Tutorial

The Basic Elements of NEPLAN

To understand the NEPLAN environment, it is essential that certain concepts used in the system are described:

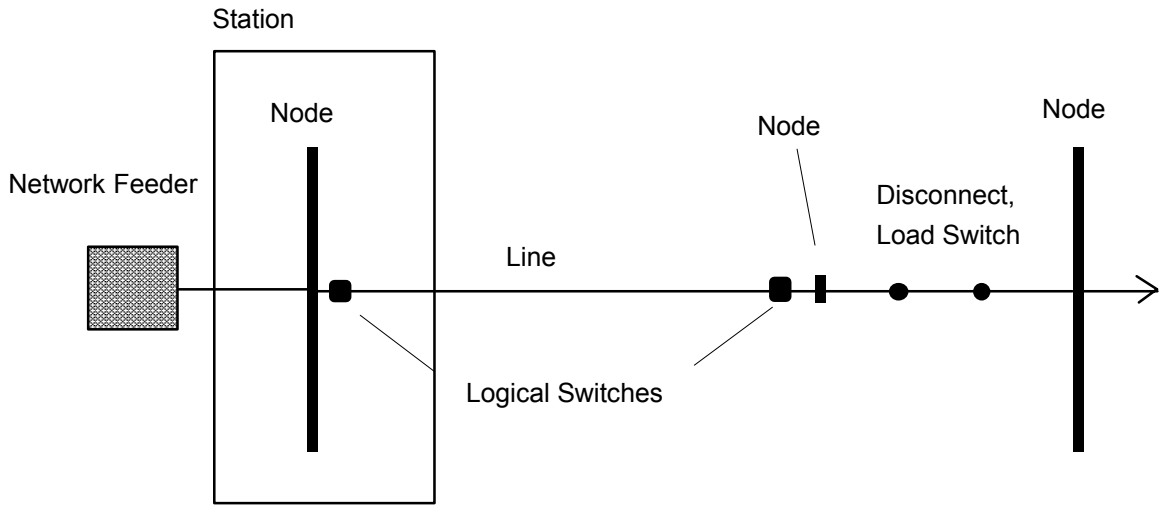


Fig. 2.1 One line diagram with network components

An electrical power system consists of nodes and elements.

Nodes

A node is the connection point of two elements or a location, where electrical energy will be produced or consumed (generator, load). A node is described by its

- name (identification with 17 characters),
- nominal system voltage in kV,
- description (31 characters),
- network group or area,
- type of node (main bus bar, bus bar, sleeve, special node),
- association with a station.

The nominal system voltage Un is the line-to-line voltage, for which a power system is designated and on which several characteristics of the power system has been referred. In NEPLAN the nominal system voltage of the nodes must be

entered during the node data input. Every voltage is given as a line-to-line voltage (delta voltage).

Elements

An element corresponds to a network component, like e.g. cable, transformer or electrical machine. There are active elements and passive elements. An element is described topological by a starting and an ending node. For three windings transformers a third node must be given. The elements will be described electrical by

- the rated current, rated power and rated voltage and
- its parameters, such as losses, reactances, ...

In NEPLAN these parameters are entered with the help of input forms.

The **active** elements are network feeders, asynchronous machines, synchronous machines and power station units. A network feeder represents a neighboring network.

The **passive** elements are lines, couplings, switches, reactors, two and three windings transformers, shunts and loads. The loads can also be entered along a line without entering nodes (line loads).

Modeling of Active Elements

For a short circuit calculation the active elements are modeled with the help of their sub-transient reactances.

For a load flow calculation these elements will be represented by resistive and reactive powers **(PQ-nodes)** or by voltage magnitude and angle (**slack nodes**) at the node. The network feeder usually will be modeled as a slack node.

Protection Devices, Current and Voltage Transformers

Protection devices (overcurrent relays, distance protection relays, circuit breakers) and current or voltage transformers are associated with the built-in node and the switching element. They have no influence on the load flow and short circuit calculation. Only their limits are checked during the calculation. These elements are used in the relay coordination modules.

Field

A field can be assigned to every node/element connection. If there were protection devices at the node and in the element the field name will be associated with

the devices. A field has no meaning for the calculations or for protection device coordination and will only be used in relation to the data base.

Station

A station can contain several nodes and has no meaning for the calculations or for protection device coordination. It will only be used in relation to the data base.

General Element

NEPLAN provides a general element type, which can be used for documentation and information purposes. These elements will not be used for the calculation. New general element types can be defined with the symbol editor. For each element type it is possible to assign a SQL database table (requires the optional NEPLAN SQL database driver module). The fields of this SQL table can be defined by the user.

Symbol

Each element has a standard symbol. If you like to have a different symbol, you can select a symbol from the library before adding a new element to the network diagram. A symbol library is included in the NEPLAN package. You can add your own symbols with the Symbol Editor (see chapter "Symbol Editor"). It is also possible to change the symbol as well as the angle of the element later (see "Element - Symbols" in chapter "Menu Options").

Switches

In NEPLAN the switches are used to change the network topology (switching on/off elements). There are two different types of switches:

- physical switch and
- logical switch.

Physical switches are couplings, circuit breakers and disconnect or load switches. They can be entered in two ways: either they are entered as an element with a starting and an ending node or the are entered as protection devices, which are topological defined with the element to protect and one node of the element.

Logical switches are fictive switches, which are assigned to all elements (see section "Elements" on page 2-2) by the system. A line, for example, has two logical switches, one at the starting and one at the ending node. A physical switch has no logical switch, because it will already be switchable.

During the input of a network, the physical switches can be neglected, because switching can be done with the help of the logical switches. These has a disadvantage, when a line leads to a double bus bar system. Switching from one bus

bar to an other, the user has to change the starting or the ending node of the line. If the user enters two disconnect switches (one to each bus bar) with an additional node in between, the switching can be done with the disconnect switches. The physical switches can be reduced during the calculation (see "Disconnect Switch Data" in chapter "Element Data Input and Models").

Network Group

A network group is a network area. The network is defined in the node data mask. Each node with the same group name belongs to the same network group or area. Each area can be drawn in a different color.

Partial Networks

Unlike network groups, a partial network is an independent network. A partial network has no connections to any other networks. You can make partial networks by opening logical or physical switches. It is possible to color each partial network differently (Fig. 2.2).

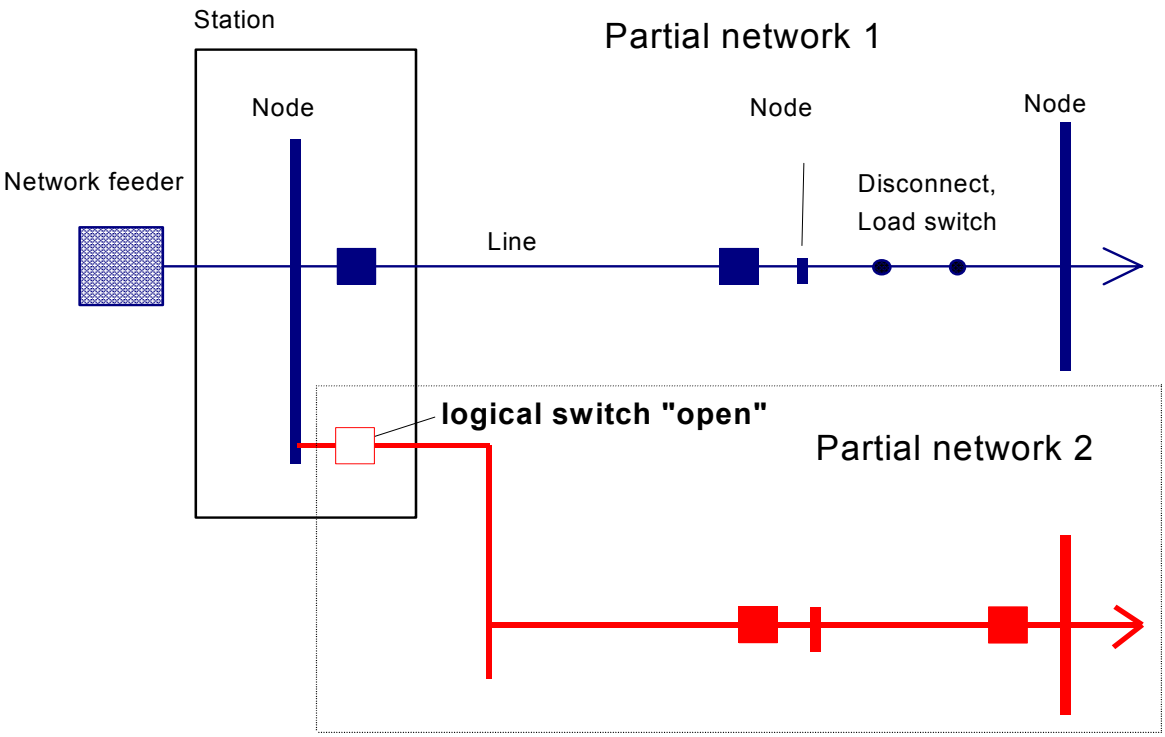


Fig. 2.2 Partial networks

Graphic Layer

To each node or element a graphical layer will be assigned. Before you insert a new node or element graphically, you can choose the graphical layer. The graphi-

cal layers can be displayed selectively. For example, it's possible to use different layers for current transformers and relays. If you are doing load flow calculation, you could switch off the layer for the relays. If your doing relay coordination you can switch on again the relay layer. The graphical layer of each element is displayed in the title oft the data form of the element. **Please note the difference between the graphical layers and the network layers** (see "Network Layer Technique" on page 2-15). Each graphic layer can be colored differently. If a DXF file will be imported, NEPLAN takes all layers which have been defined in this DXF graphic. It is possible to color DXF layers and NEPLAN graphic layers differently.

Saving the Network Data

The topological, the electrical and the graphical data of the elements, protection devices, current transformers, etc. as well as the load data are saved in the project file.

The project file is an ASCII file with the extension .MCB. The previous version of the project file will be saved in a file with the extension .MC_

The load data or the so called node data can be additionally saved in the node data file. These are the active or reactive power of the loads or generators as well as the voltages of the slack nodes (usually network feeder). The file extension is .NDB.

The topology can also be saved in a file. The state of all logical and physical switches as well as the taps of tap changing transformers are saved. The file extension is .ZDB.

Automatically Saving the Data

If this option is active (standard setting) the project will be automatically saved before each calculation. The network data will be saved in the file **tmp_proj.mcb**. After a computer breakdown the user can immediately read this file and save it under the valid project name.

Note! After computer breakdown no other project must be opened, otherwise the temporary file will be overwritten!

Thus the file name **tmp_proj.mcb** is reserved by NEPLAN for internal use.

Variant Management System

For calculating different cases (variants), NEPLAN has the possibility to separate and save the loads, the topology and the single line diagram from the project data.

NEPLAN is also able to manage all differences form a base network system, this means NEPLAN can notice all changes which have been done to a network (e.g. insert new elements, change an element, delete an element). These differences can be saved in a different file.

As soon as the base case or root network (*.MCB file) has been loaded, variants can be defined. The variant can consist of the following data:

- loads (node data), feeders (*.NDB file)
- topology data (*.ZDB file)
- graphic (single line diagram) data (*.NGR file)
- Network differences (*.VEL files)

In the project files (*.MCB file) all network data (incl. loading, topology, graphic data) of the base case system or root network are saved.

If the user changes data of the root network (e.g. deleting, inserting, changing nodes or elements) NEPLAN recognizes the changes. It is now possible to save only these changes or differences into a separate file without saving the whole project into a new project file (*.MCB file). These changes are saved into a differences (variant) file (*.VEL file). If the network will be changed interactively after having loaded this differences file (e.g. VAR-1), the user can save the changes into a new (sub-) differences file (e.g. VAR-12). In this way the user can built up a whole differences tree. If a sub-differences file (e.g. VAR-43) is loaded, all "parents" differences files (e.g. VAR-4 and VAR-42) will be loaded first. If the root network or a differences file will be changed, all sub-variants are automatically changed too.

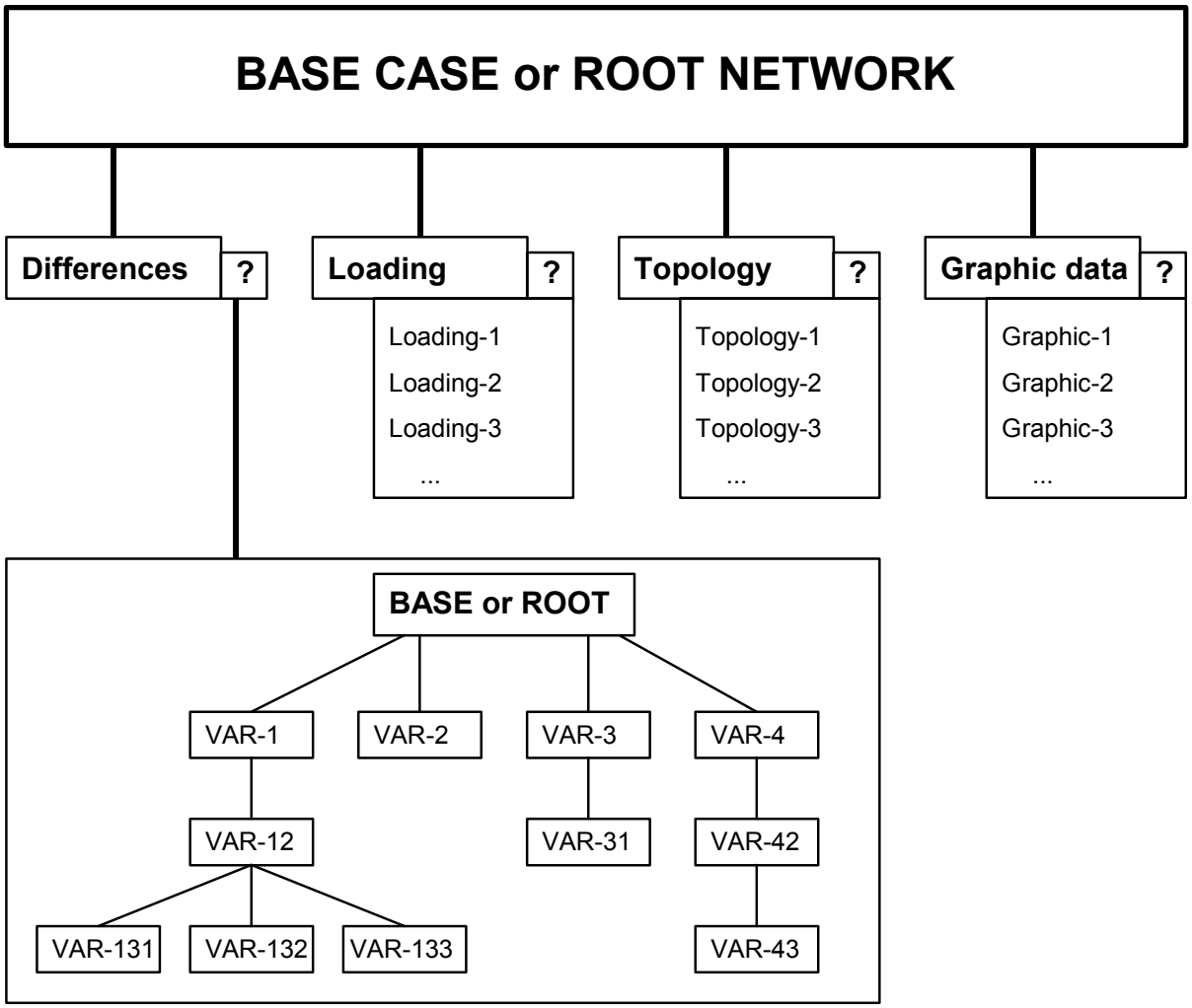


Fig. 2.3 Variant Management System with NEPLAN

The following functions are available for the management of variants:

- save of the changes into a sub-level differences file
- save of the changes and the last loaded differences file (saving a differences file on the same level as the loaded one in the variant tree)
- save of all sub differences files to a root net differences file (a branch of the variant tree can be joined to a single differences file)
- coloring the interactive network data changes
- coloring the present changes in the differences file
- coloring all changes in the differences files referred to the root network

Before saving data into a differences file, it is recommended to color the changed data (see "Drawing-Parameters" in chapter "Menu Options").

Additionally to the differences file a loading and/or a topology file can be read in. It is also possible to have several graphic files for one network. In this way the user

can create a simplified diagram with a few interesting nodes and elements in order to document calculation results. It is also possible to have a geographic and a schematic graphic of the network for the same network database.

Each variant can be reproduced by loading the corresponding files.

Important functions are available for generating project files (new base cases or root networks), such as

- delete partial networks
- delete exclusive partial network
- join project files

The user has the possibility to define and to describe variants, which can consist of a base case file (project file), differences file, topology file, load file and graphic file. These variants are saved on the project file (the base case file). An arbitrary variant can be selected from the predefined variants. All files (base case file, differences file, topology file, load file and graphic file), which belongs to the selected variant, are sequentially opened.

Organization of Project Data

The organization of the network data and of the projects will be done

- with the file name and
- the name of directory.

During installation of the systems the following directories are created:

- C:\NEPLAN\DAT (data directory)
- C:\NEPLAN\LIB (libraries directory)
- C:\NEPLAN\USER (user directory for EMAIN.PAR, result and text files)
- C:\NEPLAN\TEMP (for temporary files)

In the first directory all examples are saved, in the second all libraries.

If there are several variations assigned to one project, it is reasonable to save the files into one directory, such as C:\NEPLAN\DAT\BASE or C:\NEPLAN\DAT\PROJECT1.

Usually the user will work with only one directory for libraries. The directories are defined in the NEPLAN.INI file in the Windows directory.

Interfaces to NEPLAN

NEPLAN has several interfaces to external applications:

- Import/Export through ASCII file
- Export to data base
- Result data base
- Clipboard
- DXF-files
- Raster-Graphics (e.g. BMP, PCX, TIFF, etc.)
- DVG-Format (Format der Deutschen Verbundgesellschaft)

Import/Export

The import and export of topological and electrical data of the elements can be done with the help of ASCII files. The ASCII file can be an ELD- or an EDT-file. ELD-files are for the former NEPLAN-DOS version. The data fields are separated by comma.

EDT-files are the real import/export files for external programs, such as MS-Excel. The data fields are separated by semicolon.

The import and export of loading data and topology data can also be done with the help of ASCII files. The loading data can be imported/exported with the KND- or NDT-files. KND-files are for the former NEPLAN-DOS version. The data fields are separated by comma. NDT-files are the real import/export files for external programs, such as MS-Excel. The data fields are separated by semicolon.

The extension of the topology file is *.ELT.

The file structure of the import/export files are given in the appendix (see "Appendix").

Former NEPLAN-DOS graphic files with extension .SLD can also be imported.

If data is imported without graphic, then it is possible to generate the graphic of the network automatically by the NEPLAN "Auto-Layout" function.

The data of fuses, overcurrent-time relays and circuit breakers can also be exported into an ASCII file (protection device files *.SDT). These files can be imported in the module Selectivity Analysis.

Export to SQL Database

The data of a project file can be exported into a SQL database, if the database mode is active. The SQL database structure will be generated automatically. The whole network with the graphic will be saved into corresponding tables. The database can be MS-Access, Oracle, etc.

In the database mode all data are saved directly into the database.

Result SQL Database

After a calculation the results can be saved into a result file (optional). Depending of the calculation the result file will get an other extension, for example .RSC for short circuit result files and .RLF for load flow result files. The file is an ASCII file, which fields are separated by a semicolon. Result files can be read by MS-Excel. In this case the user has the possibility to evaluate the results in an arbitrary way.

If the SQL database driver is loaded, the calculation results can be saved in a SQL database table.

Clipboard

The diagram can be exported onto the clipboard. The clipboard data can be imported by an external program, such as a word processing program.

DXF-Files

DXF-files can be imported. All layers are identified and displayed. The user can select the layers to be imported from a list. The imported drawing can be additionally scaled. The imported layers are managed by the program in different graphic layers, The imported drawing can be changed.

Cadaster and Raster-Graphics Files (BMP, PCX, TIFF)

Raster graphics files (BMP, PCX, TIFF, etc.) can be imported in any layer. It is possible to import a raster graphic (e.g. PCX) as a cadaster. The cadaster can be used as background for the NEPLAN network data. The cadaster can be calibrated to use real world coordinates.

SQL Database Connectivity

Instead of working with a project file (*.MCB file), it is possible to connect NEPLAN to a relational SQL database like MS-Access or Oracle.

The user can activate the SQL database mode before opening a project file. The database mode is active when the database driver has been loaded before. All data changes are online, that means if data are changed in the program, the corresponding record in the SQL database will also be changed immediately. Therefore, the menu item "File - Save" is not activated when working with a database. Functions for changing and requesting database fields are available. User defined fields (e.g. "MANUFACTURER"), which are not used for the calculation can be added in the SQL database. All fields (NEPLAN and user defined fields) can be displayed on the single line diagram. Forms with the SQL data fields can defined by the user and can be displayed and edited with a mouse click on the NEPLAN objects (e.g. lines).

The Screen Structure

The screen consists of the following elements:

- window title,
- menu bar,
- toolbar,
- status bar,
- vertical scroll bar (at left),
- horizontal scroll bar (at bottom),
- drawing area.

In the window title the application name and the current project are displayed. The menu bar is for selecting the options. The scroll bars are for changing the view of the one line diagram (see "Changing the View of the Diagram" on page 2-14). In the drawing area the network will be entered or changed.

Status Bar

The status bar shows the state of the system. The following is shown:

Status:	Program mode.
Layer:	Network Layer.
Graphic Layer:	Graphical Layer.
Zoom:	Zoom Level in %.
x, y:	Mouse Position. The xy-display can be switched on and off.

The settings for "Ortho" and "Autoroute" can be changed on the toolbar with a mouse click.

Program Mode

The system works with different program modes (states). In the status bar the current program mode is shown. The input mode, e.g. "Input lines" allows the user to enter lines. In the delete mode elements can be deleted. The current mode is active until an other program mode has been chosen. The choice is made in the menu bar or in the toolbar.

Toolbar

The following menu items are also available in the toolbar:

- Create a new project file
- Open a existing Project file
- Save the project file
- Print
- Change the view from normal to layout
- Project - Info
- Insert new nodes
- Insert new lines
- Delete
- Undo
- Move and move group (note both menu items are in the same function)
- Redraw and neutral program state
- Show data masks
- Zoom all
- Zoom in
- Zoom out
- List editor active (L-Edit on/off), The data can be entered list-oriented (not recommended) instead of graphically
- Insert elements orthogonal (Ortho on/off)
- Insert element with auto routing (Auto on/off)
- Display xy-coordinates in the status bar
- Display the grid

Keyboard

The following short-cuts are available in NEPLAN:

<PgUp>	move the drawing are one page up
<PgDn>	move the drawing are one page down
<Home>	move the drawing are one page right
<End>	move the drawing are one page left
<cursor keys>	move the drawing area
<Ctrl + Insert>	save string in buffer
<Shift + Insert>	paste string
<Del>	delete selected elements
<F1>	activate the help file
<F9>	redraw

Entering the Network

The network, that means the nodes, elements, protection devices, stations, etc. can be entered in two different manners:

- list-oriented
- graphic-oriented

The **list-oriented** input allows the user to enter the element's data with the help of lists (list editor). The list editor makes available functions for insert, delete and edit. No graphical data (diagram data) are needed.

The **graphic-oriented** input allows the user to enter the network graphically with the help of the mouse. After having positioned an element in the diagram, the mask for entering the electrical element's data appears.

A combination of both input modes is possible. In the list editor there is a function to insert an element graphically.

Default values for nodes and lines can be entered for the graphic- and list-oriented input.

List-oriented input can be useful if you have already some network data in a MS-Excel format before starting with NEPLAN and you want to import these data into NEPLAN. The list editor allows you to add the one line diagram later.

If you do not have any data before starting work with NEPLAN **use the graphic-oriented, NOT** the list-oriented approach.

Graphical Input

In the following sections the basic skills of the graphical input are described.

The Mouse Buttons

All operations can be done with the left mouse button (called <MB1>), if the corresponding menu option has been selected before.

Left Mouse Button

- A simple click will activate an option (selection of an element) or position the cursor.
- With a double-click, the mask of the selected element is called, the properties (toolbox) of the selected drawing element appears or a logical switch will be switched on/off.

- Dragging the mouse while <MB1> is pressed, a zoom area can be selected or elements and supporting graphic elements can be moved.

Right Mouse Button

A context sensitive pop-up menu is available for:

- the drawing area
- elements
- nodes
- text and results strings
- supporting graphical objects
- text edit fields

Element Input

The input of an element will be done with the left mouse button inserting every point with a mouse click. A symbol will be inserted at the current mouse position, if a double click (or two times a click on the same place) has been done. With a further mouse click the orientation of the symbol can be defined. The double-click is not needed for the line input, because there is no symbol defined.

Changing the View of the Diagram

The view of the diagram can be changed with the layout mode or zooming. In the layout mode, which can be chosen with the menu option "View - Layout", the diagram will be displayed as it will be printed/plotted (WYSIWYG). The margins, which can be entered with "File - Page Setup" are also considered. The page size has to be entered in the printer setup. The diagram can be moved in horizontal and vertical direction (see "Move View" in chapter "Menu Options"). The layout mode is significant before printing. In the layout mode the zoom function also is available.

The view of the diagram can be moved by:

- clicking the scroll bar between arrow and button. A half screen will be scrolled. Scrolling can be done horizontally and vertically.
- dragging die scroll button (move dynamically).
- the cursor keys of the keyboard (e.g. PgUp, Home)

Any network layer or graphic layer can be displayed on the screen. The network layer and graphic layer can be selected with the menu option "View - Network Layer.. " and "View - Graphic Layer.. ".

Network Layer Technique

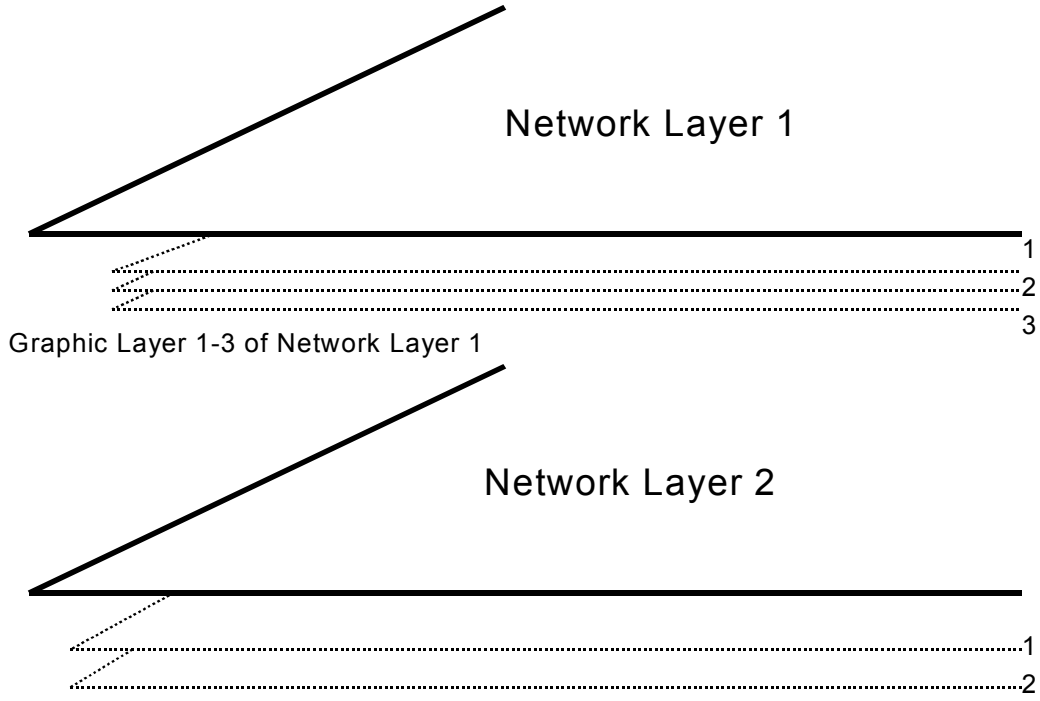
The user has the possibility to enter the network in different network layers or diagrams. With the help of this function, the user can for instance enter the high voltage network in one diagram and the low voltage network in several other diagrams. The high voltage network can also be drawn into several network layers or diagrams. An other use is zooming into stations. In the general diagram the station is drawn as "black box". In an other network layer the station can be drawn in detail with all protection and switching devices as well as the outlets. The following rules apply:

1. All elements, nodes and stations can be drawn in several layers. The nodes are the connection points of the networks and the layers. During the input of the same node/element in a different layer it is only required to select the name from the list (use the "?" button), because the other data are already known from an other layer.
2. It is possible to draw the line in one layer (e.g. general diagram) and the line outlet in an other layer (e.g. detail diagram). To enter an outlet a separate menu option is available. Before entering a line outlet, the line itself has to be defined!
3. Nodes and elements, which are entered with the list editor are also assigned to the current network layer.
4. The user can decide with the menu option "Edit - All Network Layers", if the system should display all nodes and elements of the network in the selection lists or only these in the current network layer.
5. To each network layer a name and a description can be assigned.
6. The calculation are made with the entire network, independent on the number of network layers defined.
7. Protection devices are also assigned to layers. Protection devices can also be drawn in several network layers.
8. An arbitrary number of protection devices can be assigned to an outlet.
9. The network layer can be changed by a mouse click on a node/element which is drawn in several layers (see "Change Layer" in chapter "Menu Options").
10. Two network layers can be joined together (see "Join Network Layers" in chapter "Menu Options").

The layer technique is taken into account when joining networks (joining project files). Every network will be assigned to one or several layers.

Important: Note the difference between a network layer and a graphic layer (see "Graphic Layer" on page 2-4). Each network layer can have any number of graphical layers. It is not possible to display two different network layers on the

same diagram. However all graphic layer can be displayed simultaneously on the same diagram.



Supporting Graphic

The system allows the user to insert supporting drawing objects, such as circles, lines, text, rectangle, etc. Pen size, font, font size, foreground and background color can be defined for every object. An existing drawing object can be changed later. Copy / paste functions for a single graphic object or a group of objects are available. It is also possible to use the symbol editor to add a supporting symbol to the single line diagram. Note that these symbols are stupid graphic objects without any connected data like e.g. line data. Raster images (TIFF, BMP, PCX, etc.) can be imported.

Graphic Layer for Supporting Graphic Objects

The user can define any number of graphic layers analogous to the network layers (see "Select Graphic Layer" in chapter "Menu Options"). In each graphic layer any number of supporting graphic elements can be entered or bitmaps imported. One or several layers can be displayed on the screen. Graphic objects and network elements (nodes and elements) can be in the same graphic layer. (see "Graphic Layer" on page 2-4). The layers to be displayed can be selected from menu option "View-Graphic Layer.. ".

When importing graphic objects through DXF-files with layers the system recognizes all DXF layers and saves the layers in the corresponding NEPLAN graphic layers.

Using the Tablet

If this option is active, the program works in the tablet mode. That means, that all inputs are made by the graphic tablet. The screen corresponds to the tablet size. The tablet-setup is made with the option "Option - Tablet". In the tablet mode the scroll bars are not available and the drawing area gets a red frame.

Tablet Setup

The menu option "Option - Tablet" is used to setup and to calibrate the tablet. The user has to enter the length of a diagonal twice, first in the dialog box "Tablet Setup" (two values in horizontal and vertical direction in millimeters) and on the tablet.

After having entered the values in the dialog box (close with OK) the user has to digitize two points on the tablet. The position of the first point is arbitrary. Important is the position of the second point, this one has to be exactly the entered number of millimeters (in horizontal and vertical direction) from the first one. Therefore it is recommended to draw a diagonal with a pencil on the tablet before calibrating the tablet (for instance 100 mm in horizontal and 100 mm in vertical direction).

Entering the Diagram, if the tablet is smaller than the diagram:

The tablet setup is independent of the diagram size. If the diagram is larger than the tablet, the user has to enter the diagram sheet by sheet. The diagram can be entered as follows:

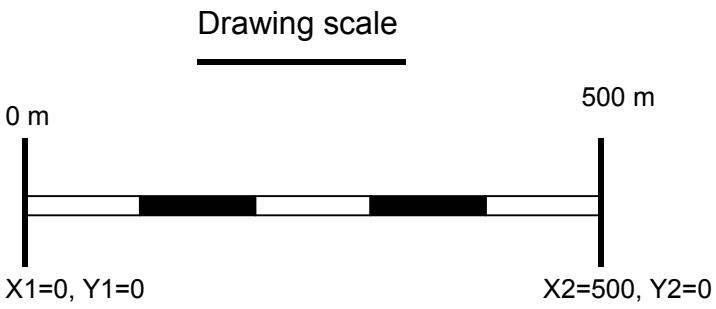
The diagram (paper) has to be fixed at the bottom left corner of the tablet. The part of the diagram on the tablet can be digitized. Afterwards move the diagram for an arbitrary number of cm to the left and re-fix the diagram on the tablet. Now the user has to move the view of the diagram. This can be done with the menu option "View - Move View". The same number of cm, the user has moved the diagram in horizontal direction to the left, must be entered here. The value to be entered here is negative **(Diagram move to the left means: negative value, Diagram move to the right means: positive value).** It is advisable not to move the diagram for the whole wide of tablet, because the overlapping area between the sheets would be missed. The same procedure has to be done, when moving the diagram in vertical direction.

Using a Map as Background Information

Instead of using the tablet for digitizing a network, it is possible to import a scanned raster image. This can be a geographic map, which can be used as background information. This map (cadaster) can be calibrated, so that real world coordinates can be used. The length of the lines will be calculated automatically during data input. The calibration has to be done with two points. After the first mouse click the first reference point in world coordinates (X and Y) can be

entered. Move the mouse and click for the second reference point. For the automatic calculation of the line length only the distance between the two points is important. If there is a scale on the map, which shows how much a certain distance is in meter or feet, then drawing scale can be calculated with a horizontal line (e.g. scale on the map is 500 m: X1=Y1=0 and X2=500 m, Y2=0).

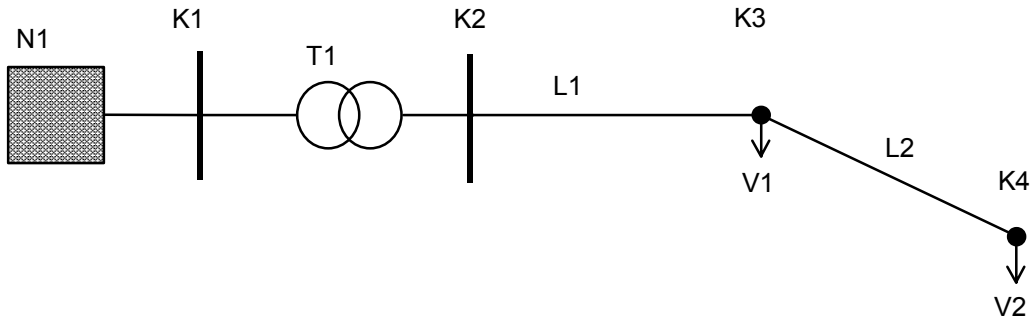
The import of a raster graphic is only available with the 32-Bit version.



Working with NEPLAN

In the previous sections the basic elements of the system have been described. In this section the above network will be entered step-by-step:

Network Example



The electrical data are:

Node K1:

Name = K1 Un = 16 kV

Node K2:

Name = K2 Un = 0.4 kV

Node K3:

Name = K3 Un = 0.4 kV

Node K4:

Name = K4 Un = 0.4 kV

Network feeder N1:

Name = N1

Sk"max = 110 MVA Sk"min = 100 MVA

R/X = 0.3 Z(0)/Z(1) = 100.0

LF-Type: SL Uoper = 100%

Transformer T1: Name = T1

Ur1 = 16 kV	Ur2 = 0.4 kV
Sr = 0.63 MVA	Vector group DY.05
uRr(1) = 1.03 %	ukr(1) = 4.0 %
uRr(0) = 1.03 %	ukr(0) = 3.8 %

Lines L1, **L2** (same data):

Name = L1 resp. L2	l = 0.3 km
R(1) = 0.101 Ohm/km	X(1) = 0.077 Ohm/km
R(0) = 0.683 Ohm/km	X(0) = 0.356 Ohm/km
Ir = 399 A	Perm. temp. = 80 degrees

Loads V1, **V2** (same data):

Name V1 resp. V2

LF-Type: PQ

P = 0.1 MW	Q = 0.05 Mvar

The following steps have to be done:

- Start NEPLAN.
- Open a new project with "File - New Project" from menu.

Node Input

- Select "Input - Node" (program mode will be set).
- Move the mouse cursor into the drawing area (approx. 3cm from the left border and 2cm from the top border). Insert a point (simple click). Move the mouse downwards (approx. 2 cm) and click again. In the appearing mask enter the node name and the nominal system voltage, e.g. K1 and Un = 16 kV. Close the mask with "OK".
- Insert node K2 in the same way. When inserting the nodes K3 and K4 give both points (two mouse clicks) at the same position. The nodes with different voltage levels have different colors.

Transformer Input

- Select "Input - Transformer".
- Enter a point in the middle of node K1 (simple click). Enter a second and a third point (or one double click point) in a distance of 1 cm from the first point. The transformer symbol will appear. The orientation of the symbol can now be changed with the mouse. Enter a point, so that the symbol will be drawn horizontally. The transformer input terminates when giving a point on node

K2. Enter the electrical data (see above) in the appearing mask and close it with "OK".

Line Input (graphic-orientated)

- Before entering line L1, it is better to deactivate the option for orthogonal drawing. Thus the line can be entered in an arbitrary way. Select "Options - Drawing Parameter" and deactivate the option "Orthogonal". Close the mask with "OK".
- Select "Input - Line".
- Enter a point on node K2 and afterwards a point on K3. The name and the electrical data (see above) must be entered in the appearing mask. Close the mask with "OK".

Line Input (list-oriented)

- Select the list editor with the toolbar button "L-Edit".
- Call the line editor with "Input - Line". Select "Insert" and a new line will be inserted. Enter the name L2 and select the node name K3 from the node selection list. This list will be shown, after having selected the question mark at the "From node". Do the same for the "To node" K4. Do not enter the data now. Close the mask with "OK".
- Select with double click line L1 from K2 to K3 in the list. The line mask will appear. Copy the line data with push button "Copy" and close the mask ("OK" or "Cancel"). Now re-select line L2 with double click and press "Paste" in the mask. The data from line L1 are inserted now. Close the mask with "OK". The functions copy and paste can also be called directly from the list editor.

Data Input from a Library

- Select line L1. In the mask of L1 the push button "Library" has to be pressed. A file selection box with libraries appears. Select a library. This library will be active until you will select an other one.
- Press the question mark behind the edit field "Type". All standard line types of the selected library will be listed. Select one and the data will be transferred. If you close the mask with "OK" the data will be considered and saved. You can get the old line data pressing "Paste" or if you close the mask with "Cancel". Please do so.

Graphical Input of an Element from the List Editor

- The line L2 has been defined (topological and electrical), but there are no graphical data for this line. You will enter now these data. To do this select

"El.Selection" (graphic data) in the line editor. All lines without graphical data are listed.

- Select with simple click the line L2 and press "Insert". The program returns to graphic mode and the starting and ending node of the line are marked in a different color.
- Enter the line starting at node K3 and ending at node K4 (remark: the line can not entered from K4 to K3). Afterwards the system will return to the line editor.
- Now quit the line editor with "Exit" and push the toolbar button "L-Edit" to return in graphic-oriented input mode.

Graphical Input of Network Feeder and Loads

- Select "Input - Network Feeder". Enter a point on node K1 and move the mouse to the left (approx. 0.5cm). Give a double click or two simple click at the same position and the symbol will be drawn. The orientation of the symbol is given by the mouse. Give a point, so that the symbol will be drawn horizontally. In the appearing mask enter the above data and close it with "OK".
- Select "Input - Load/Consumer" and enter the load analogous to the input of network feeder. The starting points must be given on K3 and K4. Do not forget the double click for symbol. Because the electrical data are the same for both loads, work with copy (load V1) and paste (load V2).

Saving the Data in a File

- Now you have entered the entire network. Select "File - Save as.." from the menu to save the network data and project to a project file. In the file selection box you have to enter the file name, e.g. test. The extension .MCB is not needed. After the file name input close the box with "OK".

Free Graphic (supporting graphic)

- We insert a text into the drawing area. Select "Draw - Graphic" and in the appearing dialog box select "text". The text "Network Example" can be entered in the text-line field. Close the box with "OK", move the mouse and give a simple click. The input is finished now.
- The font size of the text can be changed. Click with the **right** button on the text "Network Example". A popup menu appears. Select the menu item "Change Graphic". In the appearing dialog box the font size, font style and font type can be selected. The text can be changed. Enlarge the font size to 10 mm and close the box.
- Enter a line. Select "Draw - Graphic" from the menu, select "Line" in the appearing dialog box and close the box. Move the mouse to the starting point of the line. With dragged mouse button the line can be entered (e.g. under-

line the text "Network Example"). If you release the mouse button the input will be finished. The line can be changed in a same way as the text.

Saving the Data in the Same Project

- Save the project with menu option "File - Save" without giving a file name.

Editing the one Line Diagram

- After having selected "Edit - Element - Move" the nodes and elements can be moved. Click the transformer T1. The transformer will be redrawn dashed. The small quadrates at the nodes K1 and K2, as well as in front and behind the symbol are the "moving points". Click the quadrate in front of the symbol and move the mouse with dragged button to the left and to the right. The position of the symbol will be changed. Click one of the quadrate at the nodes (e.g. K1) and move the mouse with dragged button up and down. The quadrate can not be moved over the node, because the connection must remain the same.
- Click the node K2. The node will be redrawn dashed. Move one of the quadrate at the ends of the node with dragged button up and down. The length of the node can not be arbitrary, because the connection with the elements must remain.
- Selecting "Edit - Group - Move" the user can select and move entire networks or part of them without changing the connection of the elements (topology). Select this option and move the mouse to the left beyond node K2. If you drag the button and move the mouse, you can define a rectangle. The network in the rectangle will be selected and can be moved afterwards. Select such a rectangle that the nodes K3 and K4 are included. The selected network will be redrawn dashed after having released the button. Move the cursor on the node K4 and move the mouse with dragged button. The network will be moved according to the mouse position. Releasing the button the network will be redrawn at the new position.
- Nodes, elements, groups or partial networks can be deleted. Select "Edit - Element - Delete " from menu and click the transformer T1. The transformer is now deleted electrically and graphically. Select node K2. If a node is deleted, then also all connected elements are deleted.
- It is also possible to delete only graphic data. Select "Edit - Element - Delete Graphic" and click on the line L4 between K3 and K4. The line is deleted graphically, but the line is still defined. The line L4 would be listed in the line editor.
- If you have deleted a node, an element, a network group or a partial network for mistake use menu option "Edit - Undelete" to return to the old state. Only the last DELETE function can be undeleted.

Opening an Existing Project

- Because our example network has been changed due to the movements and deletions, we open the project again. Select "File - Open - Project". The system will ask you if the changes should be saved. Press "No", otherwise the old project will be overwritten (or you can save it in an other project). Give the file name in the file selection box, e.g. test. The old project (TEST.MCB) will be opened and the network will be displayed.

Load Flow Calculation

- Select "Calculation - Load flow - Parameter" and activate the check box "Text file". Select push button "Res.format" and activate the check box "All" (List). Close both dialog boxes with "Close" and "OK". Calculate the load flow with "Calculation - Load flow - Calculation". The results are displayed in the diagram.
- With the menu option "Results - List - Last Calculation" the results are listed. With the scroll bars the document (listing) can be scrolled. Close the document window with "File - Close".

Moving the Results in the Diagram

- The result boxes can be moved or deactivated. Use the right mouse button and click a result box to activate the popup menu Select " Move - Text" and click the result box of node K1. With dragged button you can move the box around. In the same way you can move around all other result boxes. The node names can also be moved.
- Select "Edit - Text - On/Off" to deactivate a result box. Each box that has been selected (clicked) will be deactivated. A deactivated box can be reactivated clicking the logical switch of an element or clicking the node. The position and the state (on/off) of a result box will be saved in the project file.

Short Circuit Calculation

- Select "Calculation - Short circuit - Parameter" and activate the check box "Text file". Close the dialog box with "OK". Before we can calculate short circuit we have to enter the fault locations. Select "Calculation - Short circuit - Fault Location" and mark the desired node(s) with double click (e.g. K4). Faulted nodes are marked with "**". Fault locations can also be defined with the right mouse button and the popup menu.
- After having closed the dialog box select "Calculation - Short circuit - Calculation". The calculation will be done and the results inserted. In the same way as explained above the result boxes can be moved. At the moment only the results at the fault location are displayed. To get all results of the network the user has to change the fault distance in the short circuit parameter mask (from 0 to 3).

Changing the Description Fonts

- You can select different fonts for the labels of the stations, nodes, elements and protection devices and for the result display in the menu "Options - Font - ...".

Changing the Colors of the Diagram

- Now the network is colored according to the voltage levels. The network can be colored according to partial networks. Select "Options - Drawing Parameter" and activate the check box "Part. networks" (Coloring). Close the dialog box with "OK". The diagram is now colored with the same color. Built now partial networks switching off the line L1 at the node K2. This can be done double clicking the left mouse button on the logical switch of the line. The system recognizes the two partial networks and each one get its color. This option is especially helpful when working with large networks.

Saving Partial Networks

- Entire partial networks can be deleted and saved separately. Select the options "Edit - Partial Network - Delete" and click (left mouse button) on node K3 or K4. The partial network with nodes K3 and K4 will be deleted. The remaining network can now be saved in a project file with an arbitrary name (see "Saving the Data in a File" on page 2-22).
- The option "Delete excl." can also be chosen. In this case all partial networks are deleted exclusive the one which has been selected.
- Select option "Edit - Undelete" to get back the deleted partial network and switch on the logical switch on line L1 with double click.

Printing the Diagram

- Before you print the diagram make sure the printing orientation is landscape. Select "File - Printer Setup" from the menu and check the orientation. Close the dialog box with "OK". The page margins can also be set. Select therefore "File - Page Setup". Enter no margins and close with "OK". The printing can be started with "File - Print". The diagram displayed in the drawing area (screen) will be printed.
- The diagram can also be printed in an arbitrary zoom-level. Select therefore "View - Zoom - Zoom In". Enter an arbitrary rectangle with the dragged button (entering the zoom-level). Now release the mouse and select "File - Print" to print the diagram. Thus only a part of the diagram will be printed.

Quitting the Program

- Select "File - Exit" to quit the program. If the user has made some changes after having opened or saved the project, the program will ask, if the project has to be saved.

In this short introduction only the basic functions of the program are shown. In the following section all menu option are explained.

Tips from the Practice

Important tips from the practice are given here.

Asymmetrical Network Structure

Representation of an Asymmetrical Line

It is recommended to enter lines in a compact way. A 3-phase line from node A to B can theoretically be entered as three single phase lines, which are coupled between each other. In this way the program will work not only with the current circuit resp. the series impedance matrices but also with the coupling matrices. This increases the calculation effort. The better way is to represent the 3-phase line with one 3-phase line. The same is valid for a 2-phase line.

Graphic

Inserting a New Node into a Line

- Graphical input of the new node.
- Selecting the line data (right mouse button popup menu Mask...).
- Coping the data selecting the push button "Copy" and quitting the mask with "Cancel".
- Select menu item "Put Element to other Node". Move the end point of the line to the new node.
- Entering a new line between the new node and the ending node. In the line mask the data can be transferred with "Paste". The line length has to be corrected.

Deleting a Node in a Line (Sleeve)

- Select menu item "Put Element to other Node" with the popup menu of the line (right mouse button). Move the end point of the line to the new end node.
- Deleting the node with "Edit - Element - Delete" and clicking the node.

Load Flow

Divergence using PV-Nodes

- When using PV-nodes the Newton-Raphson method should be used. The user has to take care that no disconnect or load switches as well as short lines will be connected to the PV-node, because of numerical problems. If disconnect and load switches are connected to PV-nodes it is advisable to reduce them during calculation. If for example a generator is connected to a bus bar through disconnect switches (one open, one closed), the generator node should be marked as reducible (see "Node Data Input" in chapter "Element Data Input and Models").
- When using the current iteration method the same is valid. The convergence can be additionally influenced by the accelerating factor (see "Calculation Parameters (LF)" in chapter "Load Flow"). Probably the value must be reduced until 0.05 to obtain convergence. The convergence criteria should also be reduced.

Switching Topology or Connecting Motors

- If the motor starting module is not available and the user like to make voltage drop calculation when switching the topology or connecting motors, the network impedance (network feeder) must be represented by a line. In the normal load flow calculation the internal network impedance (Sk", Un) is not considered.