1 Installation of the Simulation Objects

Before you can start a simulation you have to create a simulation coordinator object and the simulation objects corresponding to every static node and lane object of the current network. This installation also incorporates the attachment of the dynamic cycle-objects to every crossing and the attachment of qualitative source profiles to every traffic source object.ent density calculus.

INSTRUCTIONS

To install the simulation:

1. **Click in the main menu command** Sim Install **or type in the command** Simulation Install **in the instruction pane.**

   You can browse the installed simulation objects in the object browser starting with the object category Sim-objects. **You can also browse the attached Cycle objects in the** Cycle **category and the attached source objects in the** Source **catagory.**

3.2 Inspecting a traffic source profile

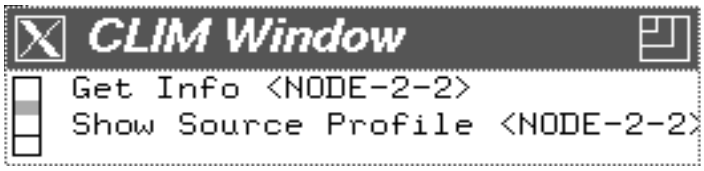
After the installation of a simulation environment, you can inspect the quantitative source profile on a traffic source node, as well as the qualitative source profile computed with the current density calculus.

INSTRUCTIONS

To show a traffic source profile:

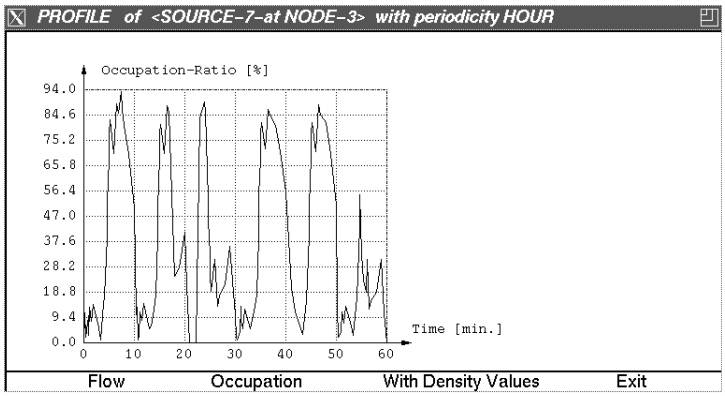
1. **Move the mouse pointer over the source node in the Map pane until it is highlighted and click on it using the right mouse key.**

   The pop up menu Operation on Nodes **appears.**



2. **Click on the menu command** Show Source Profile

   The output window Profile at <source> with periodicity **appears**

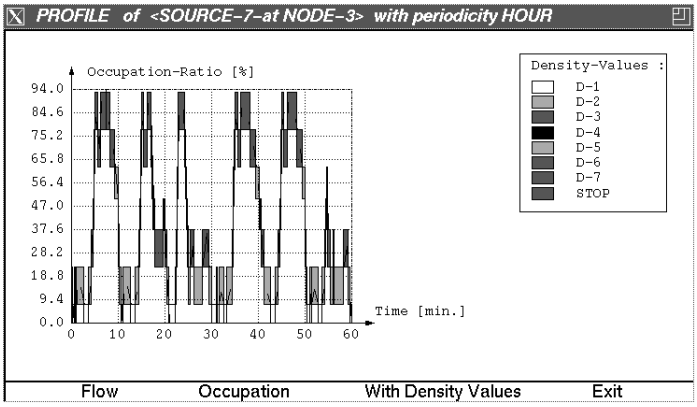


   It shows you the profile of a specific traffic variable on your node.

3. **Click on the menu command for other traffic variables (if there are), to get the profiles for this variables.**

4. **Click on the menu command** With Density Values **to see the transformation of the quantitative source profile to the qualitative source values.**

   The profile output window shows the qualitative source profile for your node.



5. **Click the menu command** Exit **to quit and delete this window**

3.3 Monitoring the Simulation

Before starting the simulation you can switch to the monitoring mode for displaying the density-value description of the network links in the simulation pane while simulating.

INSTRUCTIONS

To Monitor the following simulation:

1. **Click in the main menu command** Monitor **or type in the command** Monitor **in the instruction pane.**

   The Title of the simulation pane tells you that the monitoring mode is switched on

   Note that monitoring a simulation will speed down your simulation velocity dependent to the monitored network size.

2. **To switch the monitoring mode off click on the main menu command** Monitor **again, or type in the command Monitor in the instruction pane again.**

3.4 Starting a Simulation run

Before starting the simulation be sure that you have loaded a network, a density calculus and that you have installed the simulation environment afterwards.

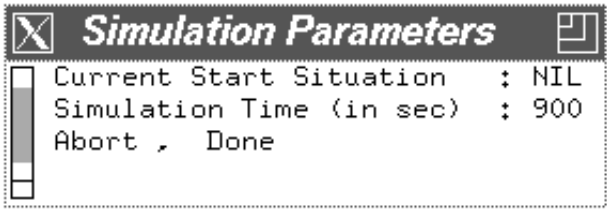
The simulation of the traffic behaviour computes the object-specific event-lists for every simulation object. These event-lists can be inspected after a simulation run using the browser. They are the root data for the later evaluation of a simulation run, described in chapter four.

INSTRUCTIONS

To start the simulation:

1. **Click in the main menu command** Sim Start **or type in the command** Simulation Start **in the instruction pane.**

   The pop up window Simulation **appears**



2. **Click on the name of the current start situation to change the starting traffic situation name. Enter your choice by pressing the** Return **key. If** NIL **or a not known situation name is entered an empty net at the start timepoint will be simulated.**
3. **Click on the simulation time value to change the default value for the simulation time horizon. Enter your choice by pressing the** Return **key.**
4. **Start the simulation by clicking on the menu command** Done**.**

   A time counter in the simulation pane title tells you progress in simulation time. When you selected the monitoring mode, you will see an animation of the density flow on your network in the simulation pane.

   Note, that there is no possibility to stop the simulation process, so be careful with the choice of too large simulation time horizons !

3.5 Creating a Traffic situation

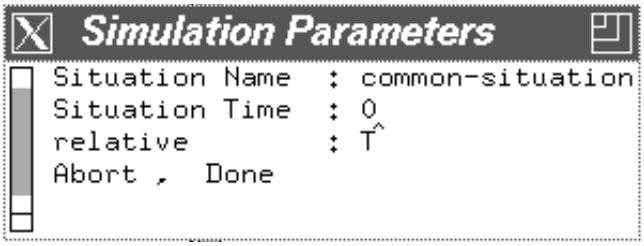
After a simulation run you can fix a traffic situation on your network at any timepoint in your simulated time interval to use it as a basis for later simulation.

INSTRUCTIONS

To create a situation:

1. **Click in the main menu command** Create Sit **or type in the command** Create Situation **in the instruction pane.**

   The pop up window Simulation Parameters **appears**



2. **Click on the situation name to specify the starting traffic situation name. Enter your choice by pressing the** Return **key. Note, that all blanks in the namestring are later changed to "-" for file management reasons if you save this situation.**

3. **Click on the situation time value to specify the default value for the situation time. The specified value has to be in the interval from 0 to the simulated time horizon. Enter your choice by pressing the** Return **key.**

4. **Start the creation of your situation by clicking on the menu command** Done**.**

   Note that creating a new situation replaces the actual current situation .

3.6 Saving a Traffic situation

If you have created a situation and you want to store it in your traffic database for later access, SAPPORO saves your situations as files on the <current-dir>/SAPPORO-DATA/databases/ **directory with an individual path depending on the current network and the current density calculus:**

<network-name>/situations/<DC-name>/<situation-name>

INSTRUCTIONS

To save a situation:

1. **Click in the main menu command** Save Sit **or type in the command** Save Situation **in the instruction pane.**

   Your situation is saved in the traffic database with the actual situation name.

   Note that saving a situation overwrites a already saved situation with the same name !

3.7 Loading a Traffic situation

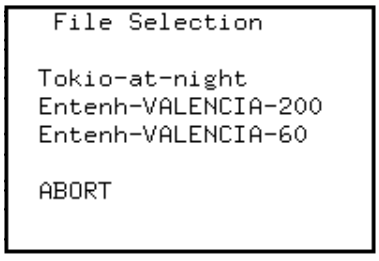
For the access of precalculated traffic situations, you can load all saved situations you have stored with the current network and the current density calculus as current situation.

INSTRUCTIONS

To load a stored situation:

1. **Click in the main menu command** Load Sit **or type in the command** Load Situation **in the instruction pane.**

   The popup window File Selection **appears**



   It shows you all accessible situations in the traffic database for the current network and the current density calculus.

2. **Click on the desired situation to load it as current situation. click** Abort **if you do not want to load any situation.**

   Note that loading a situation replaces the actual current situation !

4 Evaluation Of Simulation Runs

During a simulation run in the system SAPPORO the user can have the simulated traffic on the lanes displayed or only watch the simulation time. After the execution of the simulation it is possible to perform a post-animation of the simulation run. The development of the density zone distribution can be displayed in time-distance diagrams for each lane. The simulated density zone profile can be shown for each point on the lane.

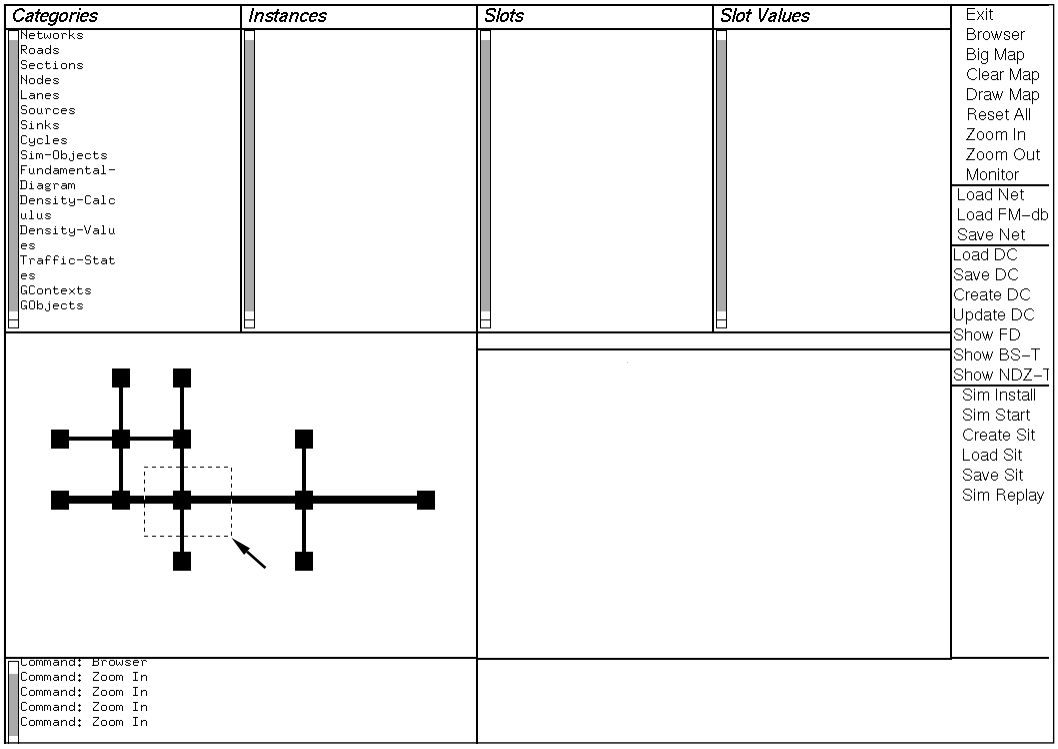
4.1 Displaying A Simulated Traffic Situation

After the execution of a simulation run you can view the simulated traffic situation on any lane in the road network and at any time point (during the simulation time).

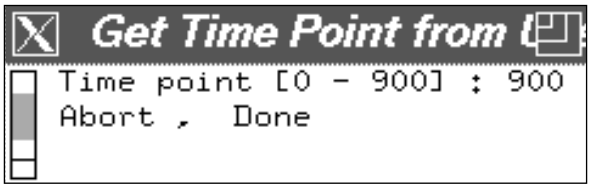
INSTRUCTIONS

To view the simulated traffic situation in the road network:

1. **Click in the main menu command intervals** Zoom In **by the arrow.**
2. **Then move the arrow to the pane, which displays the road map. Click on the map. Surround the area, which you are interested in with the appearing bounding box. Click on the map once more to confirm the selcted area.**



3. **The pop up menu** Get Time Point **appears:**



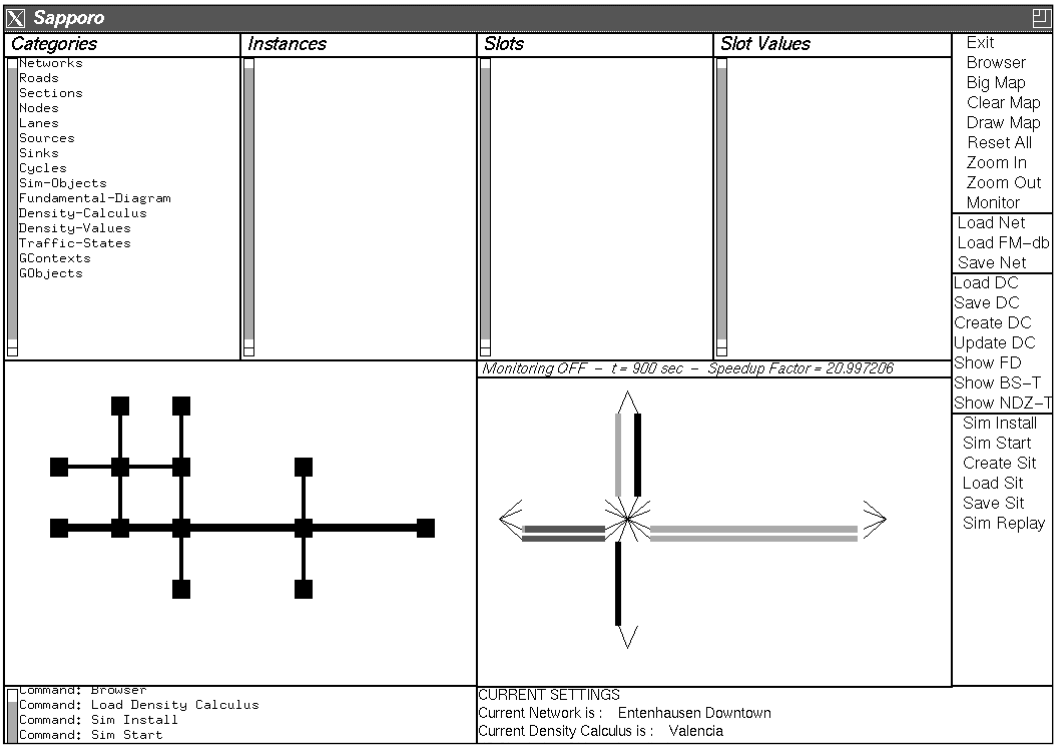
It shows the time point at which the traffic state in the selected area will be displayed.

Click Done **to use the given time point.**

If you want to use a different time point, click on the number. Now, you can edit that nnumber. Type a new number and press Return**. Finally, click** Done **to use this value.**

4. **The traffic state on the lanes in the selected area is now displayed in**

the simulator monitor window:



If you want to undo the selection of an area to be displayed in the simulator monitor window, click on Zoom Out**. Now you can select a new area according to the instructions described above.**

4.2 Post-Animation of A Simulation Run

After the execution of a simulation run you can view the development of the density zones (simulated traffic situation) on any lane in the road network during the simulation run. While the post-animation no new simulation is performed, only the previously performed simulation run is shown.

INSTRUCTIONS

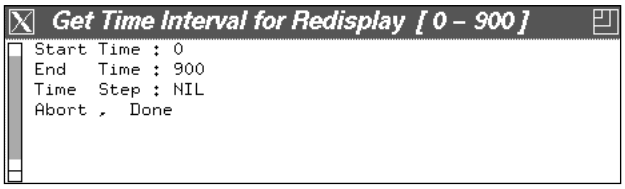
To redisplay a simulation run:

1. **Select the part of the road network, in which the simulation run should be redisplayed by using the main menu command** Zoom In **according to the instructions in chapter 4.1.**

   If you want to see the development of traffic situation in the whole road network, click in the main menu command Zoom Out **by the arrow.**

2. **Click in the main menu command** Sim Replay **by the arrow.**

3. **The pop up menu** Get Time Interval for Redisplay **appears:**

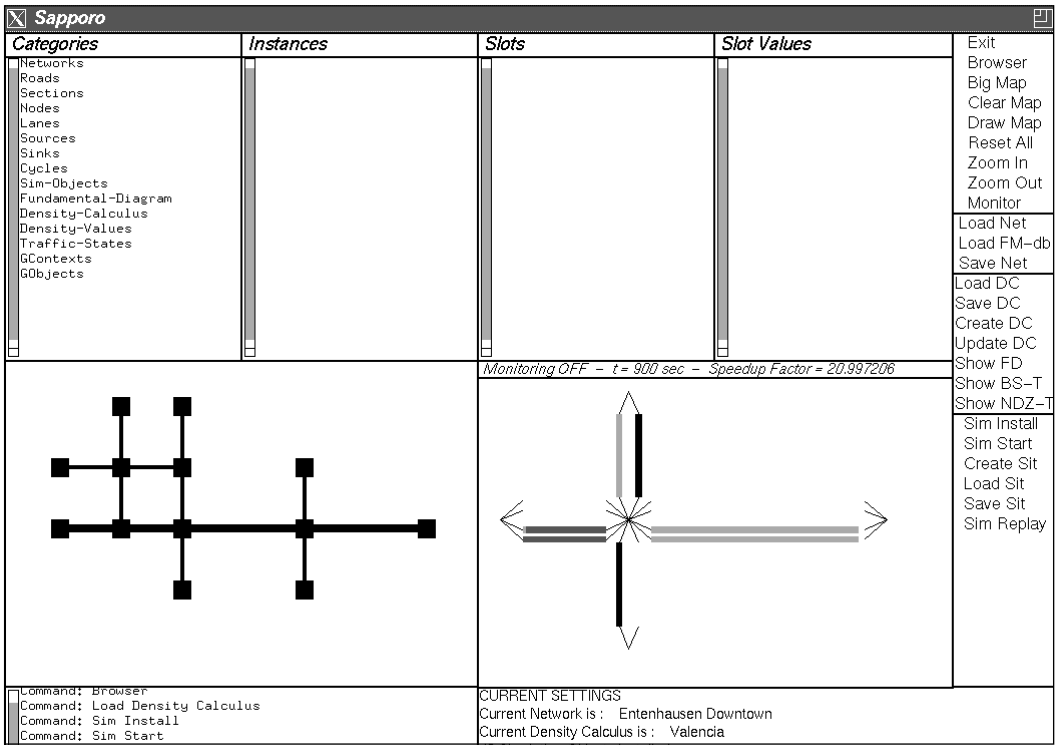


It shows the time interval during which the traffic state in the selected area will be displayed. If the time step is NIL**, the traffic state is only shown at the time points at which events happened in the selected area.**

Click Done **to use the given time interval and time step.**

If you want to use a different time interval or time step, click on the number. Now, you can edit that number. Type a new number and press Return**. Finally, click** Done **to use those values.**

4. **The development of traffic states on the lanes in the selected area is now displayed in the simulator monitor window:**



If you want to undo the selection of an area to be displayed in the simulator monitor window, click on Zoom Out**. Now you can select a new area according to the instructions described above.**

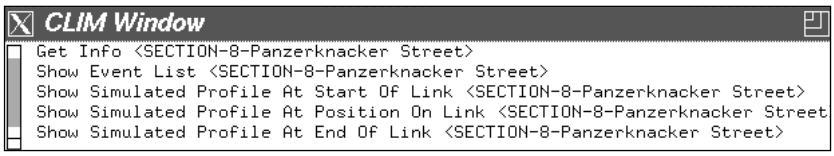
4.3 Display Of The Events In The Time-Space-Continuum

After the execution of a simulation run you can view the development of the density zones on a lane in the road network during the whole simulation run. The development of the density zones on a lane is shown in time-space-diagrams. All events that "happened" on this lane during the simulation run are described. The events are stored in the event lists of the simulation objects.

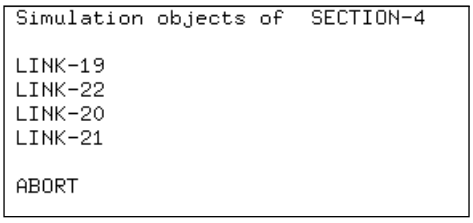
INSTRUCTIONS

To display the development of the density zones on a lane:

1. **Select the lane whose event list should be displayed.**
2. **If the arrow is in a road map window, click on the section, that lane belongs to. Then the following pop up menu appears:**



Click on Show Event List**. The pop up menu** Simulation Objects Of **appears:**



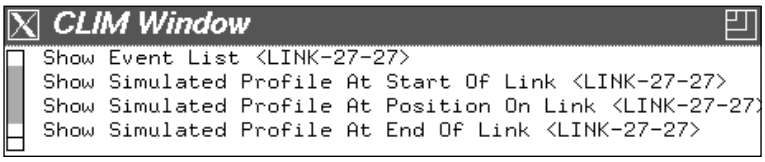
Now you can select that link from the given list of links, which corresponds to the lane you are interested in. Click on that link name to select the link.

Click ABORT **to prevent the opening of the output window for the time-space diagram.**

If the arrow is in the simulation monitor window, select the link, which this lane belongs to. Click on the link using the left mouse button.

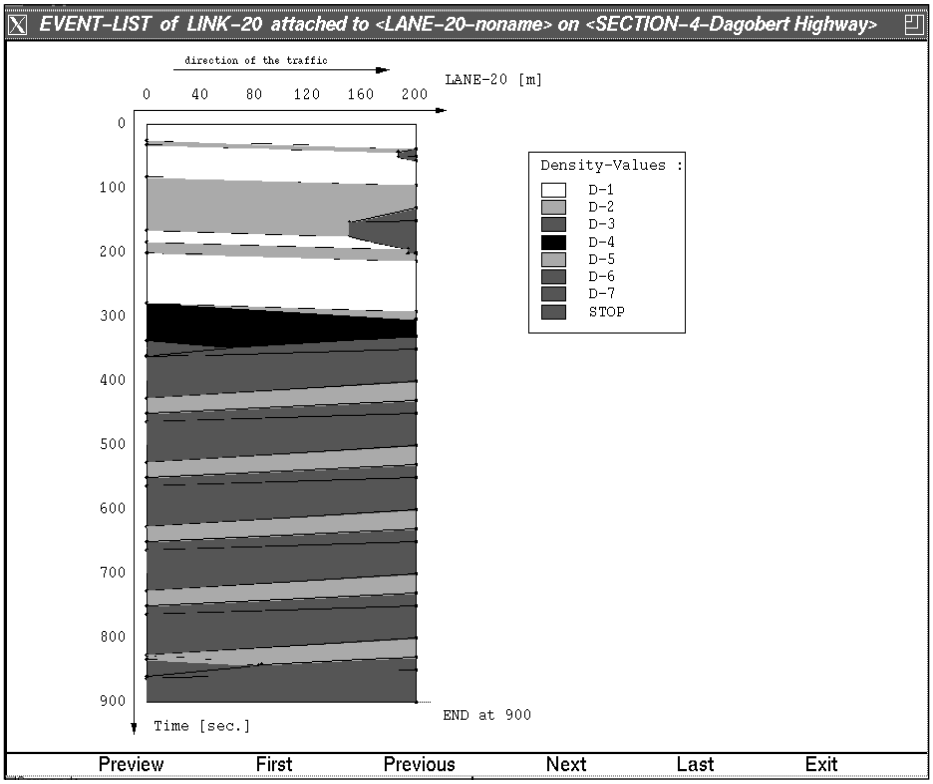
If you click on the link using the right mouse button, then the

following pop up menu appears:



Click on Show Event List**.**

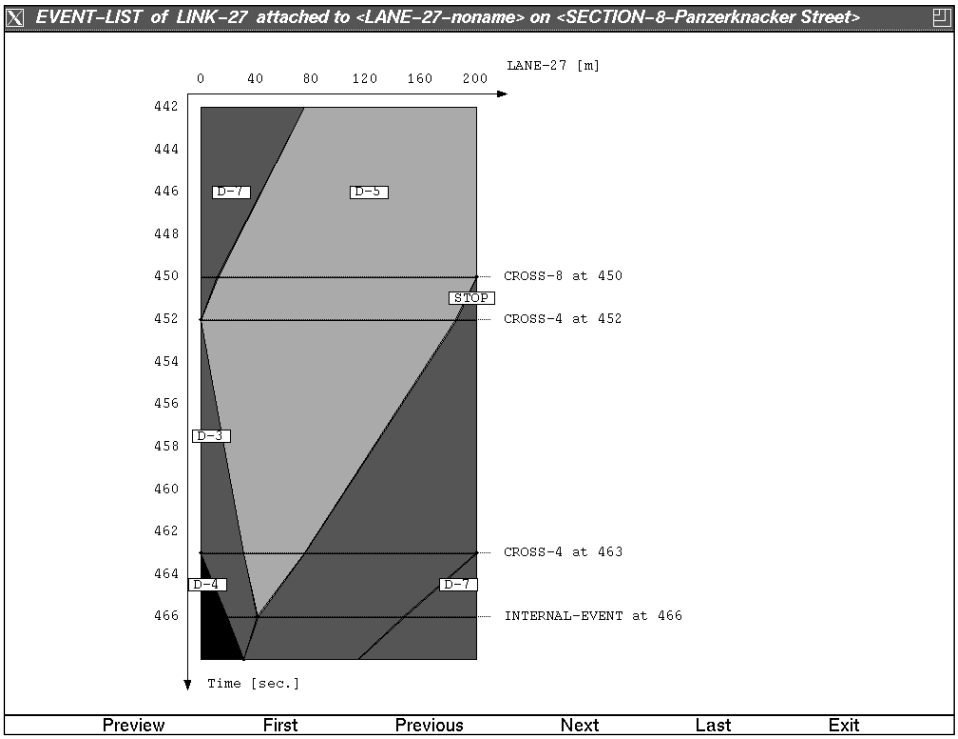
3. **Now the output window** Event-List Of Link **appears. appears. To see an overview of the total time-space diagram, click on** Preview **from the command line:**



This diagram shows the development of the density zones on the lane during the whole simulation run. In a legend the patterns or colors attached to the qualitative density values are shown.

4. **If you want to view the development of the density zones in detail,**

click on First**,** Next**,** Previous **or** Last **from the command line:**



Clicking on one of those commands one part of the time-space diagram is shown. In that diagram all the events happened on the selected lane are described.

Clicking on First **causes the display of the first part of the time-space diagram and clicking on** Last **causes the display of the last part of the time-space diagram. By clicking on** Next **and** Previous **you can move through the whole diagram.**

Click Exit **to close this output window.**

4.4 Display Of Simulated Profiles

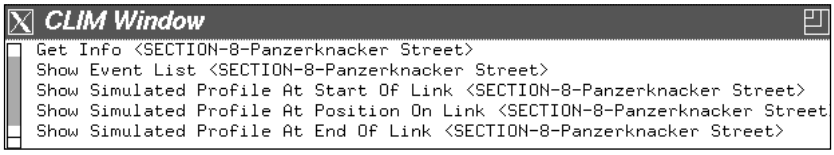
After the execution of a simulation run you can view the development of the qualitative density value in each point on a lane of the road network. The development is described by so-called qualitative profiles. These qualitative profiles might be compared to really measured values or to numeric results of non-qualitative simulator.

INSTRUCTIONS

To display the simulated profile at one point on a lane:

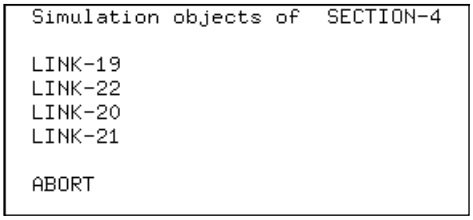
1. **Select the lane whose event list should be displayed.**

2. **If the arrow is in a road map window, click on the section, that lane belongs to. Then the following pop up menu appears:**



Click on Show Simulated Profile At Start Of Link**, if you want to see the profile at the start of a lane. Or click on** Show Simulated Profile At End Of Link**, if you want to see the profile at the end of a lane. Or click on** Show Simulated Profile At Position On Link**, if you want to see the profile at one point on a lane.**

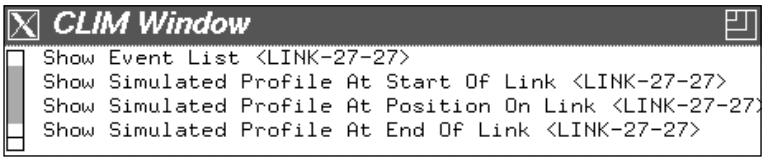
The pop up menu Simulation Objects Of **appears:**



Now you can select that link from the given list of links, which corresponds to the lane you are interested in. Click on that link name to select the link.

Click ABORT **to prevent the opening of the output window for the time-space diagram.**

If the arrow is in the simulation monitor window, select the link, which this lane belongs to. Click on the link using the right mouse button, then the following pop up menu appears:

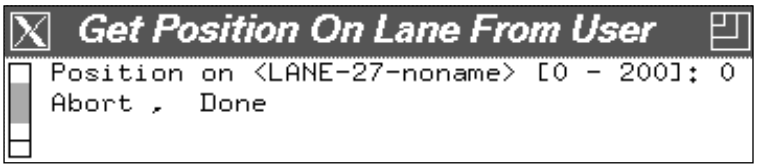


Click on Show Simulated Profile At Start Of Link**, if you want to see the profile at the start of a lane. Or click on** Show Simulated Profile At End Of Link**, if you want to see the profile at the end of a lane. Or click on** Show Simulated Profile At Position On Link**, if you want to see the profile at one point on a lane.**

3. **Select the point on the lane, in which the simulated profile should be genrated and displayed.**

If you have clicked on Show Simulated Profile At Start Of Link **or** Show Simulated Profile At End Of Link**, the position of that point is automatically the start or the end of the lane.**

If you have clicked on Show Simulated Profile At Position On Link**, the pop menu appears:**

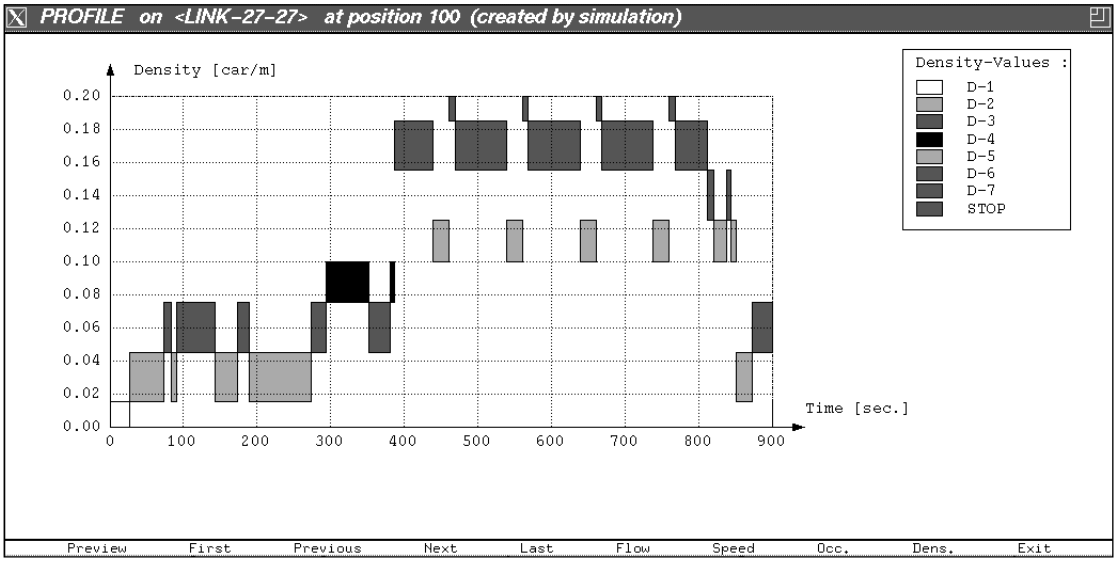


It shows the point at which the profile on the selected lane will be displayed.

Click Done **to use the given point.**

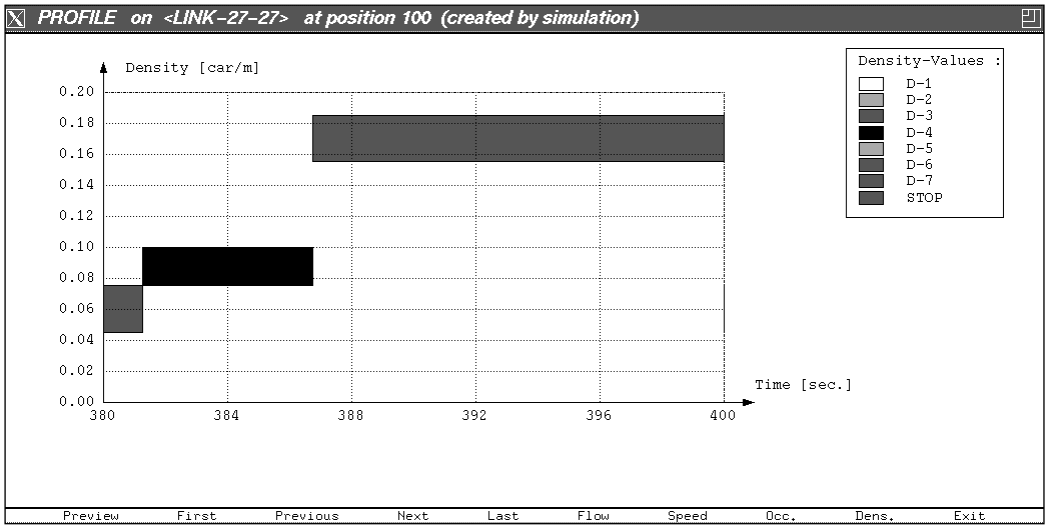
If you want to use a different point, click on the number. Now, you can edit that number. Type a new number and press Return**. Finally, click** Done **to use this value.**

4. **Now the output window** Profile on Link **appears. To see an overview of the total profile, click on** Preview **from the command line:**



This diagram shows the development of the qualitative density value in one point on the lane during the whole simulation run. It shows the intervals of traffic density attached to the qualitative density values. In a legend the patterns or colors attached to the qualitative density values are shown.

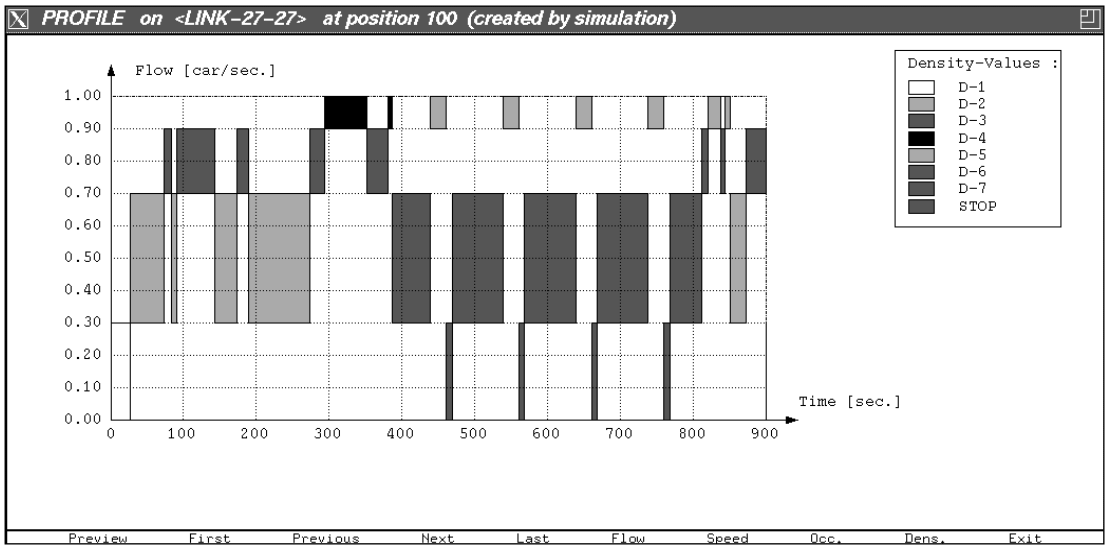
5. **If you want to view the simulated profile in detail, click on** First**,** Next**,** Previous **or** Last **from the command line:**



Clicking on one of those commands one part of the profile is shown.

Clicking on First **causes the display of the first part of the profile and clicking on** Last **causes the display of the last part of the profile. By clicking on** Next **and** Previous **you can move through the whole diagram.**

6. **If you want to see the intervals of density, speed, occupation or flow assigned to the qualitative density values, click on** Density, Speed, Occ., **or** Flow **from the command line:**



Clicking on one of those commands the same part of the profile is shown as before, but other traffic value intervals are used to describe the qualitative density values.

Click Exit **to close this output window.**

5 Auxiliary commands

5.1 Reset the current settings

If you want to change to another, maybe new network, you first have to reset your current settings, what means that you initialize the systems global variables and the display states.

INSTRUCTIONS

To reset the *SAPPORO* systems environment:

1. **Click in the main menu command** Reset All **or type in the command** Reset All **in the instruction pane.**

   The Browser pane, the Map pane and the Simulation pane will be cleared, the current settings will be initialized.

5.2 Clearing the Map

This command really is not important for your *SAPPORO* session. It will

clear and redraw the Map pane and clear the simulation pane.

INSTRUCTIONS

To clear the map:

1. **Click in the main menu command** Clear Map **or type in the command** Clear Map **in the instruction pane.**

5.3 Drawing the Map

Just the same as clearing the map. This command will clear and redraw the Map pane and clear the simulation pane. Use this command to display the current settings in the Settings pane, if they are not displayed.

INSTRUCTIONS

To draw the map:

1. **Click in the main menu command** Draw Map **or type in the command** Draw Map **in the instruction pane.**

6 The Intersection Designer IDAR

6.1 Introduction

The Intersection Designer *IDAR* (*Intersection Design and Rule System*) is a subsystem of *SAPPORO* to design traffic intersections and to build with an internal, specialized rule system signalplans for these intersections.

This chapter of the *SAPPORO*-User Manual describes the use of the first prototype of IDAR.

6.2 Starting IDAR

INSTRUCTIONS

You start the Intersection Designer with a middle-click on a node in the map-pane of the *SAPPORO*-window (cf. Chapter 1.1). A new window appears (Fig. 6.1).

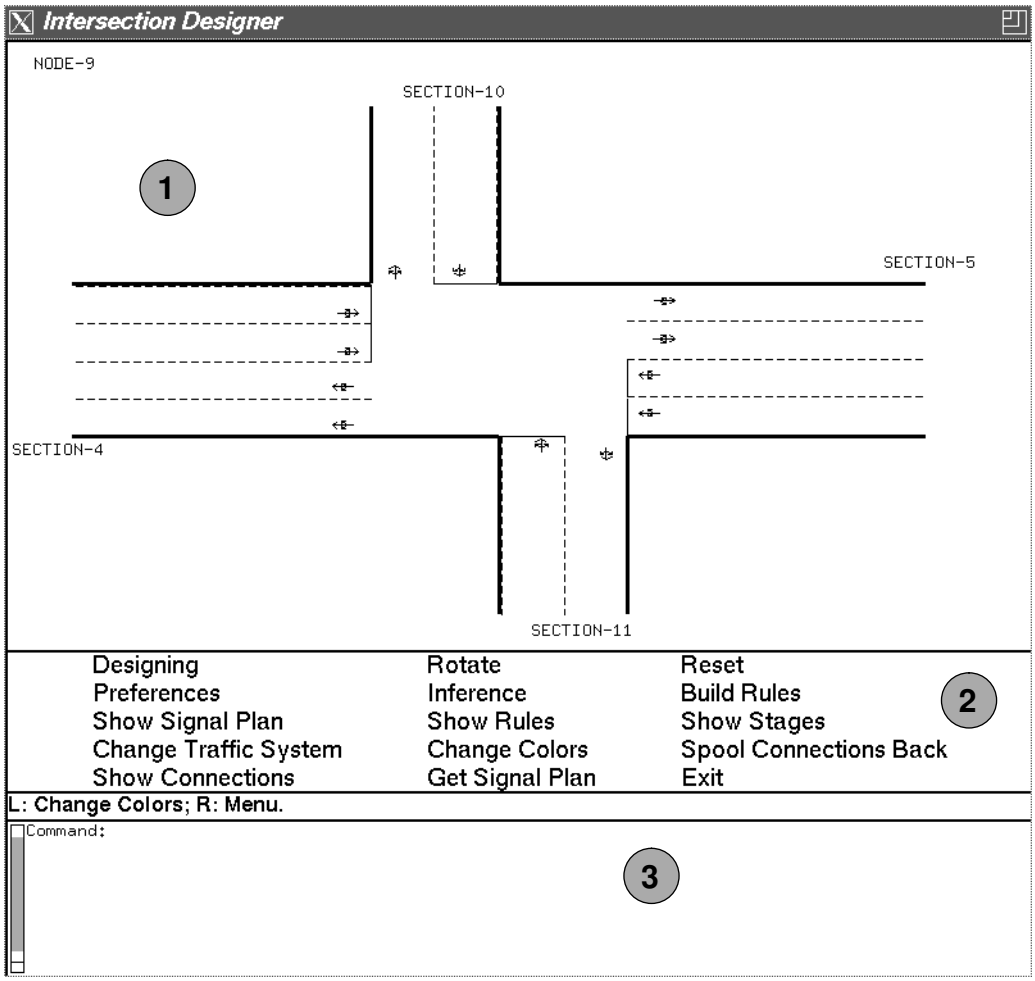


Fig. 6.1 The window appearing after you start IDAR

6.3 Description of the Panes

6.3.1 The Display Pane

The *Display Pane* (number 1 in Fig. 6.1) shows a graphical presentation of the intersection (or node). You can see the incoming and outgoing lanes of the sections arriving at the node and the names of these sections. In the upper left corner of this pane you find the name of the node in the *SAPPORO*-Network.

If you use the *Show connections* command, additionally the connections between the incoming and outgoing lanes in this node will be displayed (cf. chap. 6.4.1.2).

6.3.2 The Menu Pane

In the *Menu Pane* (number 2 in Fig. 6.1) all available commands of *IDAR* are displayed. The description of the commands you will find later in this chapter.

6.3.3 The Interaction Pane

In this pane (number 3 in Fig. 6.1) you can enter your *IDAR* commands 'by hand'. If you enter the unambiguous beginning of a *IDAR* command and press the Space key, your command will automatically be completed. Enter your written command by pressing the Return key.

6.4 The IDAR-Commands

In *IDAR* two different command types exist. The first command type will be started by clicking on a menu button of the *Menu Pane*. The second type will be started by clicking with the mouse on an object in the *Display Pane*.

6.4.1 The Commands in the Menu Pane

INSTRUCTIONS

All the commands in the menu pane will be started by clicking on their button. There are some dependencies between different commands, which must be observed by the user.

6.4.1.1 Preferences

If you click on this command a window appears. In this window some informations about the displayed intersection are shown.

First, the node-id of the displayed intersection is displayed, and the traf-

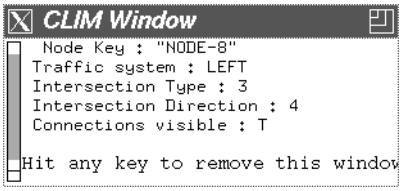


Fig. 6.2 The Preferences window.

fic system of this intersection (you can change it with the menu command *Change Traffic System* (cf. chap. 6.4.1.10)). The type (means the number of incoming sections) and the direction of the intersection are some informations about the graphical display of the intersection (If you want to see the changing of these values, try *Rotate* and choose this command again, but refer first the instructions for the command *Rotate*).

The value of "Connections visible" is T, if the connections between the lanes are shown on the screen (to change it, use *Show Connections*).

6.4.1.2 Show Connections

This command switches on and off the presentation of the connections of the node.

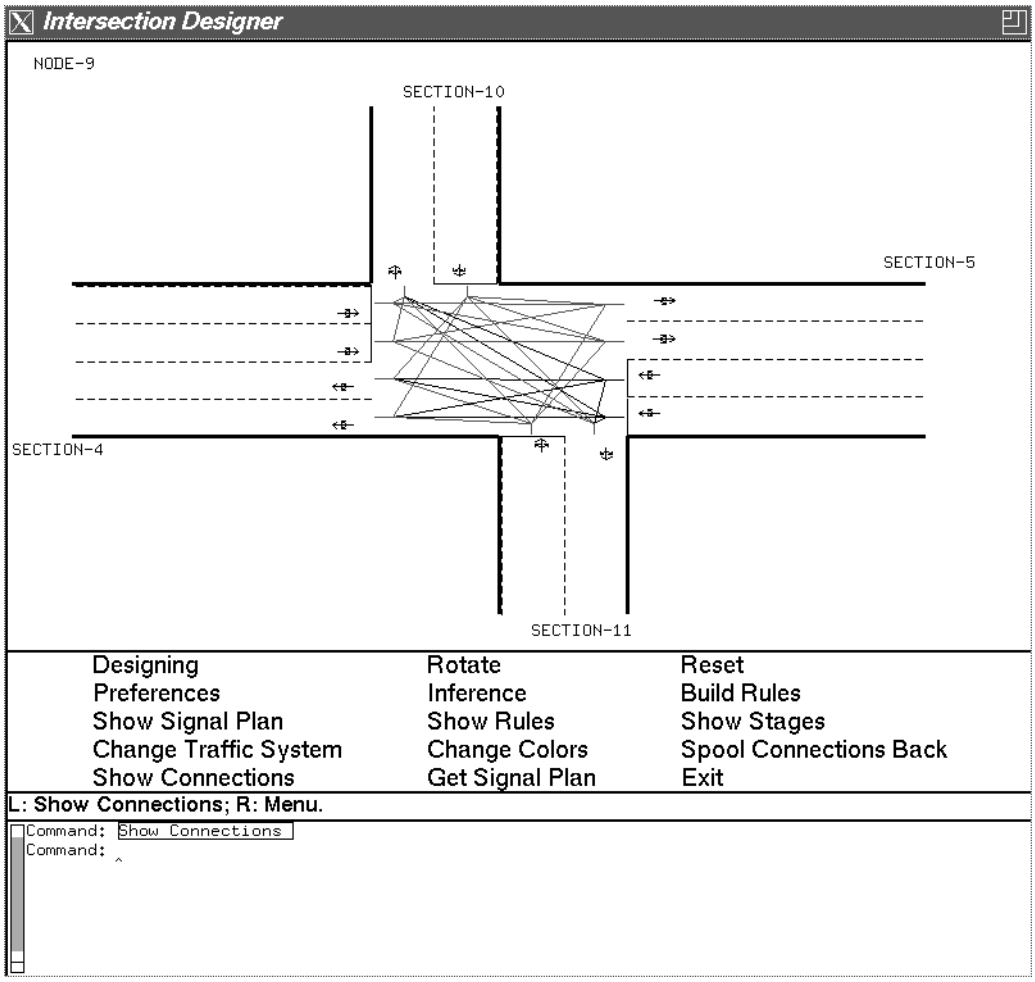


Fig. 6.3 Displaying of the connections

As an example, when you use this command with the loaded intersection from the beginning of this chapter, another representation (Fig. 6.3) appears. On a color (or gray-color) screen the connections, which start

from one section will have the same color (or gray-color).

6.4.1.3 Build Rules

This command starts the module of our system, which builds the rules especially for the loaded intersection. For syntax and semantic of the rules refer the *SAPPORO* system description.

Before you start the inference engine you have to execute this command.

6.4.1.4 Inference

If you choose this command, the inference engine starts and the signal-plan will be build.

Before you start this command, you must start the command *Build Rules*.

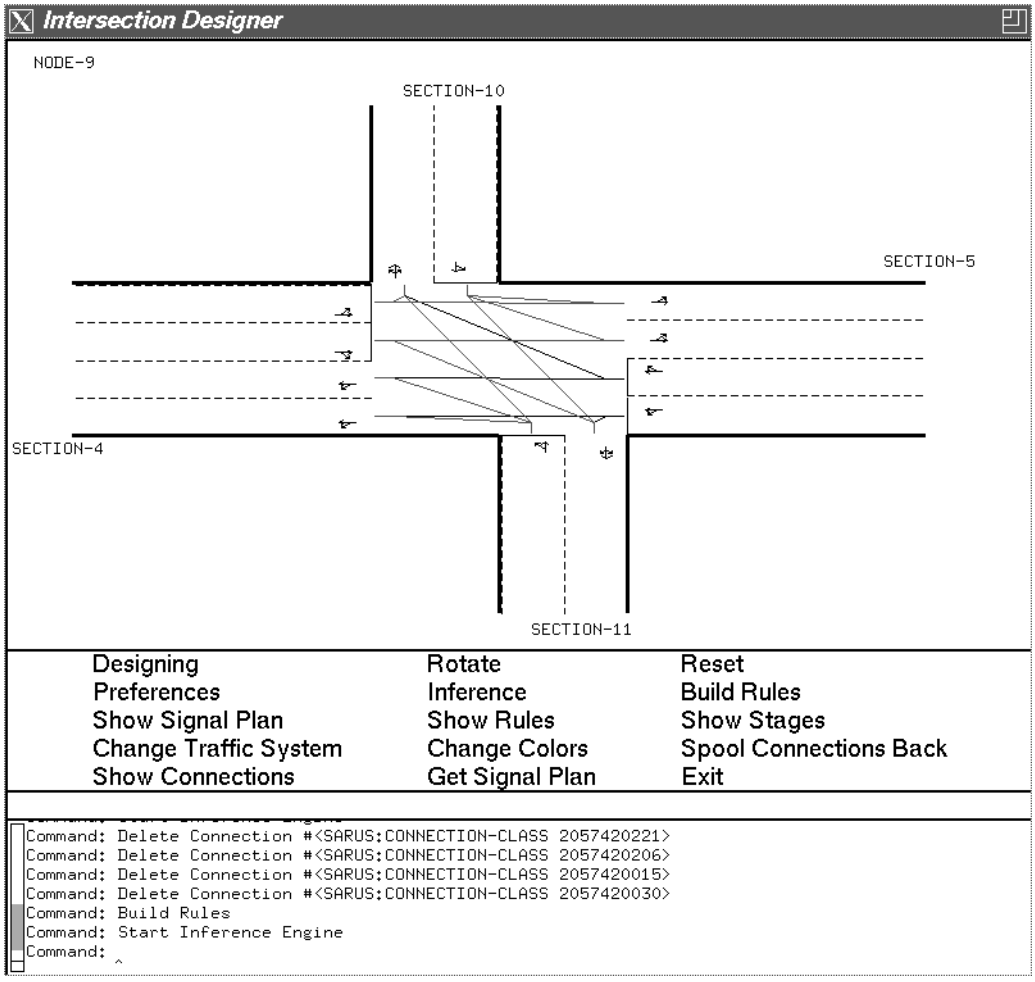


Fig. 6.4 Creating a signalplan for a modified intersection.

For a description, how the inference engine from IDAR works, refer the

SAPPORO system description.

6.4.1.5 Get Signal Plan

This command loads a signalplan. After executing this command a window appears (Fig. 6.5). You can now choose a set of stages. This set will be used in the signalplan. When you choose *Use all possible stages*, then the system uses all consistent stages it founds during the configuration process (But: when all traffic lights are red, then this is also a possible stage.). Therefore, it is better to choose *Use only the most important stages,* the system chooses then a subset of all possible stages. In this subset, every lane will have at least one time green, and the number of the stages is minimized.

You have to choose this command before you execute one of the commands *Show Signal Plan* or *Show Stages.*

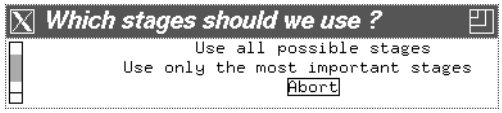


Fig. 6.5 Choosing different stage sets.

6.4.1.6 Show Rules

After building your rules and running the inference engine, you can take a look on your rules. For each lane type in each section two rules were builded by the system (refer *SAPPORO* system description): the generic and the restriction rule.

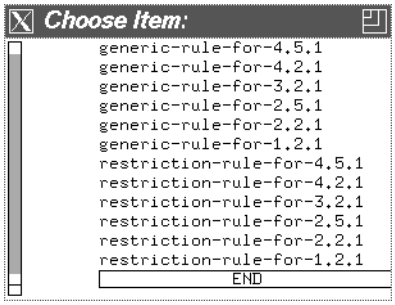


Fig. 6.6 You can choose a rule to look at it.

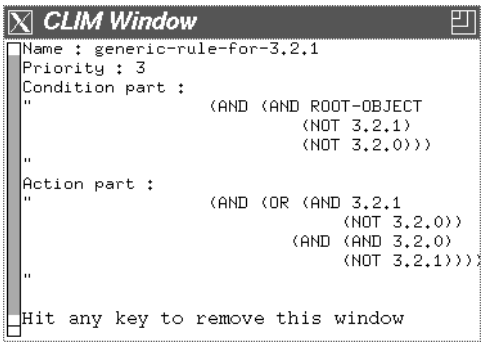


Fig. 6.7 The generic rule for the lane type 3 in the right section of the intersection (called Number 2) from the example above.

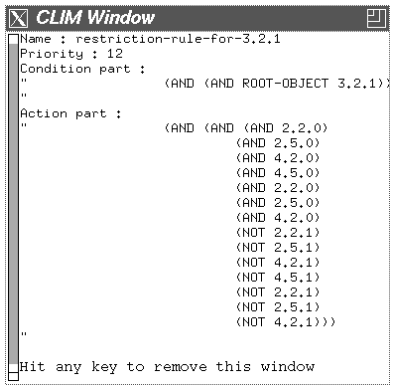


Fig. 6.8 The restriction rule for the lane type 3 in the right section of the intersection (called Number 2) from the example above.

6.4.1.7 Show Signal Plan

After loading a signalplan (via *Get Signal Plan*) you have two possibilities to look at the stages. When you choose *Show Signal Plan* you will see a graphical representation of the stages on the screen. (For the second possibility refer *Show Stages*.)

Each time you click on *show next stage* the next stage of the signalplan will be shown (Fig. 6.9). The lanes, where the traffic lights of these lanes would be green in the actual stage will be displayed with another color (mostly green) then the reds. You can choose the displaying color of these red and green lanes with the command *Change Color.*

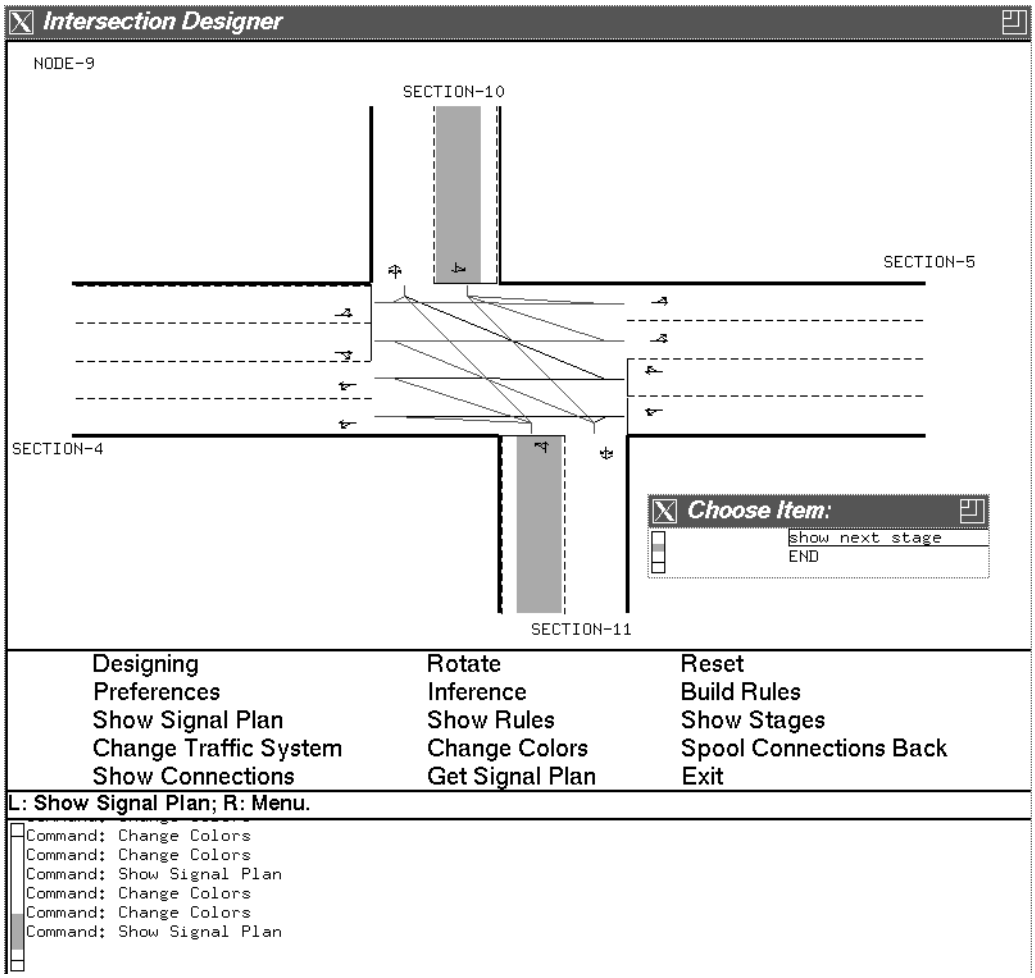


Fig. 6.9 The graphical presentation of a stage.

6.4.1.8 Show Stages

The second possibility to look at the stages is this command. A textual representation of one stage or all stages will be shown in a separate window. Every lane-to-lane connection. which is closed in this stage will be displayed. That means, that the start-lane of this connection has green in this stage. (This is a more connection-oriented look on the signalplan.)

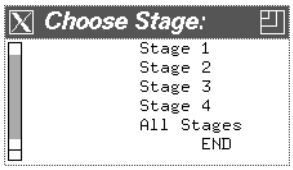


Fig. 6.10 Choose the stages you wish to see.

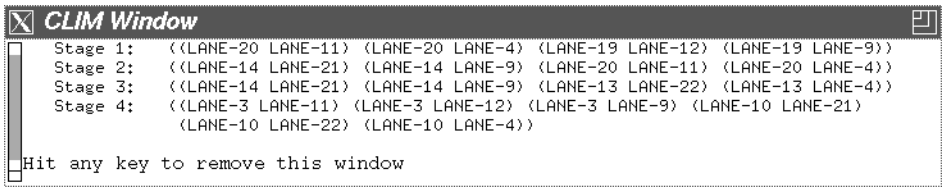


Fig. 6.11 One possibility: all stages will be shown.

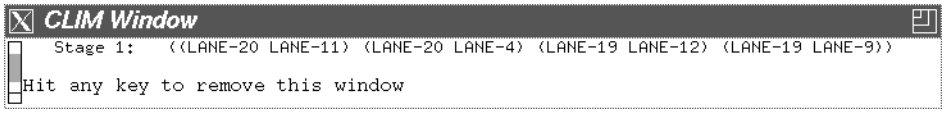


Fig. 6.12 Another Possibility: only one stage is shown. As an example: the connection between “LANE-20” and “LANE-11” will be free in this stage.

6.4.1.9 Reset

If you want an explicit redisplay of the intersection, than you choose this command. All lane types, all connections, all names of the sections and the node will be redisplayed. Sometimes, if you delete or make a connection, or if you change a color (cf. below) this command will be very helpful.

6.4.1.10 Change Traffic System

Our system works both for the left (Japan, Great Britain,...) and the right traffic system (Germany, USA,...). If you choose this command the actual intersection will be transformed in the other system.

You can now test the system (execute *Build Rules*, *Start Inference Engine*,...), but don’t save the connections back to the network (!!!), this will cause some fatal errors. The default value momently used in *IDAR* is LEFT. according to the network of the city of Sapporo.

6.4.1.11 Rotate

You can rotate the sections of the intersection. If you choose this com-

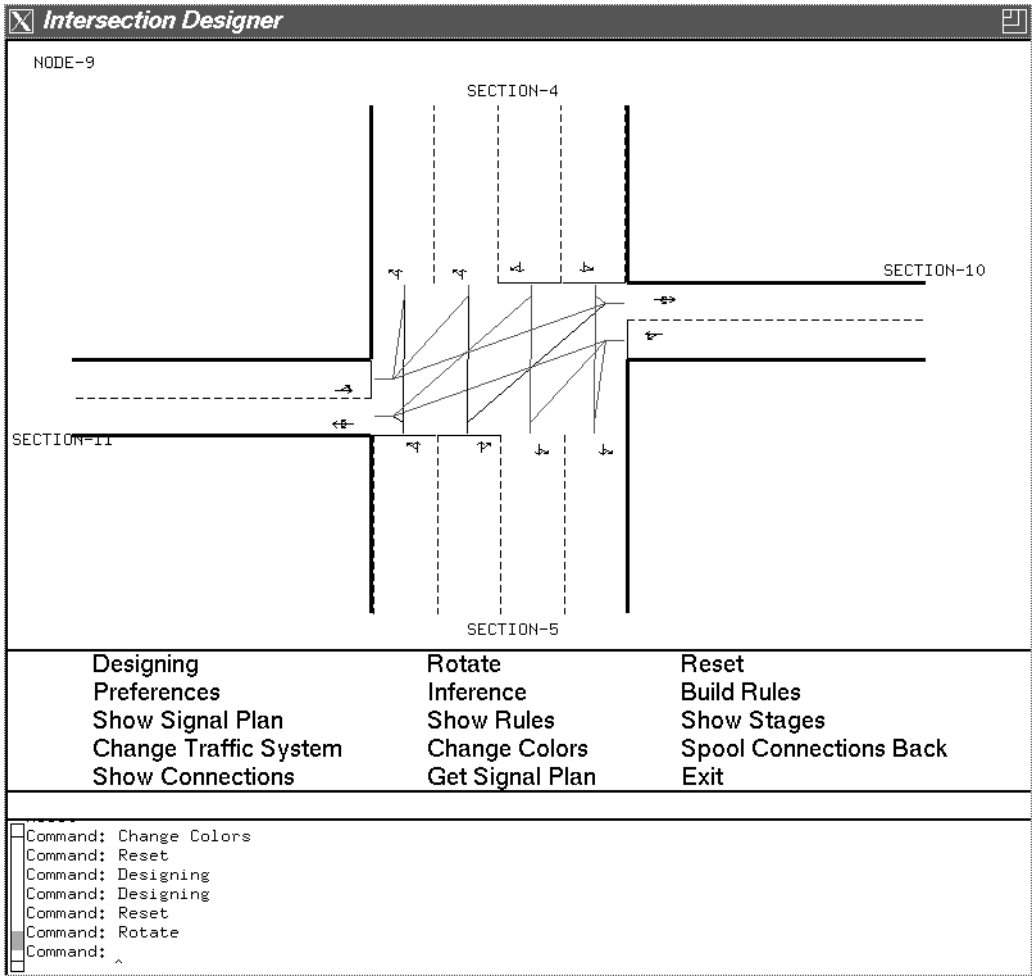


Fig. 6.13 The example above after executing *Rotate*.

mand the intersection will be turned by 90 degree (Fig. 6.13).

You can now test the system (execute *Build Rules*, *Start Inference Engine*,...), but don't save the connections back to the network (!!!), this will cause some fatal errors.

6.4.1.12 Designing

For future applications of *IDAR,* we implemented the command *Design-*

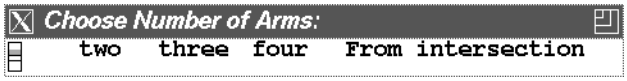


Fig. 6.14 Menu window to design own intersections.

***ing*. You can design your own intersection-types. After executing this command, the window shown in Fig. 6.14 appears. Choose now the number of arms (or sections) your intersection should have. (If you want to exit, choose *From Intersection*.) Then you will be asked in the interaction pane after the number of incoming and outgoing lanes of each section. After you entered them, the graphical representation of your intersection will be shown in the display pane. You can enter now some connections (cf. below) and use the system. But, be careful, don't choose *Spool Connections Back*, this would cause some trouble.**

6.4.1.13 Change Colors

As described above you can change the displaying color of the red and green lanes in the graphical presentation of the loaded signalplan. After choosing the command *Change Colors* the window Fig. 6.15 appears.

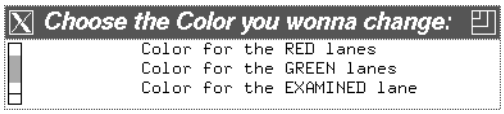


Fig. 6.15 Menu to change the colors of the lanes.

After selecting the color, you want to change, the window in Fig. 6.16 will be displayed, in which you can choose your favorite color (or gray-color or pattern). (For the meaning of *examined lane* refer chap. 6.4.2.4.)

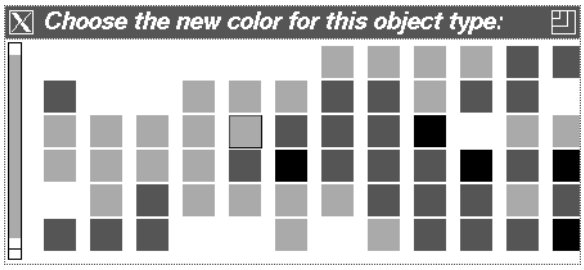


Fig. 6.16 Click simply on the color you want to use.

6.4.1.14 Spool Connection Back

After deleting a connection or initiating a new one, you can spool back the connections of the node with this command. The changed connections will be promptly used in the network. If you don't want that the changed connections will be saved for the future use of the loaded network of *SAPPORO*, simply don't choose *Save Network* in *SAPPORO* before you exit *SAPPORO*.

6.4.1.15 Exit

Exits *IDAR* without spooling back the eventually changed connections to the network.

6.4.2 The Mouse-Clicks commands

6.4.2.1 Get information about an object

To get information about an object simply click middle on this object.

6.4.2.2 Delete an object.

To delete an object, click left on this object. (Yet only implemented for connections).

As an example, refer Fig. 6.3 (the original intersection) and Fig. 6.4 (the same intersection after deleting some connections).

6.4.2.3 Change Color of an Object

To change the color of an object (or an object family), simply press control and middle-click on this object. The window from Fig. 6.16 will appear.

You can change the colors of the section names, the node-name, the connections (the color will be changed for all connections starting at the

same section), and via *Change Color* of the lanes.

6.4.2.4 Showing the “free” and inconsistent lanes for one lane

After building the rules and running the inference engine, the system shows you for an examined lane its “free” and the inconsistent lanes. (The “free” lanes are the lanes, which could be green at the same time when the examined lane is green.) You can choose via *Change Color* the colors for this presentation.

The system switches to this presentation, when hold down “Meta” and middle-click on a lane. (Press this combination for a while.)

6.4.2.5 Enter a new connection

To enter a new connection in the intersection, just click with the right mouse button on a lane (incoming or outgoing). Then you have to choose the second lane, at which the connection ends. Just click left on this lane (outgoing or incoming). After this you will be asked in the *Interaction Pane* after the turning ratio of this connection.

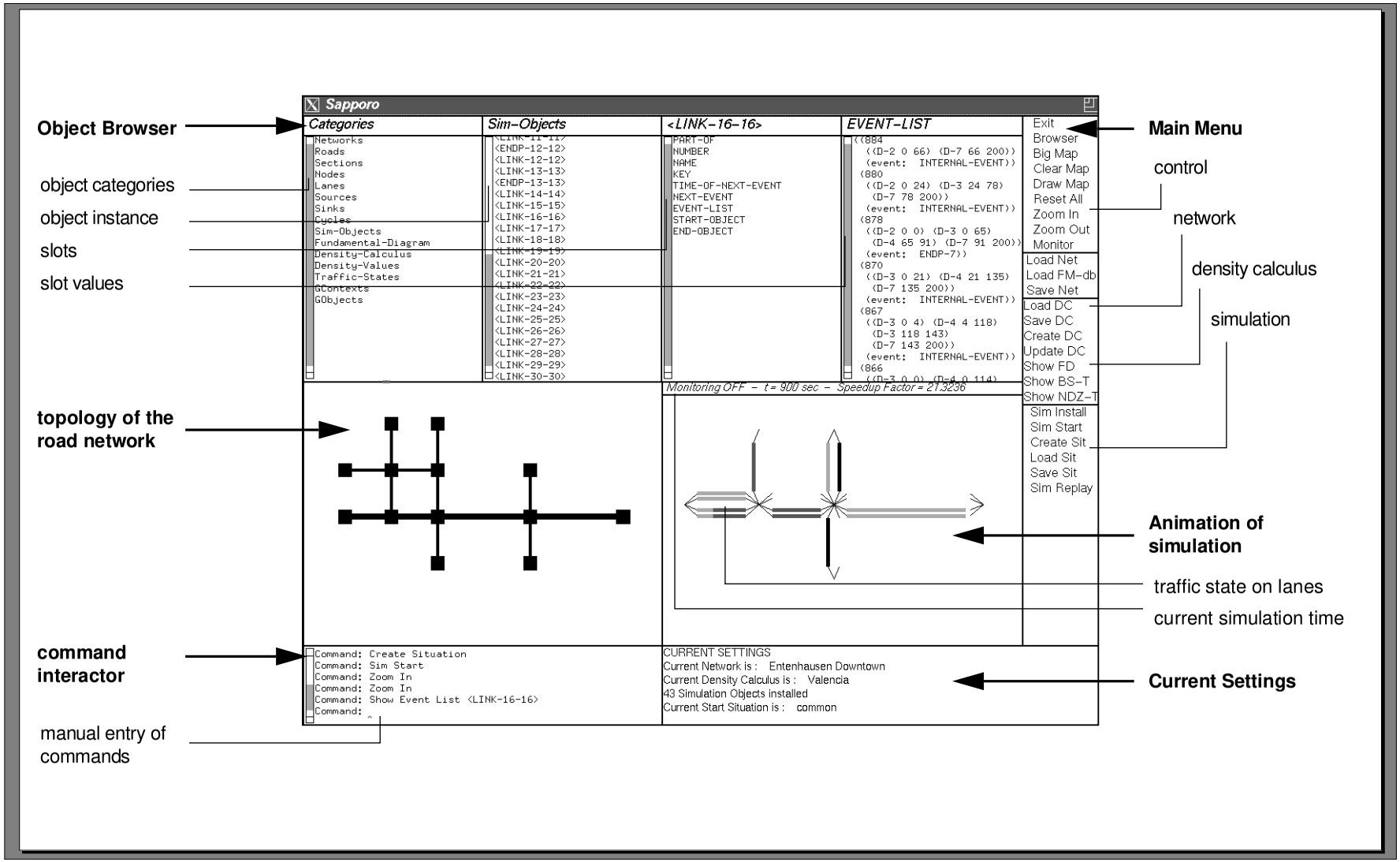
When you choose twotimes an incoming (or outgoing) lane then no connection will be initiated. To save the connections into the network execute *Spool Commections Back*.

Summary of the graphical interfaces

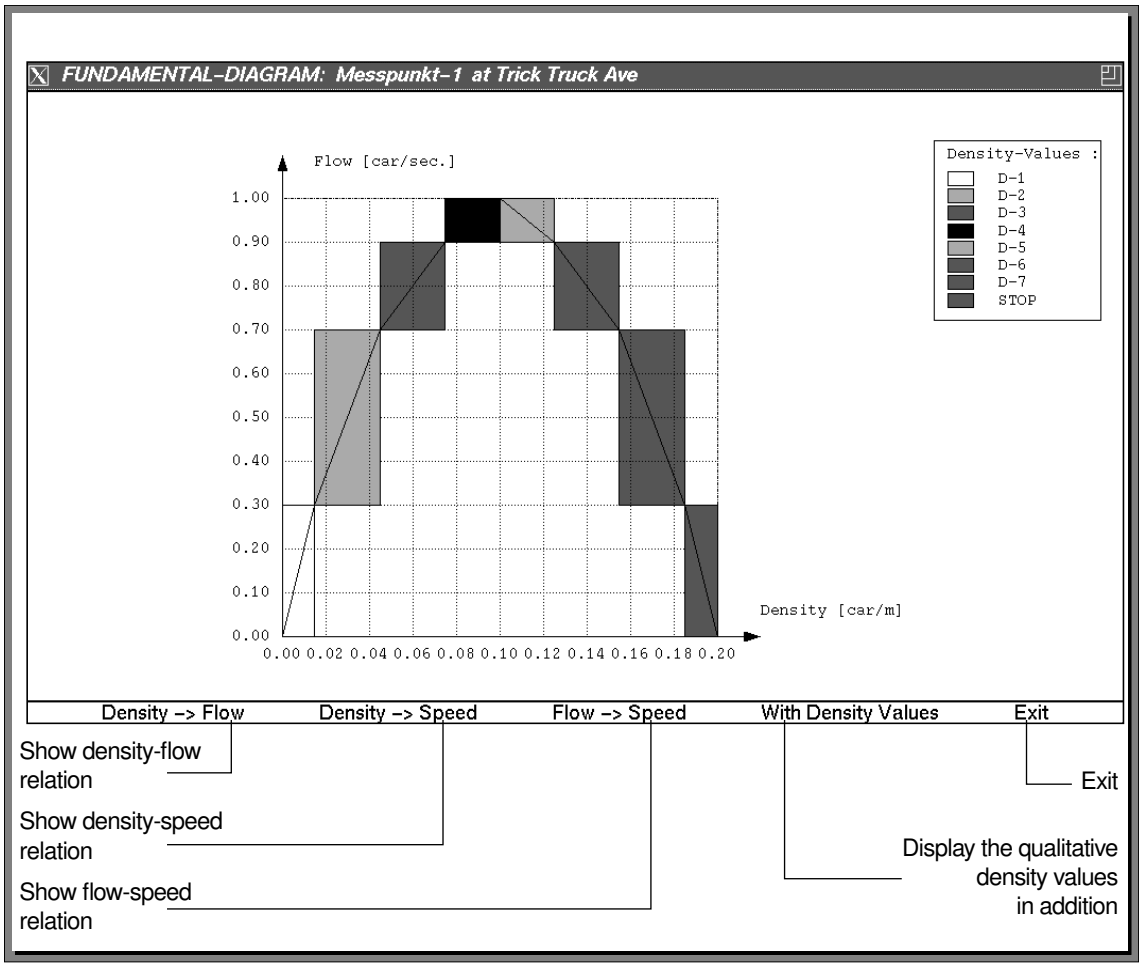
About the user interfaces of *SAPPORO*

In *SAPPORO* the user interface divides into one main interface and several secondary interfaces. The following overview gives you a short description of the interfaces. The commands which you can click on in each interface are mentioned, too.

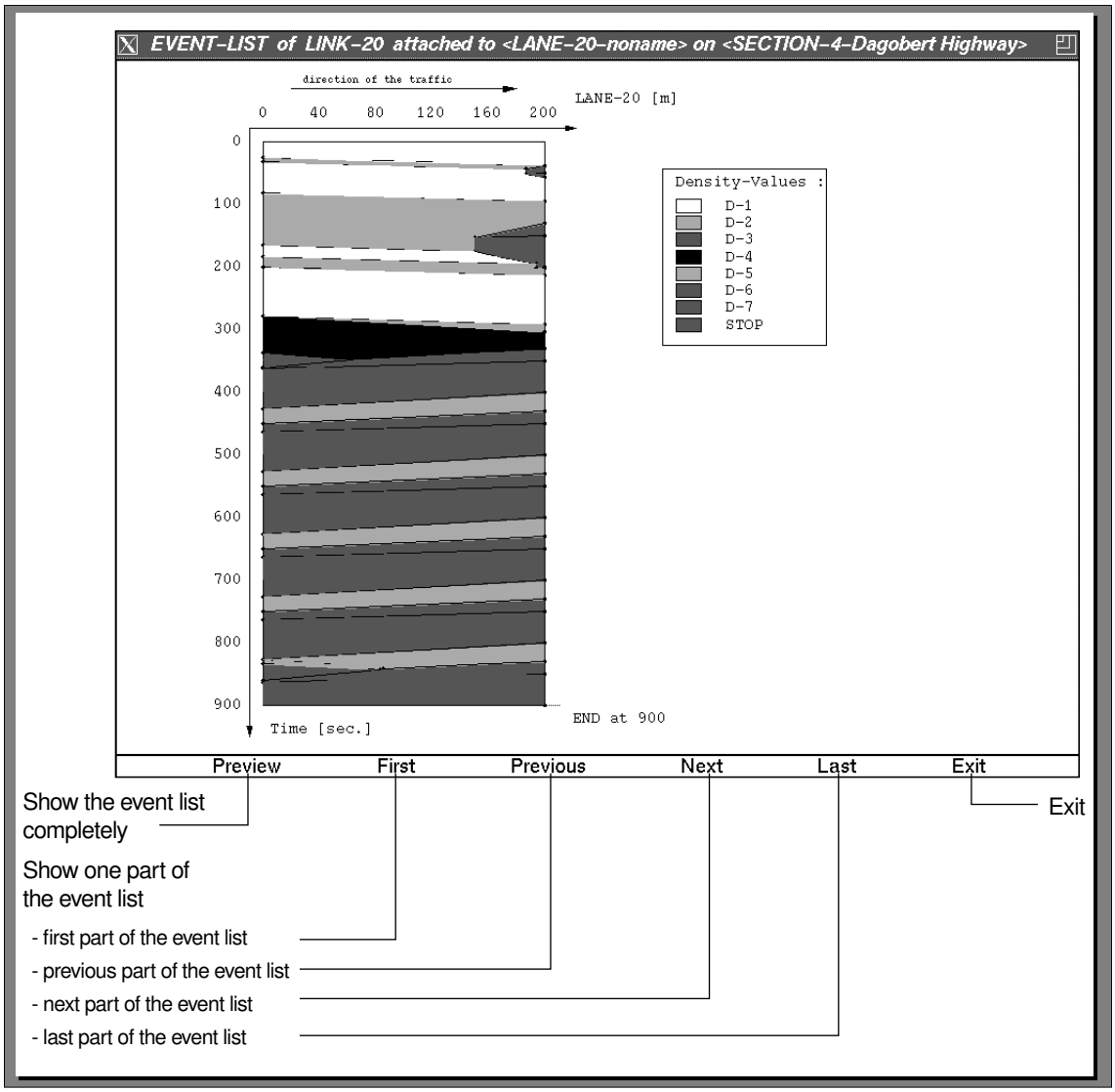
A detailed description of the commands used in the main interface or in the secondary interfaces you will find in that chapter of this manual concerning the main menu command or the content of the secondary interface.



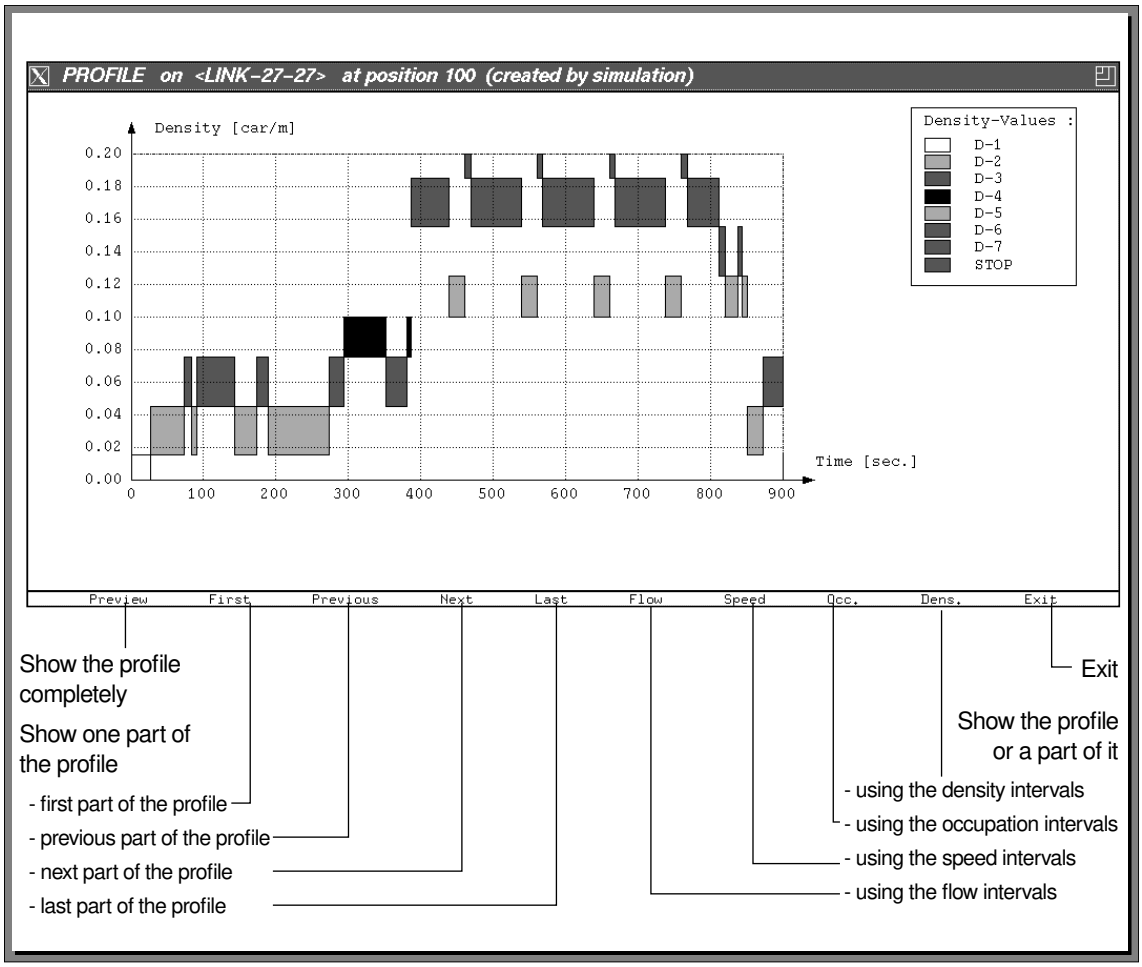
Main Interface of ***SAPPORO***



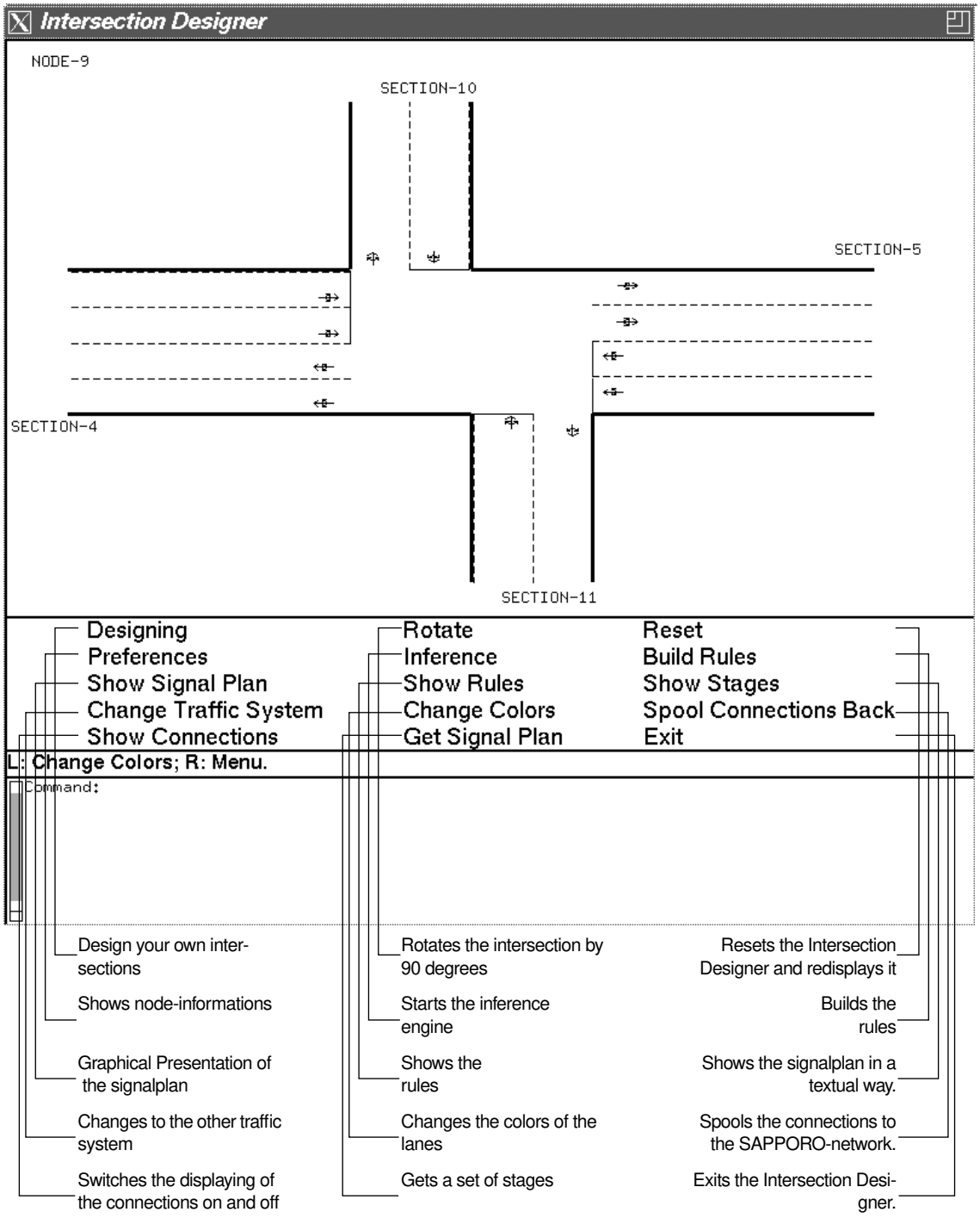
Secondary Interface: Display of the fundamental diagram used to create the density calculus



Secondary Interface: Display of the simulated event list of a lane



Secondary Interface: Display of the profile created for a point on a lane



Secondary Interface: The Intersection Designer IDAR

Command Index