

XTAP
(eXpandable Transient Analysis Program)
Version 2.4
Instruction Manual



Central Research Institute of Electric Power Industry (CRIEPI)

December 2022

XTAP (eXpandable Transient Analysis Program) Version 2.40

Development

Central Research Institute of Electric Power Industry (CRIEPI)
Energy Innovation Center, Electric Power Engineering Research Laboratory, System
Engineering Research Laboratory
<http://criepi.denken.or.jp/en/index.html>

Development assistance

Denryoku Computing Center
<http://www.dcc.co.jp/>

Support information

Access the URL below after obtaining an ID and password from CRIEPI.
<http://www.xtap.jp/>

Copyright information

XTAP Ver.2.40 was partly developed using the following redistributable programs:

AnaGram - A System for Syntax Directed Programming
Version 3.0 Copyright © 1993-1999 Parsifal Software

Out of The following components of MATLAB 8.6.0.267246 (R2015b), 1984-2015, The MathWorks, Inc.), MATLAB Compiler and its runtime libraries, MCR Libraries (Licensing to users is restricted under the license agreement between CRIEPI and The MathWorks, Inc.)

ComponentOne Studio
Copyright (c) 2004 GrapeCity, Inc.

SPREAD for Windows Forms
Copyright (c) 2004-2012 GrapeCity inc.

Developer credits

CRIEPI

Taku Noda, Rikido Yonezawa, Yohei Tanaka, Tomo Tadokoro, Guilherme Cirilo Leandro, Toru Miki, Kazuyuki Ishimoto, Toshiya Ohtaka, Naoki Gibo, Toshiaki Kikuma, Shinsuke Aida, Shogo Fukui, Jota Suzuki, Koya Naruse, Shugo Takahashi, Orie Sakamoto (currently at Sophia University), Kenichiro Sano (currently at Tokyo Institute of Technology)

Denryoku Computing Center

Kiyoshi Takenaka, Itaru Watanabe, Yoichi Sekiba, Takayuki Nakajima, Hiroshi Takizawa, Toshiyuki Okada, Masahiko Takemura, Shigetaka Suzuki, Tsutomu Maejima, Mitsuo Takateyama, Minoru Ishiwata, Yasushi Yokoi, Ryo Anan, Taro Yoshida, Wakako Ishiwata, Izumi Ichimori, Mamoru Hosoya, Tadahiko Nagaoka

Trademark

XTAP is a registered trademark of the Central Research Institute of Electric Power Industry (CRIEPI).

Contents

1	Introduction.....	1
1.1XTAP	1
1.2Features of XTAP	1
1.3Operating environment.....	1
1.4Software upgrade caution.....	2
1.5XTAP-related CRIEPI reports	3
2	Starting XTAP, Main Window	5
2.1Starting XTAP	5
2.2Terminology	7
2.3Main window.....	8
2.4Menus in the main window	9
3	Workflow From Circuit-Diagram Creation Through Analysis	22
4	Creating Circuit Diagrams	24
4.1Part types	24
4.2Part placement (electrical-circuit part/control-block part).....	24
4.3Part operations	26
4.4Part properties screen	31
5	Part Connections.....	35
5.1Creating electrical-circuit wires.....	35
5.2Electrical-circuit wire (node) properties.....	38
5.3Creating control-system wires	39
5.4Control system wire properties	40
5.5Creating multiple-signal control system wirings	40
5.6Creating three-phase buses	43
5.7Creating line concentrators.....	47
6	Analysis Processes.....	52
6.1Analysis settings.....	53
6.2Analysis procedure (analysis type: transient analysis)	54
6.3Analysis procedure (analysis type: steady-state initialization → transient analysis)	58
6.4Analysis procedure (analysis type: power flow calculation → steady-state initialization → transient analysis).....	62
6.5Analysis procedure (analysis type: frequency characteristic calculation)	68
6.6Files related to analysis processes.....	71
6.7Search function	72
7	Consecutive-Analysis Function	73
7.1Overview of consecutive analysis function	73
7.2Opening the consecutive-analysis settings window.....	73
7.3Consecutive-analysis settings window	74
7.4Setting the number of cases and of parameters.....	75
7.5Parameter input method	76
7.6Ending/canceling parameter settings.....	80
7.7Executing a consecutive analysis.....	81
7.8Statistical processes for analysis results	83
7.9Executing statistical processes.....	84
7.10Displaying analysis results in XPLT	88
8	Statistical Switching-Surge Calculation Function	89
8.1Overview of statistical switching-surge calculation function.....	89
8.2Opening the settings window for a switching-surge calculation.....	89

8.3	Analysis example.....	90
9	XPLT (Graphing Program).....	93
9.1	Starting XPLT	93
9.2	Selecting the analysis result files.....	93
9.3	Displaying graphs	94
9.4	Graph editing	103
9.5	Menus.....	106
10	Creating and Editing Macro Parts	114
10.1	Packaging procedure (creating macro parts).....	114
10.2	Changing macro parts	120
10.3	Editing macro-part icons.....	123
10.4	Connection-terminal settings	130
10.5	Registering parameters/formulas	133
10.6	Part-category window composition; creating/deleting categories	137
11	Other	139
11.1	Starting XTLC (line constants calculation program)	139
11.2	Starting XFIT (system identification program)	139
Appendix 1. XTLC (Line Constants Calculation Program) Instruction Manual		141
Appendix 2. XFIT (System Identification Program) Instruction Manual		177

1 Introduction

1.1 XTAP

XTAP (Expandable Transient Analysis Program) is a program for analyzing transient phenomena in power systems and other electric circuits at the waveform level. This type of analysis has so far been done using programs developed outside Japan, such as EMTP. XTAP is thus the first program in this field developed specifically in Japan. It provides superior analysis performance compared to conventional programs for analyzing power-electronics devices, such as inverters for solar-power generation. CRIEPI is also working on developing programs to meet the needs of the domestic market.

1.2 Features of XTAP

Feature 1: High calculation precision, value stability

XTAP uses highly efficient calculation algorithms for value integration and circuit formularization, to achieve a higher calculation precision and value stability than conventional programs (see the list of CRIEPI reports in Section 1.5 for information on internal program calculation principles and the algorithms used).

Feature 2: Reliable convergence performance (ideal for power-electronics circuit analysis)

Thanks to its proprietary convergent calculation and equation solution algorithms, XTAP quickly performs circuit analysis calculations, even in cases of extreme nonlinearity. The analysis is almost perpetual (i.e., it does not converge). This algorithm is ideal for power-electronics circuits.

Feature 3: Provision of device models for domestic power systems

Models needed for domestic power-system analysis are provided standard. Various control blocks for creating device-control mechanisms can be used. Help information is provided in Japanese for all models.

Feature 4: Graphical user interface

The XTAP interface simplifies circuit creation, enabling the creation of power-system models and power-electronics circuits. Analysis results (voltage/current values at user-specified locations) can also be checked using simple operations.

1.3 Operating environment

XTAP requires the following environment:

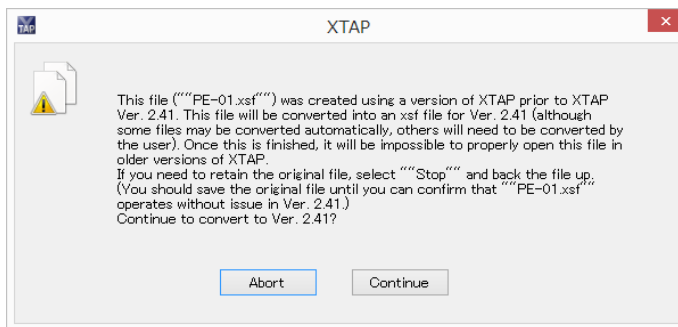
Operating system	
Operating system	Microsoft Windows 8, 8.1 (32-bit and 64-bit versions)
	Microsoft Windows 10 (32-bit and 64-bit versions)
PC	
Memory	32-bit OS: 2.0 GB or more (4.0 GB or more recommended)
	64-bit OS: 4.0 GB or more (8.0 GB or more recommended)
Hard-disk capacity	8 GB or more
Software required	
Software	MATLAB Compiler Runtime R2015b
	Microsoft .NET Framework 4.8.1 or newer

Screen DPI setting

Text may not be displayed properly by XTAP if the DPI setting of the screen is changed. XTAP will display text correctly for a standard setting of 96 DPI. An increasing in this standard setting by the user (e.g., 125% or 150%) is not problematic. The standard default setting in Windows is often 96 DPI, but the DPI setting can be changed in the 'display properties' dialog. Consult the help information provided with your operating system for instructions.

1.4 Software upgrade caution

XTAP Version 1.20 and later versions are backward-compatible, supporting circuit-drawing files (*.xsf files) created by older versions. When the below dialog box appears, click 'Continue' to start conversion of the file so that it can be used in the current version. The file conversion process is done either automatically or manually by the user (for example, in the case of user-defined components). Note that once a circuit-drawing file has been converted to a more recent version of XTAP, it can no longer be opened using the older version.



For details regarding this file conversion, see *“Using a circuit diagram file (xsf file) created in an older version”* on the XTAP user support site.

1.5 XTAP-related CRIEPI reports

Reports released by CRIEPI relating to the development of XTAP are listed below. The full text of these reports is available free of charge from the CRIEPI download site. (The reports are in Japanese with English abstracts.)

Report No.	Title
H06002	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 1) -- Fundamental Design --
R06017	Development of Inverter Simulation Program (Part 1) - Modeling of MOSFET and Diode -
H07005	A Transmission Line Model for Electromagnetic Transient Analysis Based on the Frequency Region Partitioning Algorithm
H07004	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 2) - Development of a Fundamental Algorithm for Steady-State Calculation -
R07016	Development of Inverter Simulation Program (Part 2) - Verification of Analysis Accuracy Based on Actual Measurement -
R08022	Development of instantaneous simulation method of an induction motor for transient analysis
R08027	Design Guideline and Marginal Performance of a SiC Diode Applied Inverter
R08015	Accelerated Electromagnetic Transient Simulations of Power Electronics Circuits using a Variable Time Step Algorithm
H08002	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 3) - Improvements of Convergence Properties for Nonlinear Circuits -
H09007	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 4): Practical Steady-State Initialization
R09007	Development of Support System of Setting and Coordination between Protection Relays- Method of Checking Coordination including Earth Fault Protections in Distribution Systems with Isolated Neutral System-
R09010	Optimum Circuit Configuration of SiC Device Applied STATCOM for Distribution System
R09015	Stable Operation Technique for PCS of PV Power Generation at Grid Recovery after Voltage Sag
R10008	Study on introduction of SN transition type FCL into looped distribution system
R10025	Development of a synchronous machine model considering the magnetic saturation in the stator core for electromagnetic transient analysis of power systems
H10008	Study on Lightning Protection Design for Distribution, Telecommunication and Consumer Circuits (1) -Analytical Study on Lightning Overvoltages on Distribution Lines with Telecommunication Cables-
H11010	Proposal of a Sparkover Model for Short Wave Tail Impulse Voltage Waveforms Applied to Distribution Insulators
H11013	Characteristics of Spark Discharge on Faulty Insulation of 6.6kV Distribution Lines -A Proposal of Evaluation Method for Low Frequency Pulsed Electromagnetic Interference-
H11009	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 5): A Fast Algorithm for LU Decomposition of a Large Sparse Matrix
H11006	Evaluation of Line Constants of Short-Length Underground Cables: Development of a Measurement Method and Validation of a Calculation Method
H12005	Review of Simulation Methods for Electromagnetic Transients in Power Systems and Simulation Cases Using XTAP (Part 1): Switching Transient Overvoltage Simulations
R12005	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 6) - Performance Improvements for Nonlinear Circuits -
R12020	Preventing Islanding by Distributed Power Generation at Transmission System Fault- Evaluation of Characteristic of Detecting Islanding by Power Conditioning Subsystem for Interconnecting Many Distributed Power Generations in Area with Rotated Type Distributed Power Generation -
R12010	Harmonic Instability at Parallel Operation of Line Commutated Converter and Voltage Sourced Converter
H13007	A Dynamic Voltage Simulation Method for Power Distribution Systems by Using Electromagnetic Transient Analysis (Part 1): Modeling of Distribution Substation and SVR

H13005	Development of an Electromagnetic Transient Analysis Program for Power Systems(Part 7): Fast Simulations by Parallelization of Updating Process of Circuit Components
H13010	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 8): Fast Simulations by Single-Phase Phasor-Based Modeling of Remote Power Systems
R13027	Development of Modular Multilevel Converters Model for High Speed Electromagnetic Transient Analysis
H14013	Development of a Frequency-Dependent Line Model for Electromagnetic Transient Simulations without Model Identification Problems (Part 1): Simulation Performance of an FDTD-Based Line Model
H14014	A Dynamic Voltage Simulation Method for Power Distribution Systems by Using Electromagnetic Transient Analysis (Part 2): Modeling of STATCOM
R14016	Control and Protection Scheme of HVDC System with Modular Multilevel Converter using Full Bridge Cell - Part2: Development of an Operation Method under AC High-Voltage Situation and a Method for Fast Start-
H15004	Development of an Electromagnetic Transient Analysis Program for Power Systems (Part 9): A Fast Frequency-Scan Algorithm
H15014	A Fast Simulation Algorithm for Distribution-System Transients Based on the Theory of Dynamic Phasors (Part 1): Proposal of Fundamental Concept
R15022	Development of Converter Model for High-speed Transient Analysis - Averaged Model with Rectifier Circuit -

2 Starting XTAP, Main Window

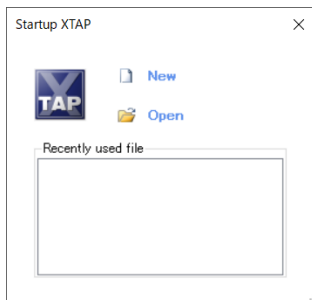
2.1 Starting XTAP

The following screen appears after XTAP is launched:

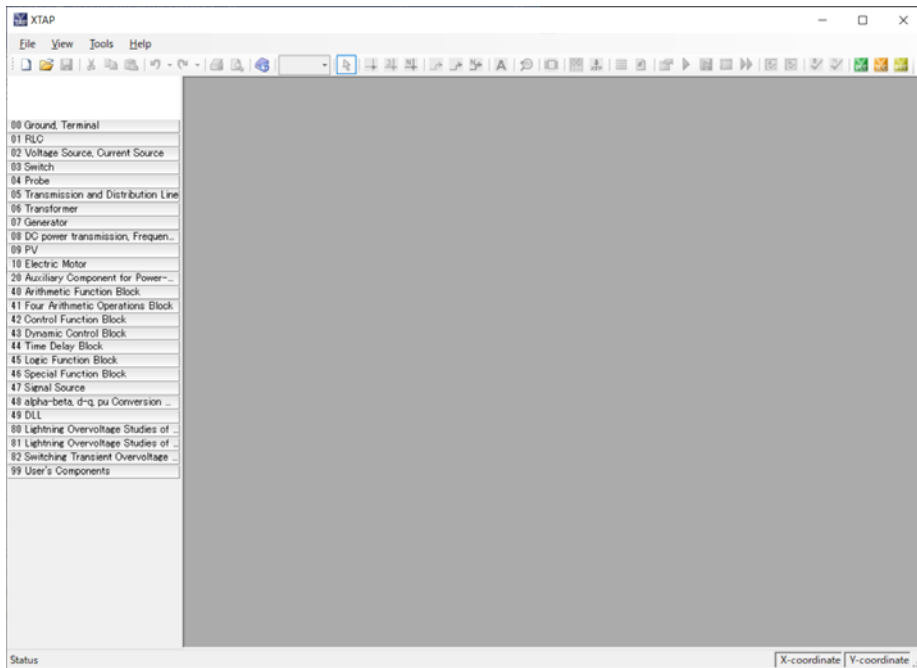


When the program starts, a menu appears showing the options 'New' and 'Open existing file'. To create a new circuit diagram, select 'New'. To open an existing circuit diagram (*.xsf file), select 'Open existing file'.

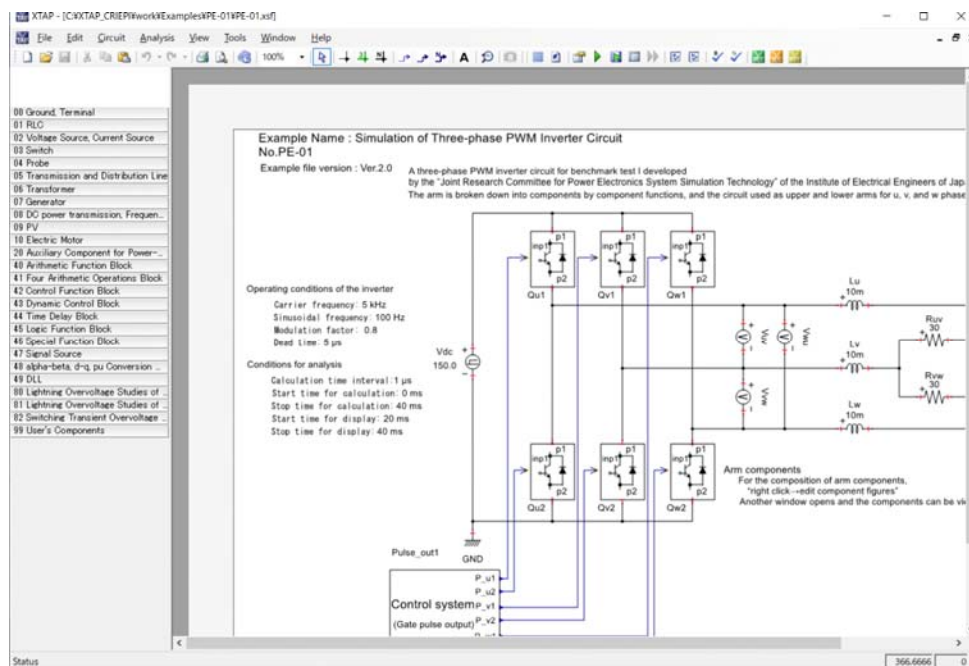
It is also possible to double-click an xsf file displayed in Explorer in order to launch XTAP and open that circuit diagram (if an xsf file is double-clicked while XTAP is already running, that file will be opened in the running XTAP). Another method of opening an xsf file is to drag and drop it onto the XTAP window.



Selecting 'New' opens a blank canvas like the one shown below.



Selecting a circuit diagram (*.xsf file) using 'Open existing file' or 'Recently used file' opens the selected circuit diagram.



2.2 Terminology

Terms used within XTAP are defined as follows.

part

Resistors (linear), sine-wave voltage sources and other component elements of electrical circuits or control blocks are referred to as 'parts'. The term refers to the items in the 'part category' window on the left side of the screen.

node

A wire used to connect parts in an electrical circuit is called a 'node'. Control-system wires connecting control blocks are used to denote a connection between the blocks, and are different from nodes.

branch

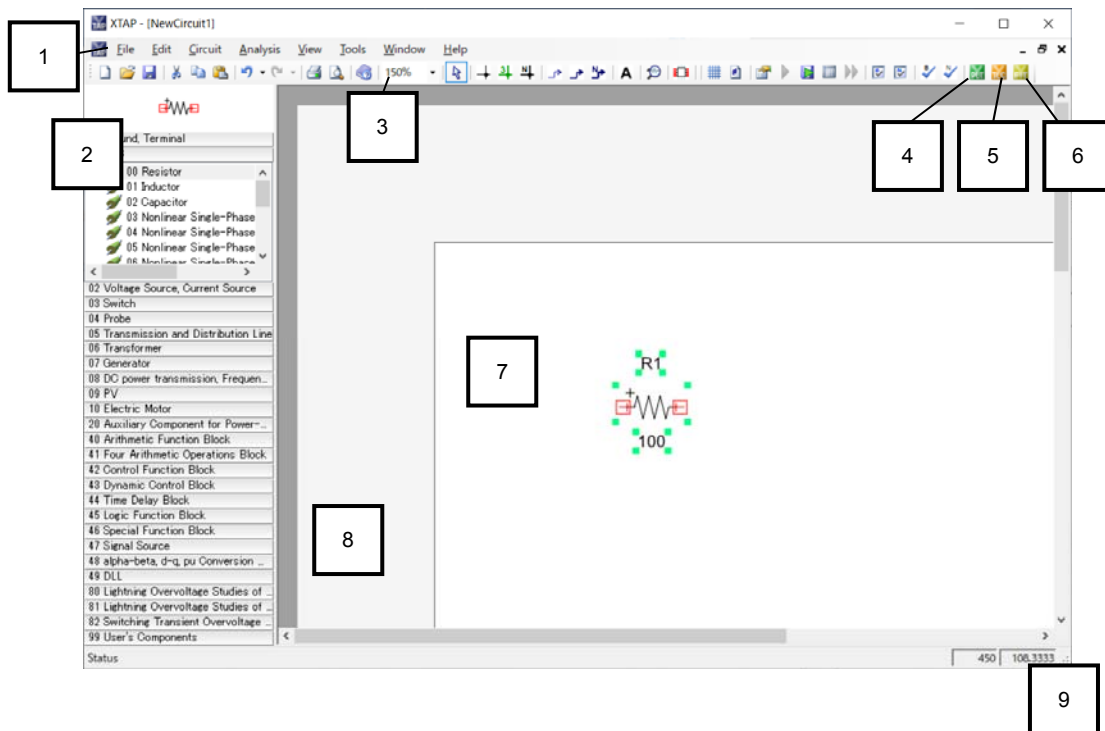
A 'branch' is a synonym of 'part'.

packaging, macroing

A set of several parts that compose an electrical circuit or control block can be registered collectively as a single part. This operation is known as 'packaging'. The operation of packaging electrical circuits or control blocks containing parts created by packaging is called 'macroing'. Parts created in this manner are sometimes referred to as 'macro parts'.

2.3 Main window

This section describes the main window, which appears when 'Create circuit diagram' has been selected from the 'File' menu and a new circuit diagram is opened.



1. Menus in the main window

See Section 2.4 for details.

2. 'Part category' window

This window is used to register parts such as electrical-circuit parts and control-system block parts. '99 User's Components' is used to register a user-packaged part. Parts other than '99 User's Components' cannot be changed.

3. Zoom setting

This setting is used to change the screen-display size by selecting the desired magnification. The screen display size can also be changed by holding down the 'Ctrl' key on the keyboard while turning the mouse wheel. At this time, it operates so that the mouse position at the moment when Ctrl key is pressed can be located at the center of the screen.

4. XPLT button

This button starts XPLT (the graphing program). XPLT is described in Chapter 9.

5. XTLC button

This button starts XTLC (the line-constants calculation program). XTLC is described in Appendix 1.

6. XFIT button

This button opens XFIT (the system-identification program for XTAP). XFIT is described in Appendix 2.

7. Circuit-diagram canvas

This space is used to place the parts and create the circuit diagram.

8. Gray circuit diagram canvas background color

The canvas background color is gray when editing a circuit diagram, or when editing a circuit in a packaged part (editing a macro part). Part editing is described in Chapter 10.

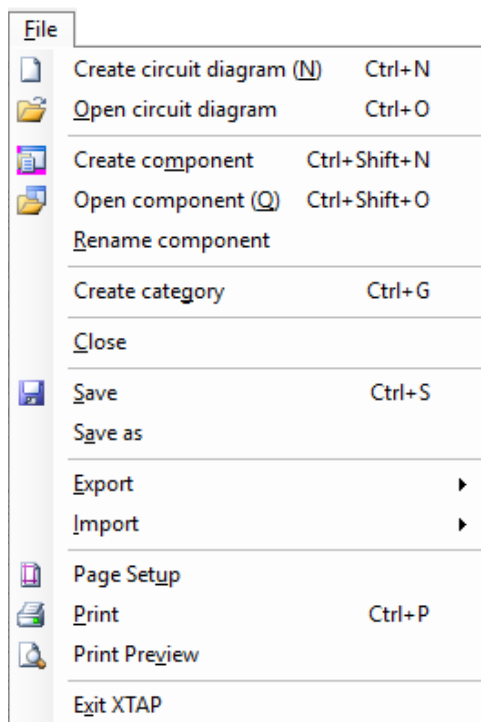
9. Circuit diagram canvas coordinate display

When the cursor is located over the canvas, its coordinates in the canvas coordinate system are displayed at the bottom-right corner of the window.

2.4 Menus in the main window

This section describes the top-level XTAP menus and the corresponding menu items.

(1) File (F) menu



Create circuit diagram

- opens a new (blank) circuit diagram. The same operation can be performed by clicking the toolbar button indicated below.

Open circuit diagram

- opens a previously created circuit diagram (*.xsf file). The same operation can be performed by clicking the toolbar button indicated below.

Create component

- allows the definition of a new part. Defined parts are registered in the '99 User's Components' category.

Open component

- opens a part already registered as a user-defined part.

Rename component

- allows a change in the name of a part that was previously registered as a user-defined part.

Create category

- creates a new category in the '99 User's Components' category.

Save

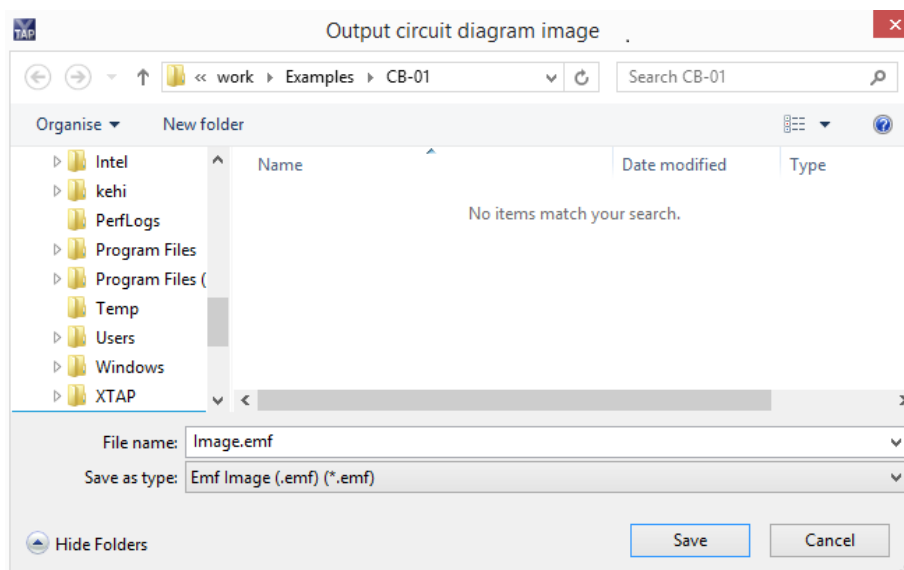
- saves the circuit diagram, replacing any previous version with the current version. The same operation can be performed by clicking the toolbar button indicated below. This button is active only when the circuit diagram has been changed after being opened. The button is inactive (with a gray appearance) otherwise.

Save as

- saves the circuit diagram under a new name.

Export

- outputs and saves the part information registered in the 'Part category' window in the 'XTAP Components Backup File' format. It is also possible to output the image of a circuit canvas as an image file. Selecting this menu item displays the window shown below. This window can be used to enter a file name and select the desired image-file type in which to save the circuit canvas. The EMF, bitmap, GIF, JPEG, PNG, or TIFF file format can be selected.



Import

- reads the part information output by the export function into XTAP, and registers it in the 'part category' window. It is possible to either import all part information as a batch (automatic part importing) or select parts one at a time (individual part importing).

Page settings

- defines settings for printing, such as the page size and printer settings.

Print

- prints the circuit diagram. This operation can also be performed by clicking the toolbar button indicated below.

Print preview

- displays a preview of how the printed circuit diagram will appear. This operation can also be performed by clicking the toolbar button indicated below.

Exit application

- terminates the XTAP application.

(2) Edit (E) menu



Undo

- Undo one of the changes made to the circuit.

Redo

- Restores one of the changes made to the circuit canceled by “Undo”.

Cut

- Cuts the selected component or node from the circuit diagram.

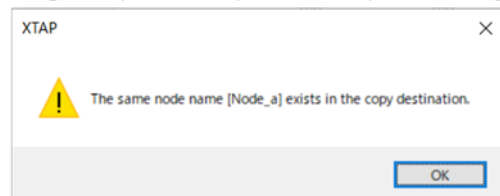
Copy

- Copies the selected component or node.

Paste

- Pastes the “cut” or “copied” component or node into the circuit diagram. The names of the pasted component and node are as follows.

- Component: Will be renamed to a unique new name.
 - Node
 - ✓ Default name: Will be renamed to a unique new name.
 - ✓ User-set name: Will be pasted with the original name (considered the same node).
- When pasting, a message indicating this will appear.



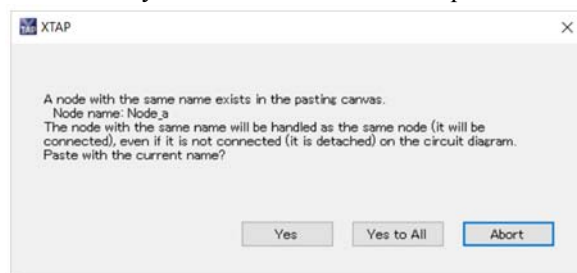
Keep Name and Paste

- Paste the “cut” or “copied” components or node to the circuit diagram while keeping the name. It operates as follows, depending on components and nodes that exist in the circuit of the paste destination.

- If a component with the same name exists:
XTAP does not allow a component with the same name to be placed on the same hierarchy. Therefore, the following message will appear, asking you to select either renaming and pasting or canceling the paste operation. If you select “Yes to all”, XTAP will automatically select “Yes” for all subsequent cases where the same message appears.



- If a node with the same name exists:
In XTAP, nodes with the same name existing on the same hierarchy are considered to be the same node (even if they are far apart on the circuit diagram campus). Therefore, the following message will appear, asking you to select either pasting while keeping the name or canceling the paste operation. If you select “Yes to all”, XTAP will automatically select “Yes” for all subsequent cases where the same message appears.



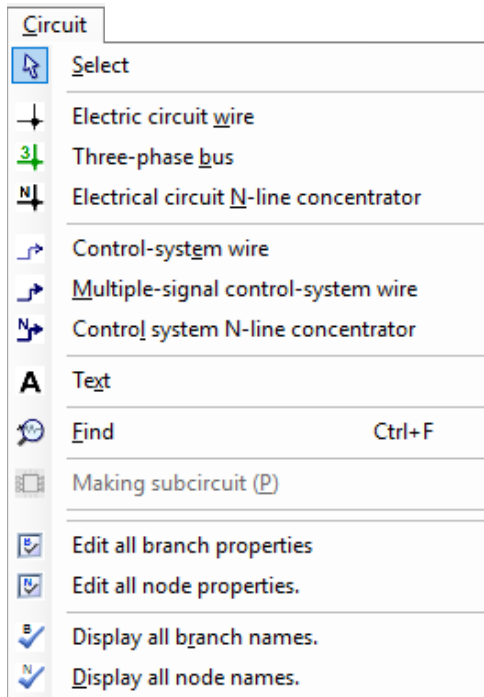
Delete

- Delete the selected component or node.

Select All

- Copies the selected component or node.

(3) Circuit (C) menu



Select

- selects a component (e.g., a part or wire) located on the circuit diagram canvas. This operation can also be performed by clicking the toolbar button indicated below.

Electrical-circuit wire

- switches to the mode used to create an electrical circuit wire. This operation can also be performed by clicking the toolbar button indicated below.

Three-phase bus

- switches to the mode used to create a three-phase bus. This operation can also be performed by clicking the toolbar button indicated below. See Section 5.6 for details of how to create a three-phase bus.

Electrical-circuit N-line concentrator

- switches to the mode used to create a line concentrator that packages multiple electrical circuit wires. This operation can also be performed by clicking the toolbar button indicated below. See Section 5.7 for details of how to create a line concentrator.

Control-system wire

- switches to the mode used to create a control-system wire. This operation can also be performed by clicking the toolbar button indicated below. See Section 5.3 for details of how to create control-system wires.

Multiple-signal control system wiring

- switches to the mode used to create wiring for a multiple-signal control system. This mode can also be selected from the 'Multiple-signal control system wiring' icon button. See Section 5.5 for details.

Control-system N-line concentrator

- switches to the mode used to create a line concentrator that packages multiple control system wires. This operation can also be performed by clicking the toolbar button indicated below. See Section 5.7 for details of how to create a line concentrator.

Text string

- enters a comment in the circuit diagram. This operation can also be performed by clicking the toolbar button indicated below.

Search

- searches for a component placed in the circuit diagram. This operation can also be performed by clicking the toolbar button indicated below. See Section 6.7 for a description of the search function.

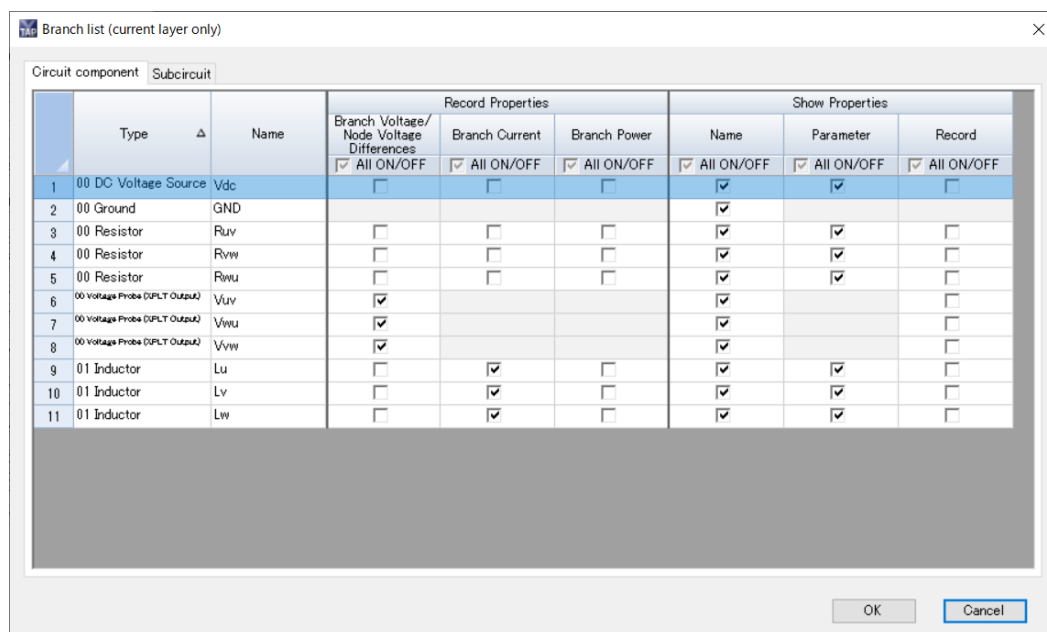
Package

- packages the parts selected in the circuit diagram and creates a new composite part (a macro part). This operation can also be performed by clicking the toolbar button indicated below. This menu option is only active if a part is selected (otherwise the icon is grayed out). See Section 10.1 for details of how to package parts (create macro parts).

Edit all branch properties

- allows editing of the recording properties and display properties of all the branches (circuit parts and control parts) in the currently active layer simultaneously. This operation can also be performed by clicking the toolbar button indicated below.

Selecting this menu item displays the following window. Use this window to turn the recording/display properties of parts ON or OFF. Clicking on one of the cells in the column headings at the top left of the table sorts the list in the corresponding order. Each click of these cells toggles the sort order between ascending and descending. Changes made in this window are applied to the corresponding parts. To switch between circuit parts and control parts, use the tabs at the top left of the window.



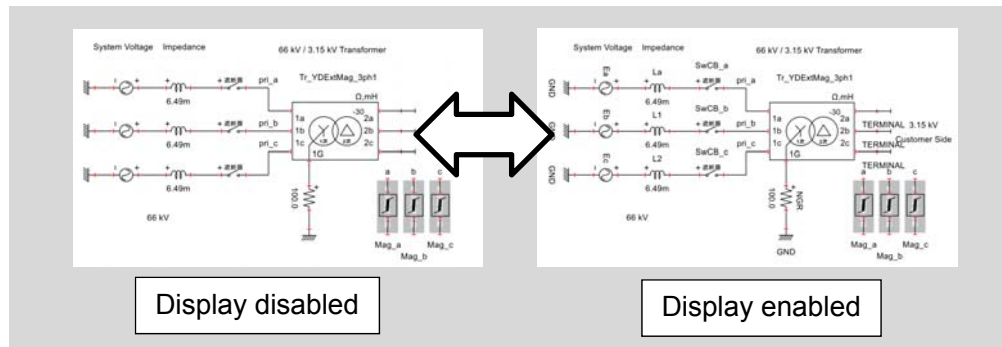
Edit all node properties

- lets you edit the recording properties and display properties of all the nodes (electrical-circuit and control-circuit wires) in the currently active layer simultaneously. This operation can also be performed by clicking the toolbar button indicated below. The operation performed by selecting this menu item is the same as for the 'Edit all branch properties' menu item.

Display all branch names

- displays the names of all the branches in the currently active layer. This operation can also be performed by clicking the toolbar button circled below.

Selecting this menu item, or clicking the corresponding toolbar button, displays all the branch names. A further selection of the menu item or click of the button disables this function, restoring the branch-name display settings specified in the properties for each part. *Note that this function has no effect on branches that are already set to display the branch name.* Also, note that selecting this menu item has no effect on the part properties, unlike when the display properties are turned ON or OFF by selecting the 'Edit all branch properties' menu item.

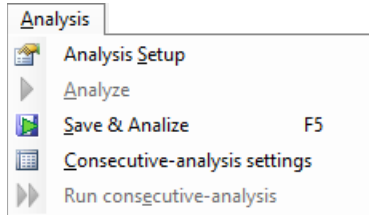


Display all node names

- displays the names of all the nodes in the currently active layer. This operation can also be performed by clicking the toolbar button indicated below.

Selecting this menu item, or clicking the corresponding toolbar button, displays all the node names. A further selection of the menu item or click of the button disables this function, restoring the node name display settings specified in the properties for each node. *Note that this function has no effect on nodes that are already set to display the node name.* Also note that selecting this menu item has no effect on the node properties, unlike when the display properties are turned ON or OFF by selecting the 'Edit all node properties' menu item.

(4) Analysis (A) menu



Analysis settings

- opens the settings screen used to set the analysis conditions. This operation can also be performed by clicking the toolbar button indicated below. Analysis processes are described in Chapter 6.

Run analysis

- performs an analysis using the previously set analysis conditions. This operation can also be performed by clicking the toolbar button indicated below. An analysis cannot be executed when the circuit has been changed. In this case, the menu item and button for this operation are inactive (grayed-out and inoperable). *You must first save the circuit before performing an analysis on it.*

Save and run analysis

- saves changes to the current circuit and then runs the analysis. You can also select from the "Save and run analysis" icon button. This function can also be operated by pressing the F5 key.

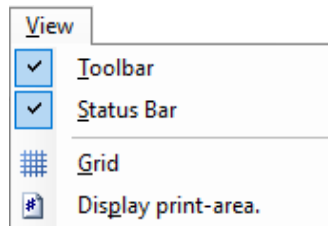
Consecutive-analysis settings

- opens the 'analysis settings' screen, used to set multiple combinations of circuit parameters such as circuit resistances, inductances, and breaker ON times (multiple analysis cases), and to execute those multiple analysis cases consecutively and automatically. This menu item is also used to define the settings for statistical switching-surge calculations. The operation of this menu item can also be executed by clicking the toolbar button indicated below. See Chapter 7 for details.

Run consecutive analysis

- performs a consecutive analysis using the previously set consecutive-analysis conditions. This operation can also be performed by clicking the toolbar button indicated below. Note that a consecutive analysis cannot be executed when the consecutive-analysis settings have not been entered or the circuit has been changed. In these cases, the menu item and button for this operation are inactive (grayed-out and inoperable). *The circuit must be saved before performing a consecutive analysis.*

(5) View (V) menu



Toolbar

- enables/disables the display of the toolbar in the XTAP main window.

Status bar

- enables/disables the display of the status bar in the XTAP main window.

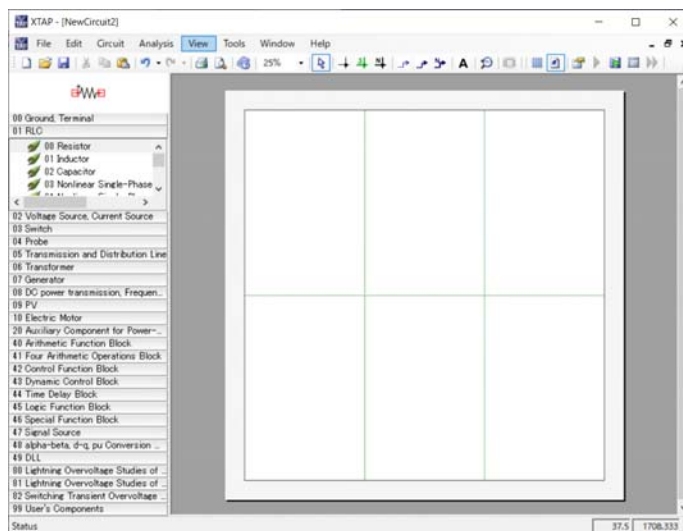
Grid display

- enables/disables the grid display in the circuit-diagram canvas. This operation can also be performed by clicking the toolbar button indicated below.

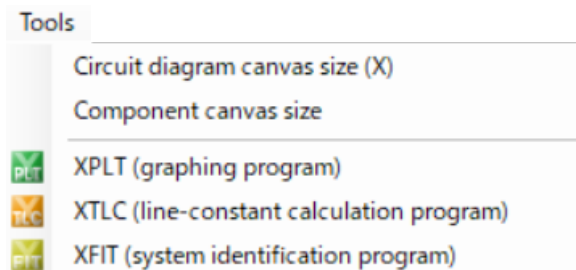
Print-area display

- enables/disables the print-area display on the circuit-diagram canvas. Enabling the print-area display shows the set print-sheet positions (the print area for each sheet) in the circuit canvas when two or more horizontal or vertical sheets have been set in the canvas-size settings. The operation of this menu item can also be performed by clicking the toolbar button indicated below.

The screen shot below shows an example of the display produced when the print-area display has been enabled, and the '3 horizontal/2 vertical' settings have been selected in the canvas-size settings (with A4 size paper). The dotted green lines indicate the print area of each sheet. Printing with these settings will result in six A4 sheets being printed.

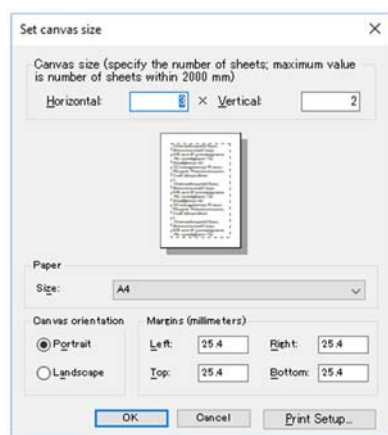


(6) Tools (T) menu



Circuit-diagram canvas size

- allows a change to the size of the circuit-diagram canvas. This size is specified in terms of the paper size supported by the selected printer driver, and the number of sheets in the horizontal and vertical directions (with a maximum of nine sheets in the horizontal direction and six sheets in the vertical direction).



Component canvas size

- changes the size of a part icon's canvas. It is possible to specify any width and height for the size of the component canvas. See Section 10.3 for details.

XPLT (graphing program)

- starts XPLT (the graphing program). XPLT is described in Chapter 9.

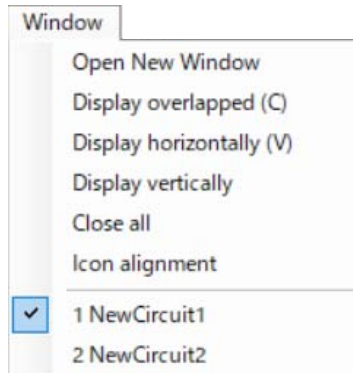
XTLC (line-constants calculation program)

- starts XTLC (the line-constants calculation program). XTLC is described in Appendix 1.

XFIT (system-identification program)

- starts XFIT (the system-identification program for XTAP). XFIT is described in Appendix 2.

(7) Window (W) menu



Opens a new window

- Opens the same xsf file that is currently open in a new window.

Cascade

- Switches the display method for the circuit diagram files to a multi-window view to cascade all circuit diagrams that are currently open.

Vertical

- Switches the display method for the circuit diagram files to a multi-window view to display all circuit diagrams that are currently open vertically at equal intervals.

Horizontal

- Switches the display method for the circuit diagram files to a multi-window view to display all circuit diagrams that are currently open horizontally at equal intervals.

Close All

- Closes all circuit diagrams that are currently open.

Arrange Icons

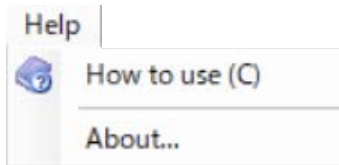
- Arranges the small windows at the bottom of the screen when the window display is minimized.

“Name of the open circuit diagram file” *May be none or multiple.

The window name of the currently-open circuit diagram file will be displayed on the list.

When editing a hierarchized component, it will be recognized as a single window. If one of the windows is selected here, the selected window will be displayed in the foreground.

(8) Help (H) menu



User's Manual

- Opens the same xsf file that is currently open in a new window.

Cascade

- Opens a PDF file of the XTAP user's manual (this document). (The file will be opened by the application associated with the PDF file.)

Version Information

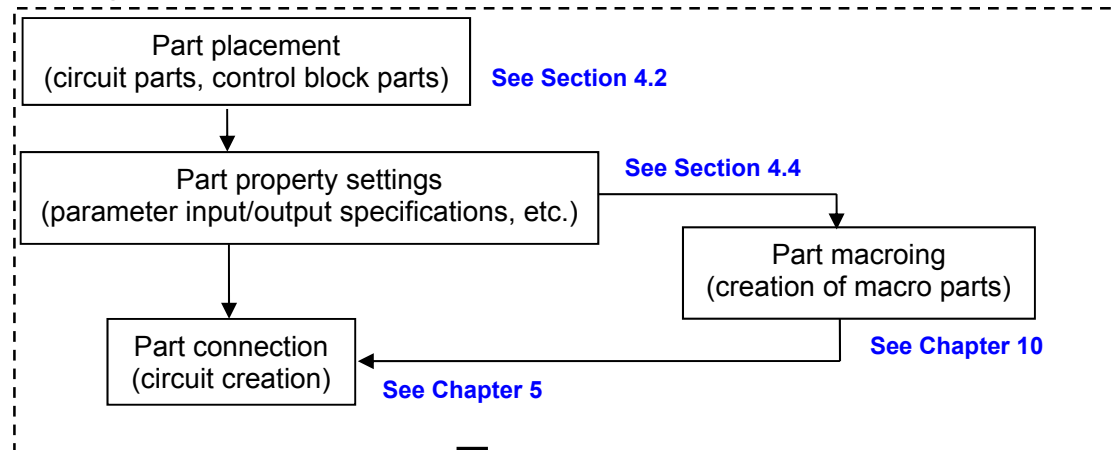
- Displays the version information of the currently activated XTAP.

3 Workflow From Circuit-Diagram Creation Through Analysis

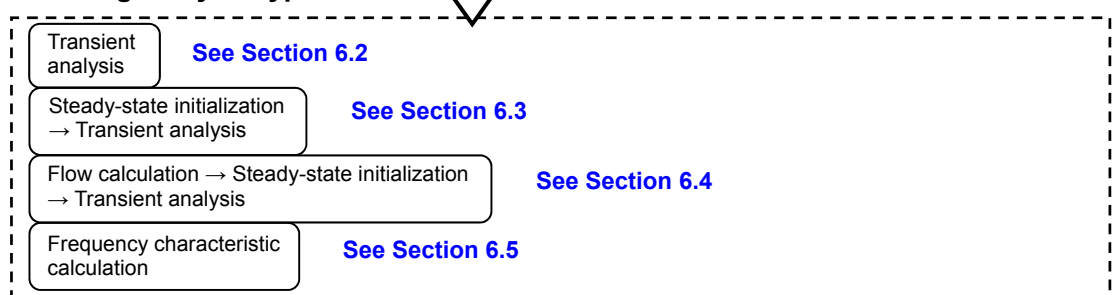
The XTAP workflow needed to create a circuit, execute its analysis, and graph the results is illustrated below. See the sections indicated in the flowchart for descriptions of the corresponding procedures.

○ Using XTAP (circuit creation/calculation program)

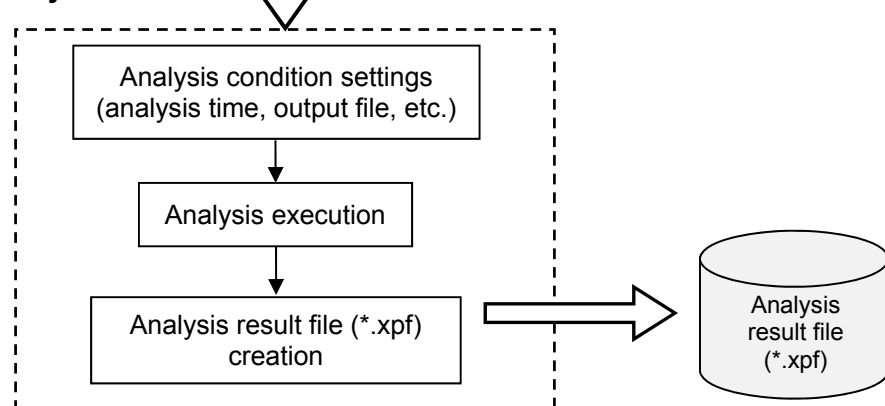
Creating circuit



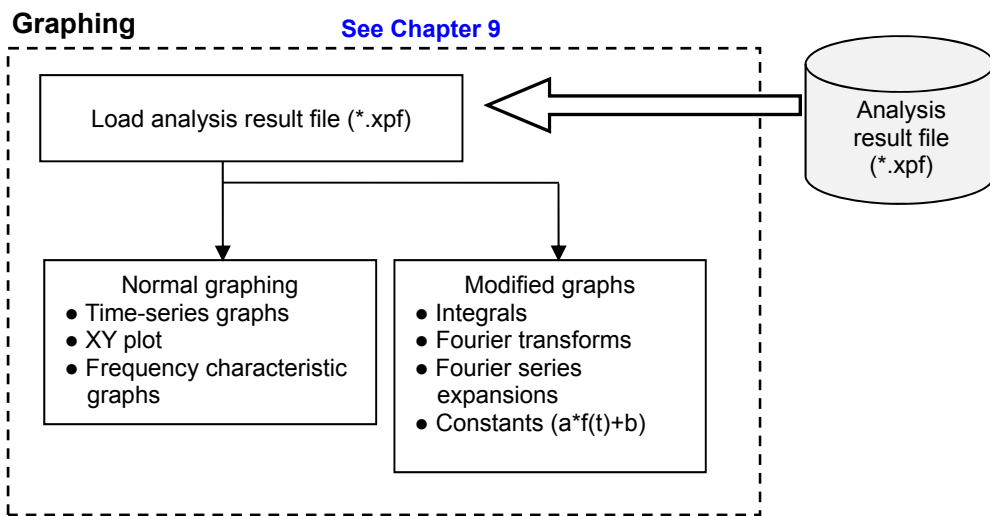
Selecting analysis type



Analysis execution



○ Using XPLT (graphing program)



4 Creating Circuit Diagrams

4.1 Part types

The parts in XTAP are of the following types:

1. Electrical-circuit parts

These parts constitute electrical circuits and include resistors, power supplies, and switches. Electrical circuit parts can be connected to each other either directly (by red square terminals), or to control parts using components such as voltage/current probes (control outputs), and controller control voltage/current sources. There are both single-phase and three-phase parts, both of which have different terminals. A voltage, current, or instantaneous power can be viewed as forming the part output.

2. Control-block parts

These parts constitute control blocks. In control blocks, signal inputs are denoted as yellow terminals, and outputs as blue terminals. A blue terminal output is connected to a yellow terminal. There are both single-signal and multiple-signal wirings, both of which have different terminals. Connections to electrical circuit parts are made using components such as voltage/current probes (control outputs) and controller control voltage/current sources. Output signals can be viewed as forming the part output.

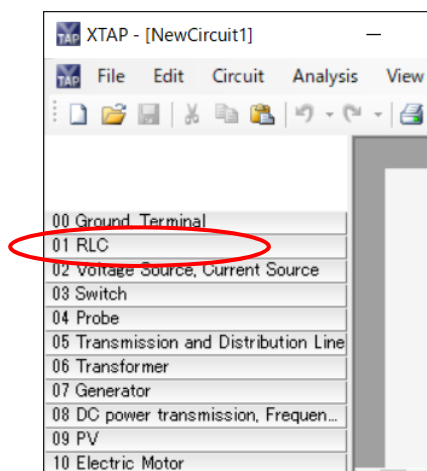
3. Macro parts

These composite parts are created by packaging circuit parts or control-block parts together. The rules for making connections to macro parts are the same as for the individual parts. A macro part can be created within another macro part. Macro parts can be referenced and edited in circuit diagrams.

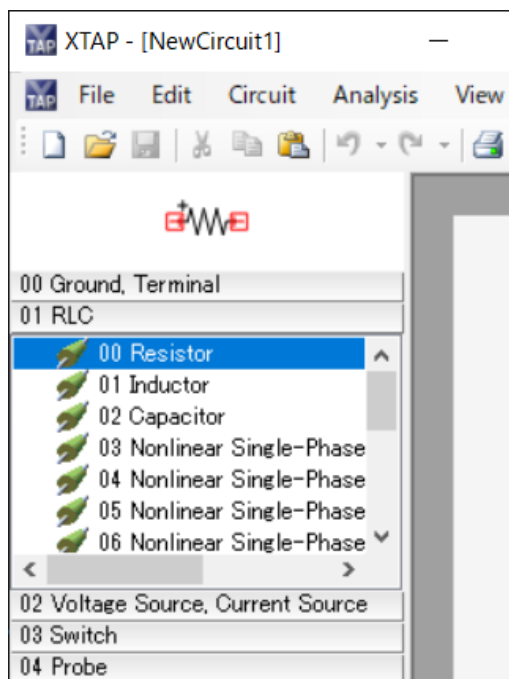
4.2 Part placement (electrical-circuit part/control-block part)

XTAP allows parts to be placed easily in a circuit diagram by 'drag and drop' using the mouse.

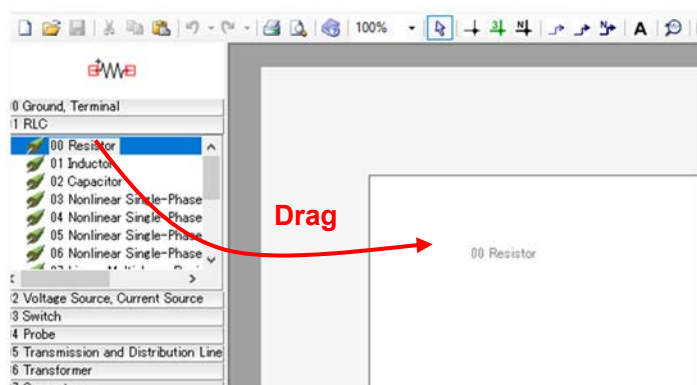
1. From the 'File' menu, select 'New (N)' → 'Create circuit diagram (X)'. In the 'part category' window on the left, select the category containing the part you want to place in the circuit diagram. In the example below, a (linear) resistor is being added, and therefore '01 RLCs' is selected.



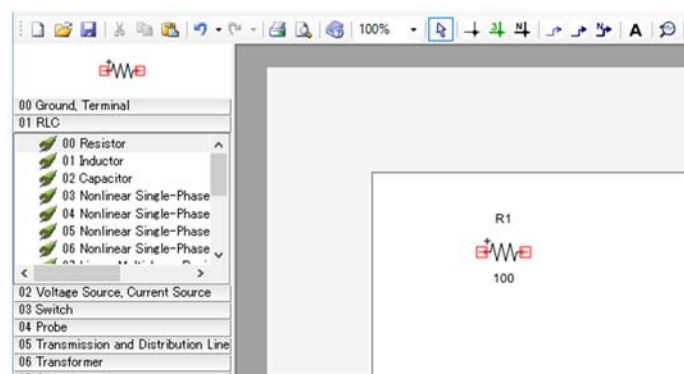
2. The parts registered in the '01 RLCs' category are displayed. Selecting one of these parts displays its icon in the upper window.



3. Select the name of the part to place, and drag the selection onto the circuit diagram. ('00 Resistor (linear)' has been selected in this example.)



4. A (linear) resistor is placed on the circuit diagram.



(1) Selecting and moving parts

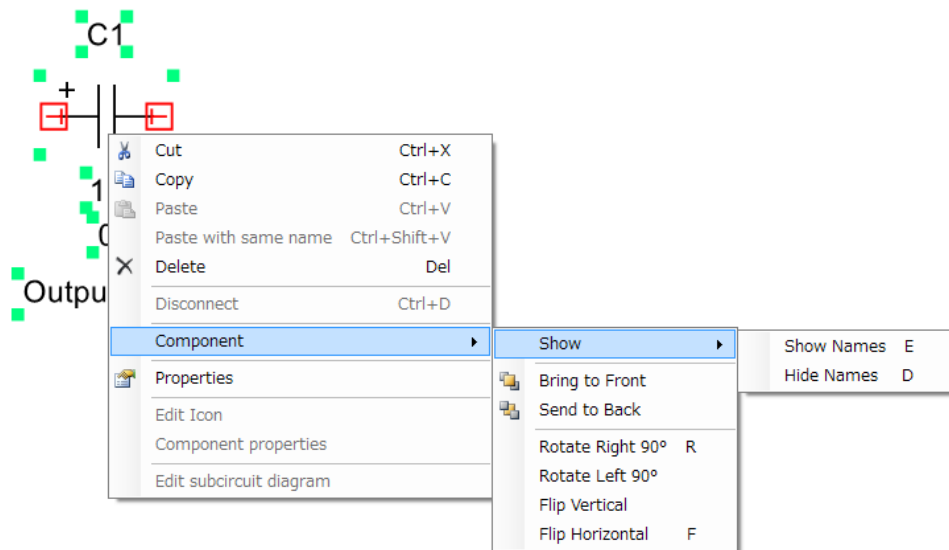
C1
1u
0
Output: V, I, P

- 1 selects the caption label (part name). A double click allows the part to be renamed directly. It is also possible to rotate the orientation by 90 degrees from 'Figure' menu that appears by right-clicking the caption label (new function added in Ver. 2.10). Text strings used as part names may contain upper- or lowercase letters, numbers, and the underscore (`_`) character, but the first character cannot be a number. *Japanese characters or other Asian-width characters may not be used in part names.*

- 26 -

(2) Part menu

The 'part' menu is displayed when right-clicking on a part.



Cut

- cuts the part.

Copy

- copies the part.

Paste

- pastes the most recently cut or copied part. (See the note on the next page.)

Paste with same name

- pastes the most recently cut or copied part together with its name. (See the note on the next page.)

Delete

- deletes the part.

Disconnect

- It is possible to cut a circuit line by right clicking the corresponding node and selecting "Disconnect".

Component - Show - Show Names

- Show the component name in the diagram.
It is possible to do the same action by selecting the component and pressing the key "E".

Component – Show – Hide names

- Hide the component name in the diagram.
It is possible to do the same action by selecting the component and pressing the key "D".

Component – Bring to front

- moves the part to the foreground plane.

Component – Send to back

- moves the part to the furthest background plane.

Component – Rotate Right 90°

- rotates the part 90 degrees clockwise. This operation can also be performed by

pressing the 'R' key on the keyboard when the part is selected.

Component – Rotate Left 90°

- rotates the part 90 degrees counterclockwise.

Component – Flip vertically

- flips the part vertically.

Component – Flip horizontally

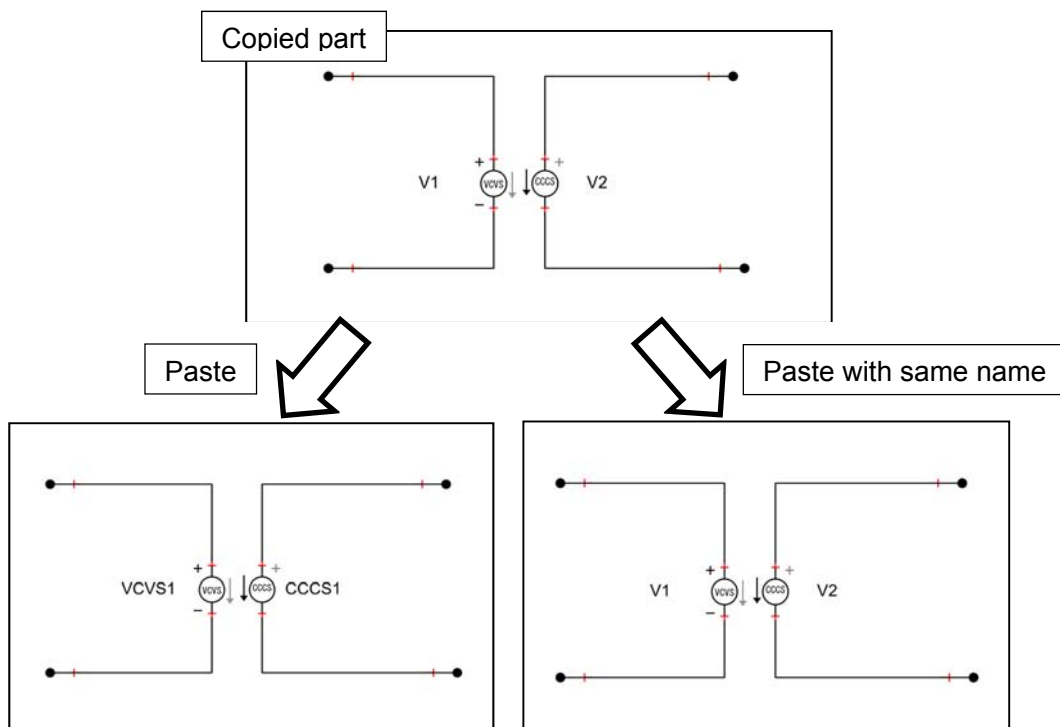
- flips the part horizontally. This operation can also be performed by pressing the 'R' key on the keyboard when the part is selected.

Properties

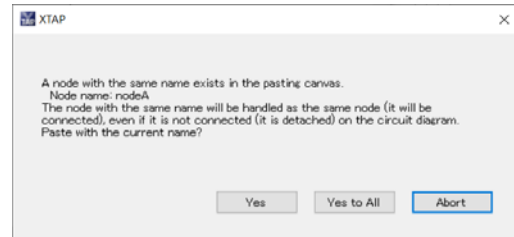
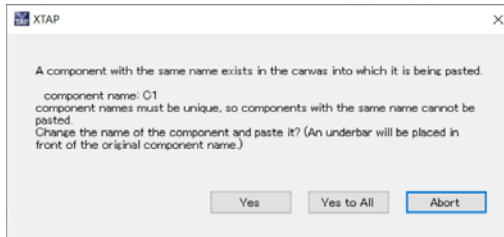
- displays the 'part properties' screen.

• Difference between 'Paste' and 'Paste with same name' in part menu

Every part name in XTAP must be unique (no two part names can be the same). Therefore, when a part is copied and pasted, the name of the pasted part is the default part name appended with a number. However, when 'Paste with same name' is selected, the pasted part keeps the name of the original part provided that no part with that name already exists on the target canvas.

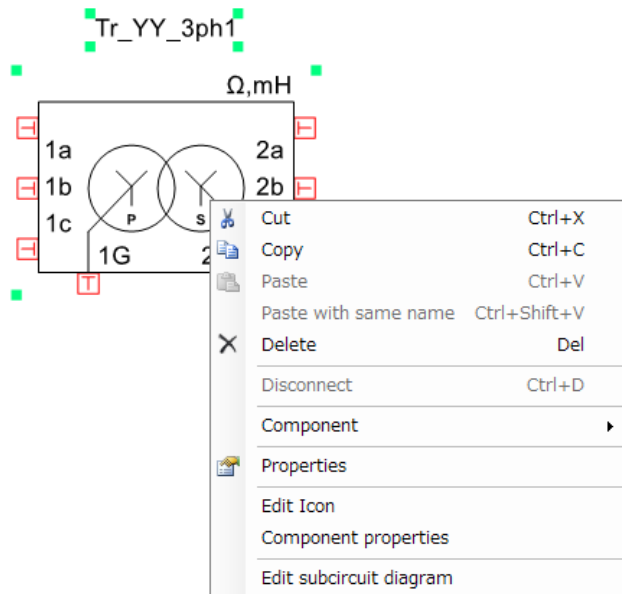


If a part with the same name already exists on the destination canvas, the message shown below on the left appears, asking for confirmation to execute the paste operation. (If the operation is executed, the part name becomes appended with an underscore character.) Unlike parts, nodes with the same name may coexist on the same canvas. However, in this case, *program calculations will handle those nodes as if they were the same node, even if they are at different locations on the circuit diagram (see Section 5.1)*. When executing the 'Paste with same name' command and if a node of the same name already exists on the canvas, the message below on the right then appears.



(3) Macro part menu

In addition to the part menu items previously described, the following additional part menu items can also be selected for macro parts. See Chapter 10 for details of how to create and edit macro parts. Macro parts are created by selecting portions of the circuit diagram and packaging them as a new composite part.



Edit icon

- displays a screen for editing the part's icon. See Section 10.3 for details of how to edit icons.

Component properties

- allows the specification of part information.

Edit packaged diagram

- displays the part's circuit diagram. See Section 10.2 for details of how to change macro parts, and Section 10.4 for how to set connection terminals.

(4) Component manipulation by using the keyboard/mouse

Besides moving components placed in the diagram by dragging with the mouse, it is possible to move components by selecting it and pressing the keyboard direction keys.

Moreover, by pressing specific keys and selecting components, it is possible to selecting several components at once, to copy them or to move them horizontally/vertically.

(Ctrl key) + (Left click)

- It is possible to select several components at once. It shows the same behavior of (Shift key) + (Left click)

(Shift key) + (Left click)

- It is possible to select several components at once. It shows the same behavior of (Ctrl key) + (Left click).

(Ctrl key) + (drag and drop)

- Copy the selected component. It can be used together with the shift key for moving the component horizontally/vertically.

(Shift key) + (drag and drop)

- Move the selected component horizontally or vertically only. It can be used together with the ctrl key for copying the component.

4.4 Part properties screen

A double click on a part displays the properties screen for that part.

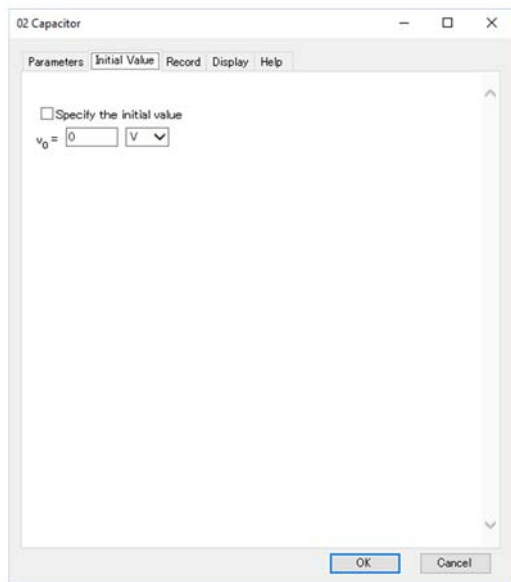
(1) Parameters

Parameters are the properties that define the calculation conditions. Information about a part's parameters is provided in the help tab of its respective 'properties' screen. Some control blocks have no specifiable parameters.



(2) Initial value

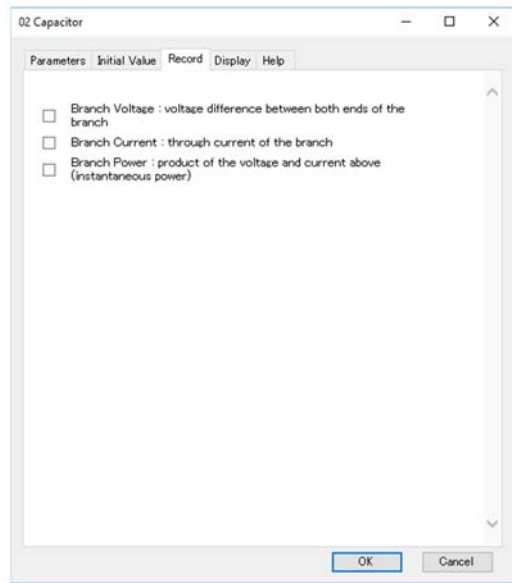
An initial value can be set for some parts. The way to set an initial value is described in the help tab of the 'properties' screen.



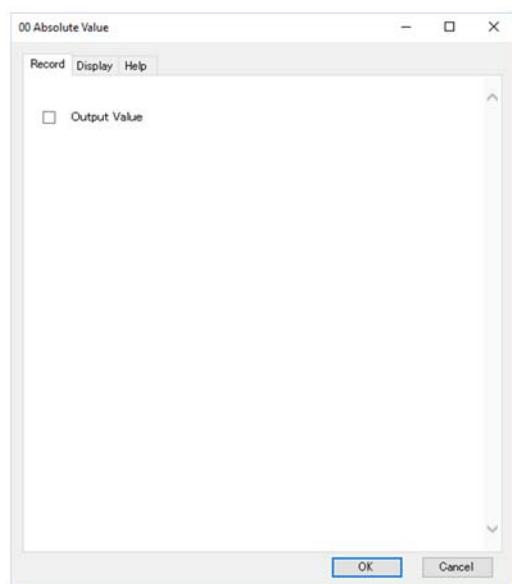
(3) Recording

- used to specify the items from which to generate outputs to graph. The checked items will be displayed by XPLT (the graphing program). XPLT is described further in Chapter 9. Even when executing an analysis, only the checked items will be output as waveforms. The appropriate checkboxes must therefore be checked to obtain the part voltage, current, or power observations, as required. The following three types of values can be output for circuit parts:

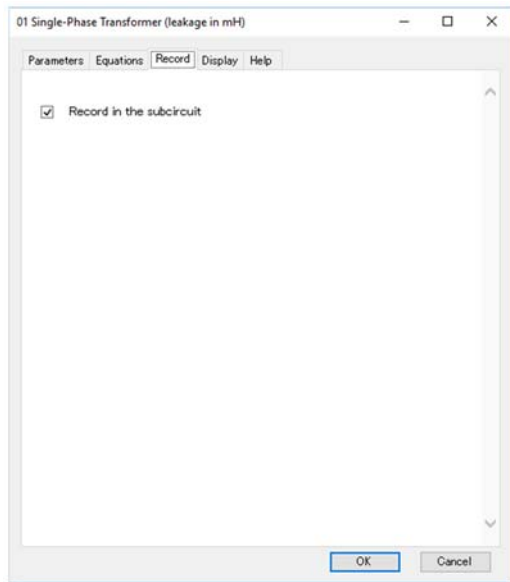
Branch voltage
Branch current
Branch power



In the case of control blocks, the output signal can be an output.

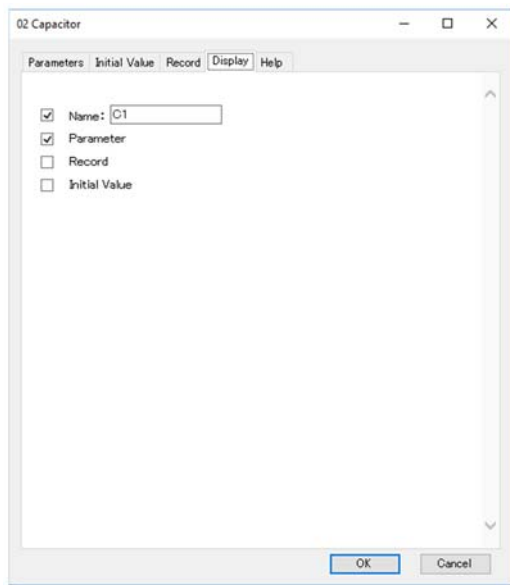


In macro parts, their internal parts, and the existence of various signals of nodes (node voltage, branch voltage, branch current, branch power, and control signal) can be specified. When a macro part is turned off, even if any part's signal recording is on inside that macro part, output is still disabled.



(4) Display

- enables or disables the display of various display labels for a part on the circuit-diagram canvas.



Name

The part name can be entered in the text box. Checking the checkbox displays the part's caption label on the circuit-diagram canvas. In some power transmission line components, the line length can be displayed.

Parameter

Checking this checkbox displays the part's parameter label on the circuit diagram canvas. The parameter displayed is determined by the part type.

Recording

Checking this checkbox displays the items selected in the Recording tab as the part's output label on the circuit-diagram canvas. In the example below, the branch voltage and branch current have been selected in the Recording tab.



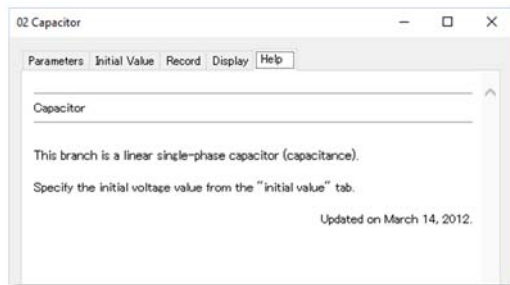
Output:V,I

Initial value

Checking this checkbox displays the part's initial-value label on the circuit diagram canvas. The initial value displayed is determined by the part type.

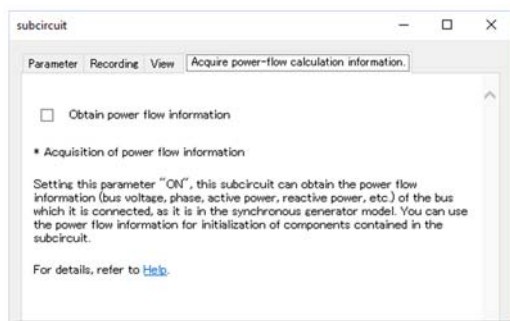
(5) Help

- displays help information for a specific part.



(6) Acquiring information from power-flow calculations (macro parts only)

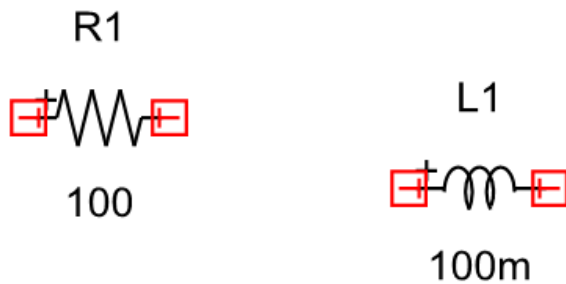
User-created macro parts have a tab for power-flow calculation information. The information in this tab is described in Chapter 8.



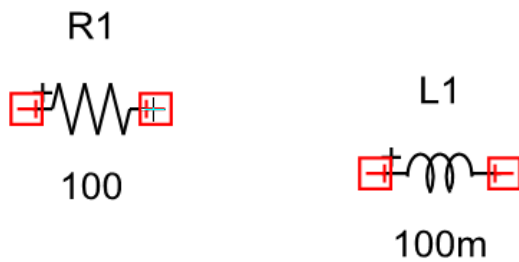
5 Part Connections

5.1 Creating electrical-circuit wires

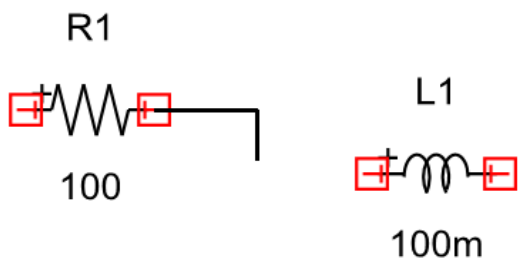
This section describes how to create the electrical-circuit wires (nodes) that connect different parts. As an example, a (linear) resistor and a (linear) inductor are first placed on the circuit diagram canvas as described in Section 4.2.



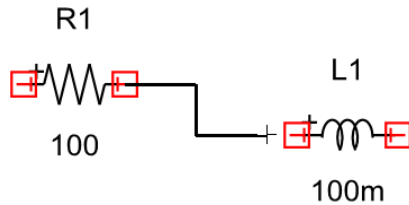
1. Click the right terminal of the resistor. The mouse cursor becomes a plus sign (+), indicating the program has switched from part-selection mode to the mode used for automatically creating electrical-circuit wires. To switch to this mode manually, select 'Electrical circuit wire' from the 'Circuit' menu.



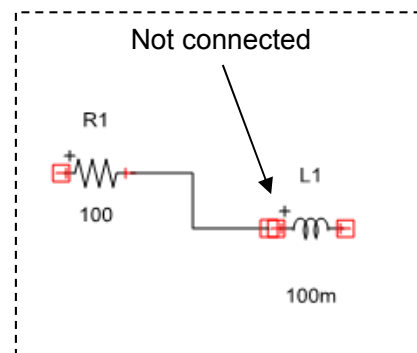
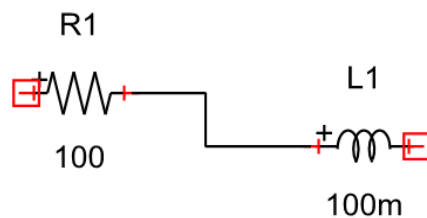
2. Move the mouse to draw a line. In this example, the mouse is moved toward the left terminal of the inductor. Lines cannot be drawn diagonally.



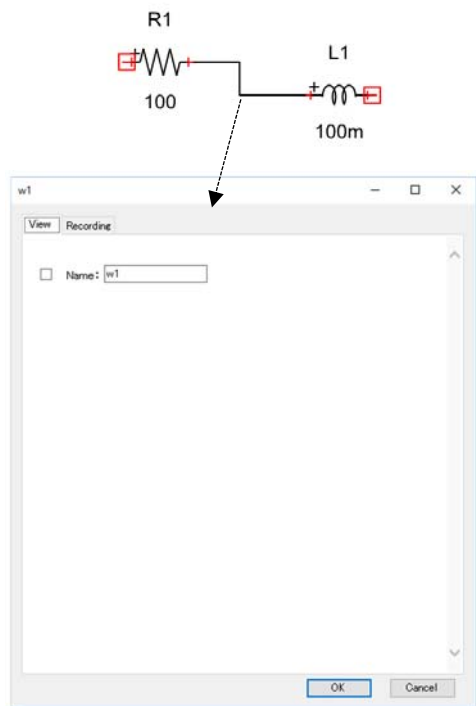
3. Change the direction of the line by clicking at the desired point of change. In the example below, the line has been drawn horizontally, vertically, and then horizontally again.



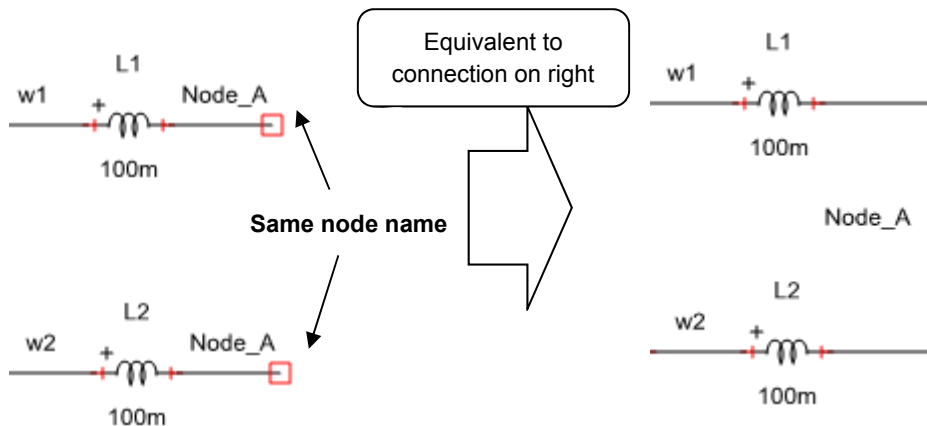
4. Move the mouse to the left terminal of the inductor, then double-click to make the connection. Once the terminals are connected, their red squares disappear. If the terminal has a horizontal orientation, the line also needs to be connected horizontally. Horizontally oriented terminals cannot be connected from a vertical direction.



5. When an electrical circuit wire (node) is created, the program automatically assigns it a unique name. Node names are not displayed under default display settings. To check or change a node name, double-click the wire to open the node's 'properties' screen. See Section 5.2 for information on node properties.



6. The node name can be changed in the node's 'properties' screen (see Section 5.2). If the assigned node name already exists in the same layer, the nodes of that name will be considered to be the same node, and will be handled as if they were connected, even if they are not connected in the canvas wiring.



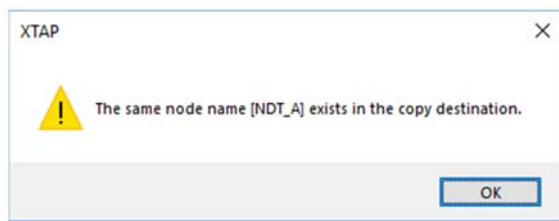
Whether or not a node name has been changed affects the node-name setting when the node is copied/pasted.

○ When an unrenamed node is copied/pasted

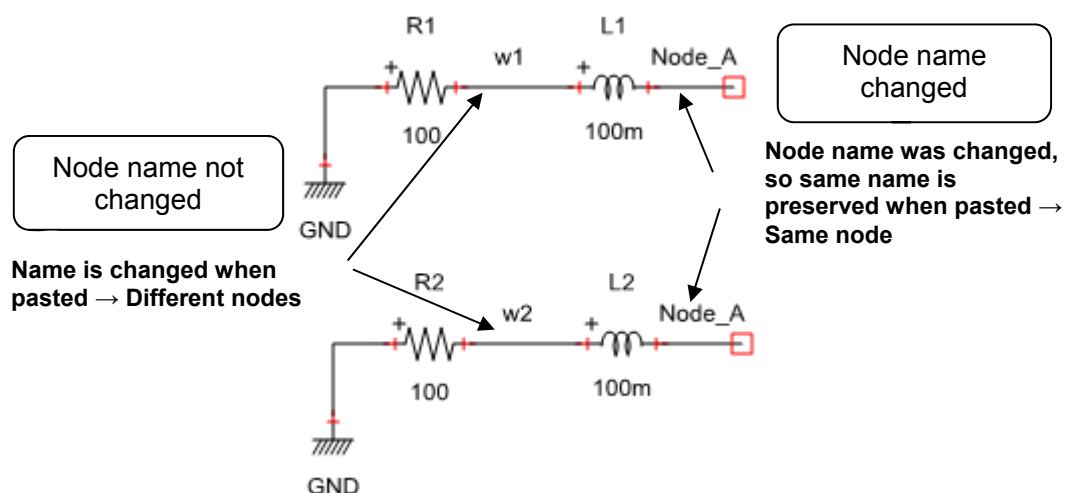
The name of the pasted node is changed to a different unique node name, and the node is handled as a different node.

○ When a renamed node is copied/pasted

The pasted node preserves the changed node name, resulting in the existence of multiple nodes with the same name. In this case, the following message appears.



As when changing a node name, care is needed since nodes with the same name are treated as the same node.



5.2 Electrical-circuit wire (node) properties

Double-clicking a node displays its 'properties' screen.

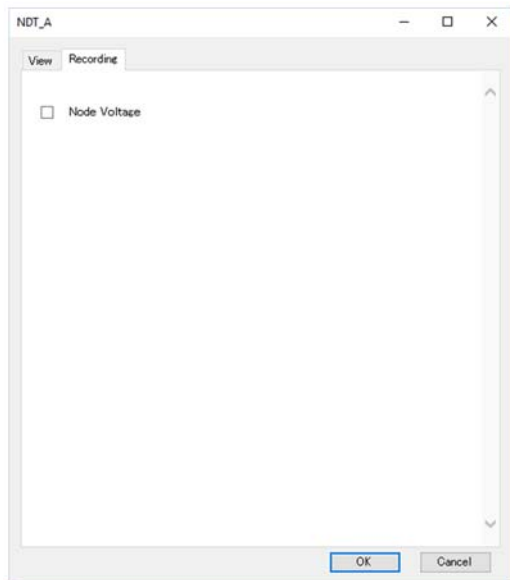
(1) Display

This tab enables or disables the display of the electrical circuit wire name on the circuit-diagram canvas. Checking the checkbox displays the node name on the circuit-diagram canvas. The text box can be used to change the node name.



(2) Recording

Checking the checkbox in this tab sets XPLT (the graphing program) to display the node voltage (voltage to ground). XPLT is described in Chapter 9.

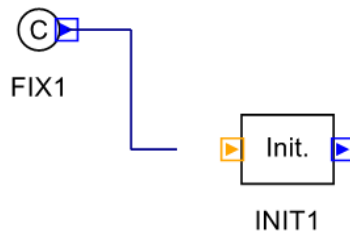


5.3 Creating control-system wires

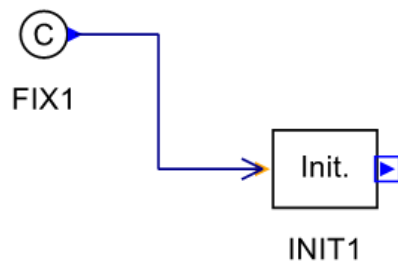
Unlike electrical-circuit parts, control system parts have input terminals (colored orange) and output terminals (blue). Input terminals must always be connected to output terminals, and output terminals to input terminals. Control system parts are connected in the same way as electrical-circuit parts. As an example, place a 'constant' part (40 Control function block) and an 'initial value' part (42 Control function block) onto the canvas.



1. Click the (blue) output terminal of the constant part (FIX1). The mouse cursor then changes to a plus sign (+), indicating the program has switched from 'part selection' mode to the mode for automatically creating control system wires. (To switch to this mode manually, select 'Control system wire' from the 'Circuit' menu.) Connect the control system wire to the (orange) input terminal of the initial value part (INIT1) in the same way as when making a node connection.



2. Once the wire has been connected correctly, the blue and orange boxes indicating the terminals disappear. An arrow indicating the wiring direction is displayed at the input terminal.



5.4 Control system wire properties

Double-clicking on a wire displays its 'properties' screen.

Display

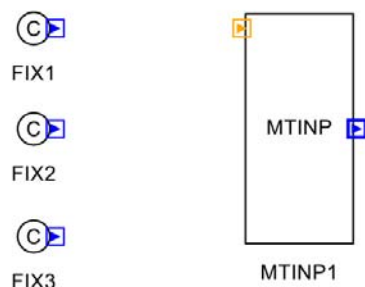
This screen is used to enable or disable the display of the control-system wire name on the circuit-diagram canvas. Checking the checkbox in this tab displays the name of the wire name on the canvas. Since the control-system wiring represents the control-block connections, the wires have no associated values, and therefore do not have a 'Recording' tab.



5.5 Creating multiple-signal control system wirings



This section describes multiple-signal control system wirings (vectors) that group together multiple control signals. Basic operations are same as for a single-signal control system wiring.

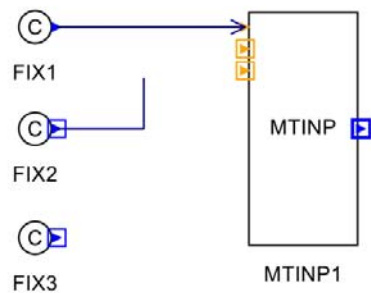
First, create a vector signal including three elements from three single signals (conventional control signals). Place any three single signals on each (in this case, 'constant' [42 Control function block] and 'vector creation' [46 Special function block]).





1. Next, double-click the part 'vector creation (MTINP1)' to open the properties screen, and change the number of elements to 3.



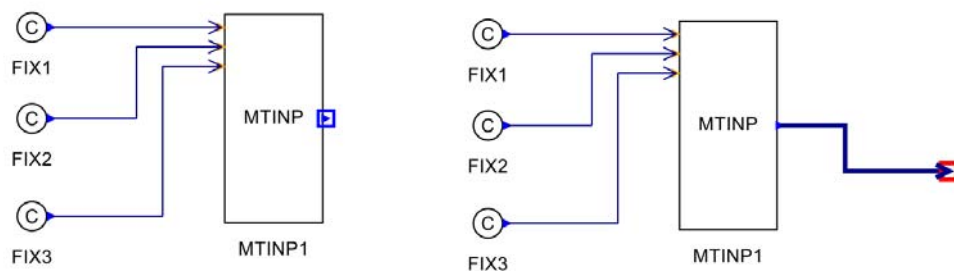
- Next, click the constant (FIX1) output terminal (blue). The mouse cursor will turn into a '+', and the mode will automatically change from the part selection mode to the control system wiring creation mode. After this, wire the control system wiring to the 'vector creation (MTINP1)' input terminal (orange) in the same way as for the node connection. Once the wiring is properly done, the   of the connected terminal will disappear, and an arrow indicating the direction of wiring will appear on the input-side terminal.



- The wiring is completed once all of the single signals are connected. The output of the part 'vector creation (MTINP1)' is the vector signal. As shown in the diagram below, the signal lines have different thicknesses.

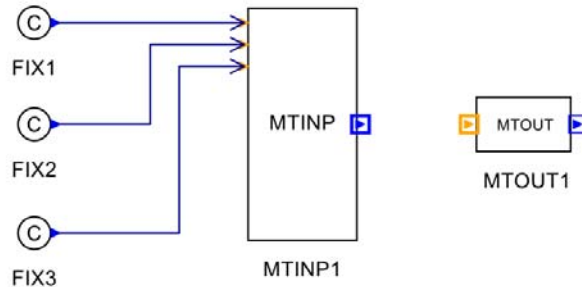
The vector signal line can only be connected to the vector signal line's terminal (the thick  .

Also note that parts that can handle the vector signal have "vector" at the front of their part names.

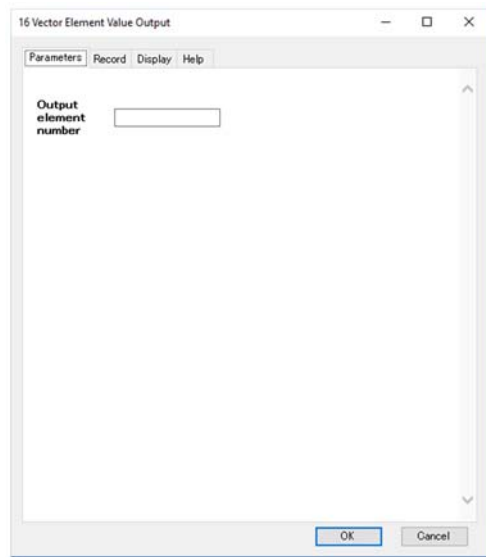


- To only isolate one element from the vector signal line, use 'Vector element value output' (46 Special function function block). Here we will isolate the second signal from the previously created vector signals.

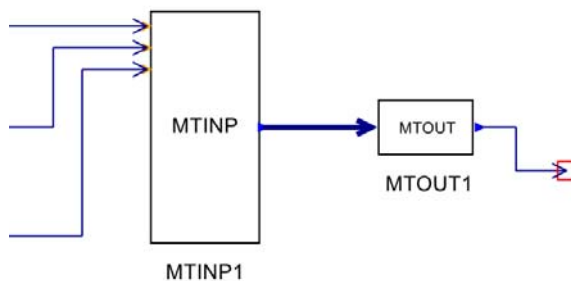
First, place 'vector element value output' on the canvas.



- Next, double-click the part 'Vector element value output (MTOUT1)' to open its property screen, and input 2 (half-width digit) as the output element number.

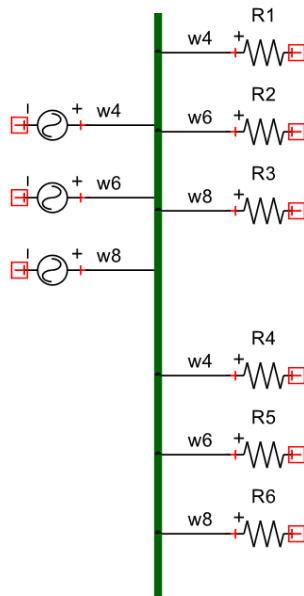


- Finally, connect 'Vector creation (MTIN1)' and 'Vector element value output (MTOUT1)'. The output of MTOUT1 becomes a single signal.



5.6 Creating three-phase buses

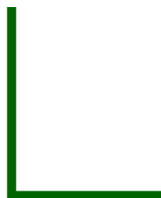
A three-phase bus enables three phases of electrical circuit wiring to be connected to a single wire (bus).



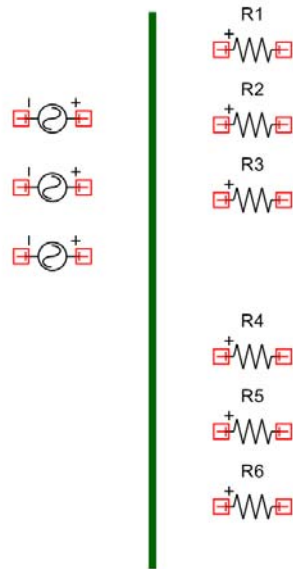
1. Select 'Three-phase bus' from the 'Circuit' menu in the main window. The mouse cursor changes to a plus sign (+), indicating that the program has entered the mode for creating a three-phase bus.
2. Click the circuit-diagram canvas and move the mouse straight downward. A line is drawn, thicker than a connection line.



3. To change the drawing direction from vertical to horizontal, click at the desired location. The bus changes direction at the click location.



4. The example below describes how to connect a three-phase power source and power-transmission line with two parallel lines to a single bus. For simplicity, the transmission line is implemented as resistors.



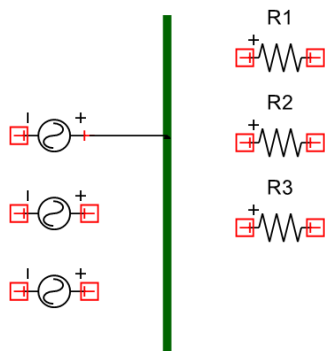
02 Sine wave voltage source

From the category '02 Sine wave voltage sources – 00 Independent voltage source/current sources' in the 'part category' box, select and place a '02 Sine wave voltage source'. Create three of these parts on the left side of the bus. From the part menu, select 'Figure – Rotate 90 degrees right' to rotate the parts.

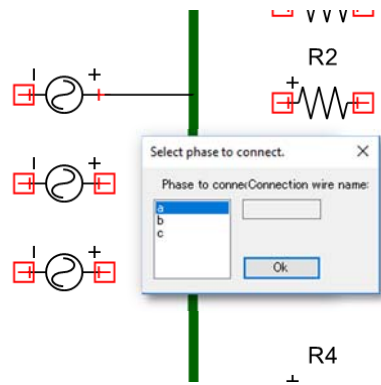
00 Resistor (linear)

From the category '01 RLCs' in the 'part category' box, select and place '00 Resistor (linear)'. Create six of these parts on the right side of the bus.

5. Connect the '02 Sine wave voltage source' part and the bus. Perform the same operation for creating a wire to connect the right terminal of the '02 Sine wave voltage source' part to the bus.



6. Move the mouse cursor to the bus and double-click.
7. If the connection is successful, a window for selecting the wire name appears.



[Wire name:]

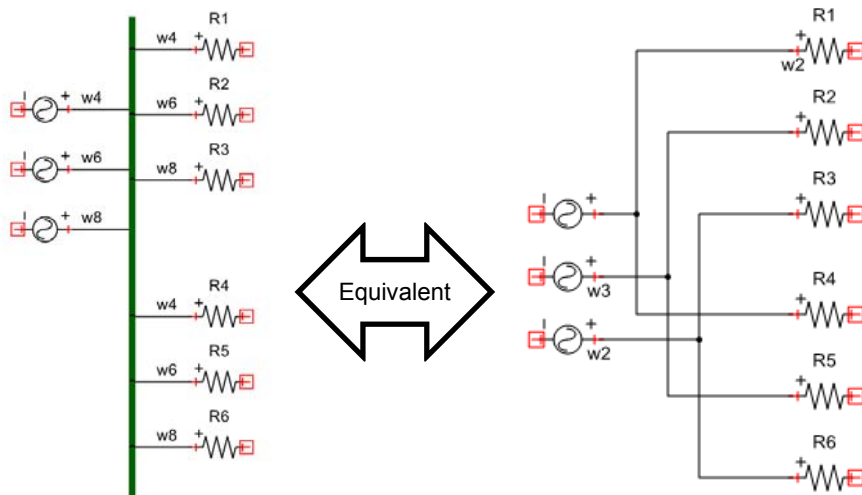
The letters 'a', 'b' and 'c' indicate phases a, b and c. Select which phase to connect the wire to.

[Connection wire name:]

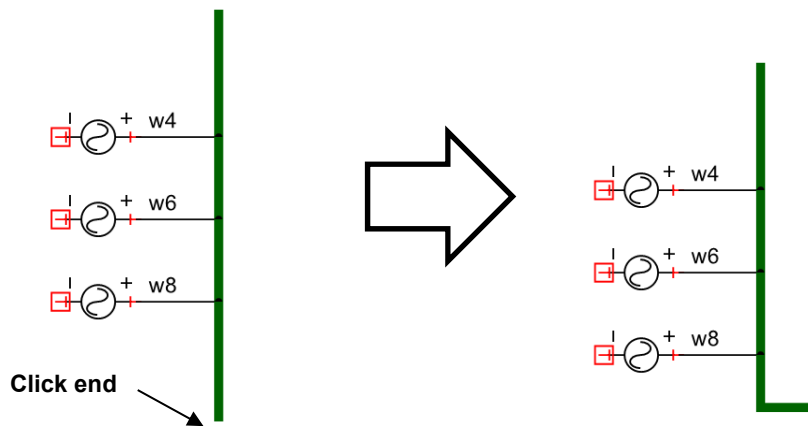
This field displays the connection wire name. 'Wire name' shows the phase, while 'Connection wire name' shows the name of the wire actually connected.

8. To connect to phase a, select 'a' as the wire name and click 'OK'. The program automatically assigns the connection wire name. This name is displayed to identify the wire. To erase the name, uncheck the [Display] checkbox in the node properties.
9. Repeat Steps 6 and 7 to connect the second '02 Sine wave voltage source' part to the bus.
10. If the connection is successful, the window for selecting the wire name appears. To connect to phase b, select 'b' as the wire name. If phase a is selected, it will be shown to be already connected to a different wire.
11. Repeat the above operation to connect the '00 Resistor (linear)' parts to the bus. When R1 is connected to the three-phase bus, the window for selecting the wire name appears. In the example below, 'a' is selected to connect it to phase a.
12. Repeat the above steps for the other resistors, connecting R4 to phase a, R2 and R5 to phase b, and R3 and R6 to phase c.

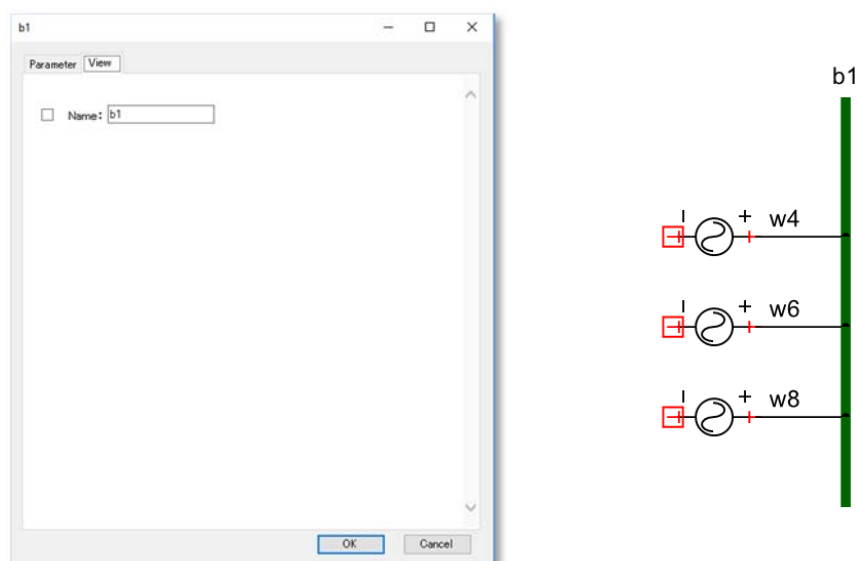
The two circuit diagrams below depict equivalent circuits.



13. To change the length of a previously created three-phase bus, switch to the 'three-phase bus' mode, and click on one end of the bus. Drag the mouse in the desired direction to extend the bus to the desired length. One bend in the bus is deleted at each right-click of the mouse.

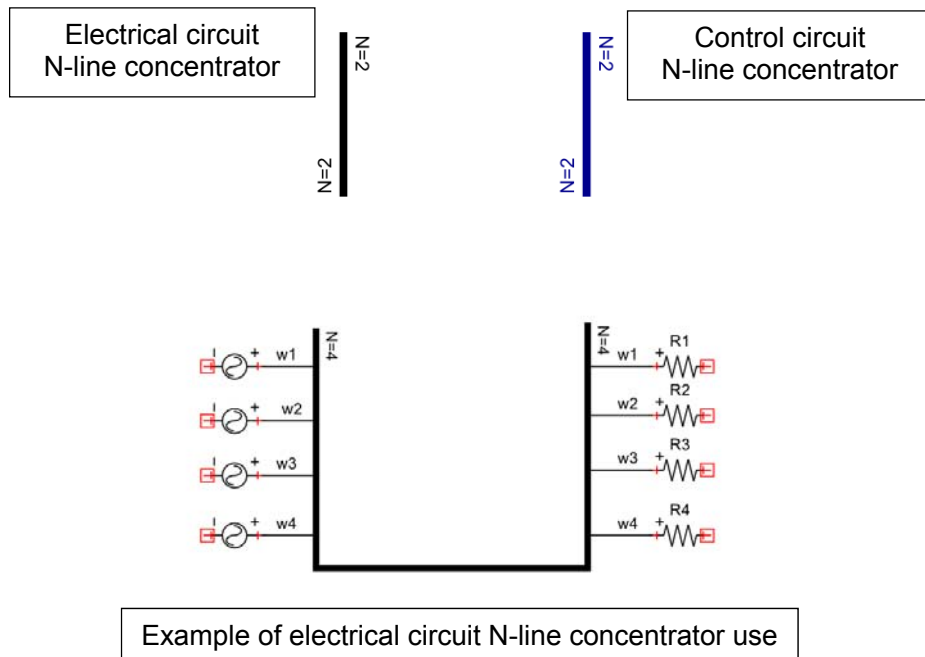


14. Buses are also assigned unique names. To display the name of a bus, double-click on the bus to display its properties. Check the checkbox in the [Display] tab to display its name. When performing a power-flow calculation, the bus name is the name of the connection node used for the parts that appear in the calculation.



5.7 Creating line concentrators

Line concentrators allow the drawing of multiple connection lines as a single line. There are two types of line concentrators: electrical circuits are displayed as thick black lines, and control systems are displayed as thick blue lines (in versions up to Ver. 2.01, electrical circuit N-line concentrators were thick blue lines, and control circuit N-line concentrators were thick green lines). Also, the number of signals included in a wiring is shown at that wiring's edge ('N = *'). The required line-concentrator type depends on the part type, but the connection method and operations are the same. *Also note that when calculating power flow, it is necessary to use a "three-phase bus," not an "electrical circuit N-line concentrator."*



1. Select 'Electrical circuit N-line concentrator' from the 'Circuit' menu in the main window. The mouse cursor changes to a plus sign (+), indicating the program has switched to the 'electrical circuit N-line concentrator' mode.
2. Click on the circuit-diagram canvas, and move the mouse straight downward. A line is drawn, thicker than an electric circuit wire.



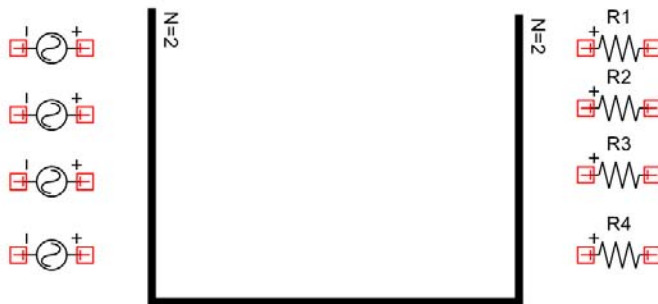
3. Click the mouse to change the drawing direction from vertical to horizontal. Then, move the mouse cursor to the right.



4. Click the mouse again to change the drawing direction back to vertical. Move the mouse upwards. Double-click to complete the line concentrator. After the line is placed, 'N = 2' will appear on the edge of the line as the number of line concentrators (two by default).



5. Place parts as shown below.



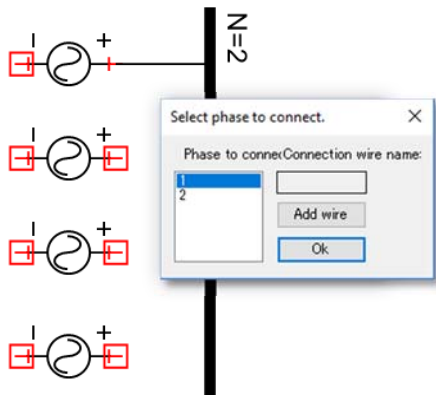
02 Sine wave voltage source

From the category '02 Voltage source/current sources – 02 Independent voltage source/current sources' in the 'part category' box, select and place an '02 Sine wave voltage source'. Create and place four of these parts on the left of the line concentrator. From the part menu, select 'Figure – Rotate 90 degrees right' to rotate the parts.

00 Resistor (linear)

From the category '01 RLCs' in the 'part category' box, select and place '00 Resistor (linear)'. Create and place four of these parts on the right of the line concentrator.

6. Connect the '02 Sine wave voltage source' part and the line concentrator. Perform the same operation used when creating a wire, to make a connection from the right terminal of the '02 Sine wave voltage source' part to the line concentrator.
7. Move the mouse cursor to the line concentrator and double-click.
8. If the connection is successful, a window for selecting the wire name appears.

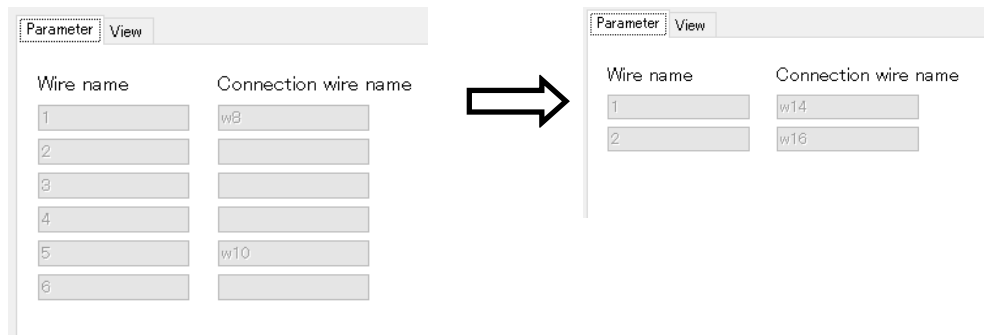
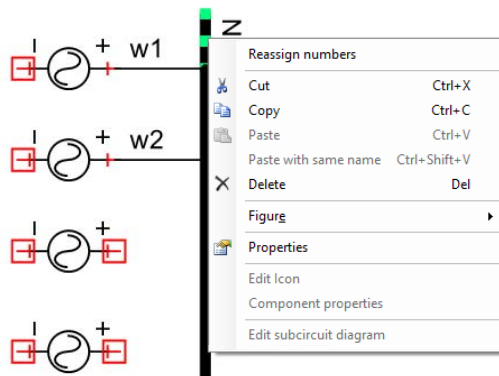


Wire name

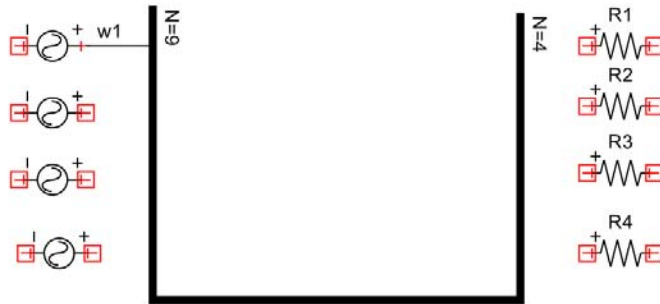
Select the names of the wires controlled by the line concentrator. A wire name is added each time the 'Add wire' button is clicked. Since the purpose is to make an N-phase connection, the wire names will be 1, 2, 3, ...N. To delete unneeded wire names, right-click on the line concentrator and select 'Reassign numbers'.

Connection wire name

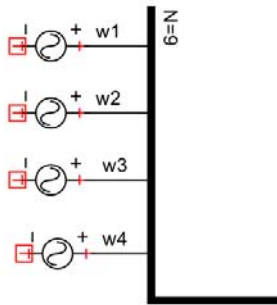
This command displays the name of the connection wire. 'Wire name' shows the phases, while 'Connection wire name' shows the name of the wire actually connected.



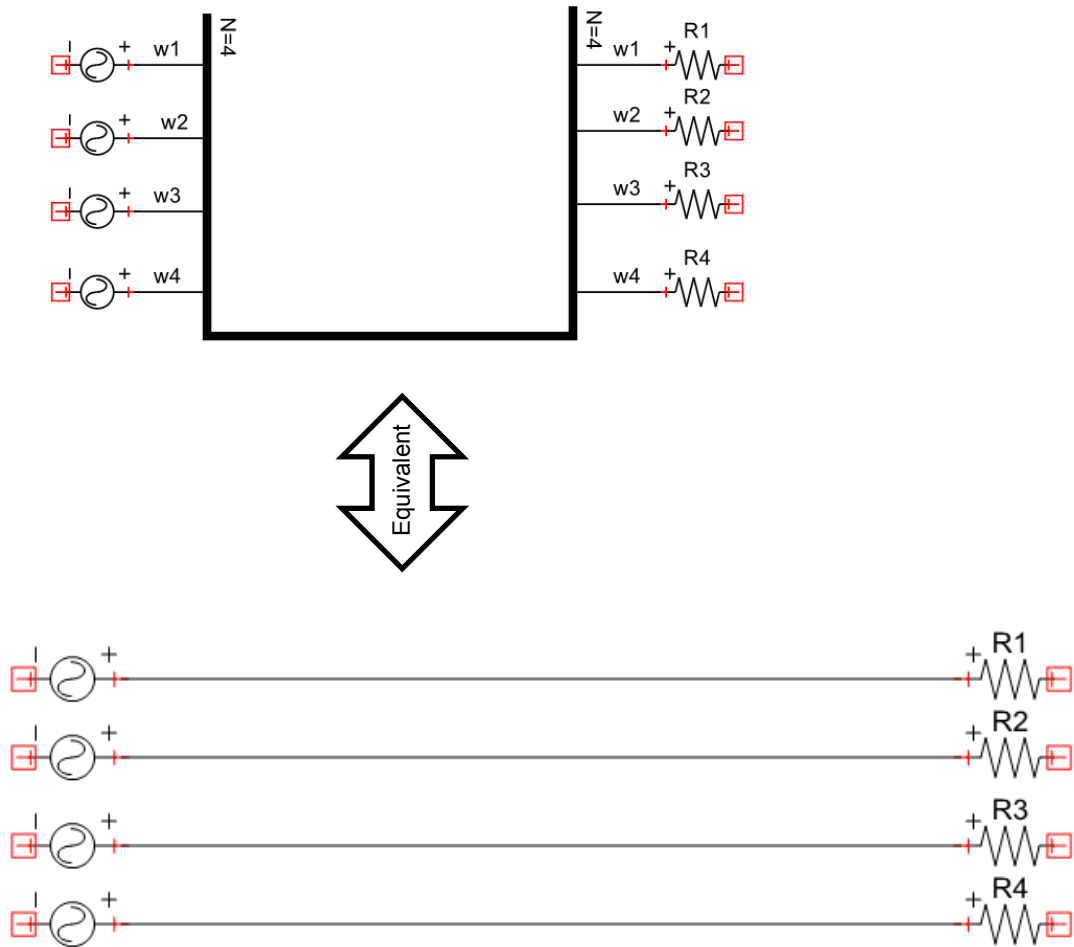
9. To connect to the first node, select '1' as the wire name, and click 'OK'. The connection-wire name is displayed, identifying the wire. To erase the name, uncheck the [Display] checkbox in the properties.



10. Repeat Steps 6 and 7 to connect the second '02 Sine wave voltage source' part to the line concentrator.
11. If the connection is successful, a window for selecting the wire name appears. To connect to wire 2, select '2' as the wire name.
12. Repeat Step 10 to connect the remaining '02 Sine wave voltage source' parts to the line concentrator.



13. Repeat the above operation to connect the '00 Resistor (linear)' parts to the line concentrator. Connect the '00 Resistor (linear)' parts R1, R2, R3, and R4 to 1, 2, 3, and 4, respectively. The two circuit diagrams below are equivalent.



6 Analysis Processes

XTAP performs transient analysis (instantaneous-value analysis) and has a steady-state initialization function to enable the transient analysis to start from the circuit's steady-state condition. Either of two methods can be used for steady-state initialization: one defines the initial values of the dynamic-circuit elements by calculating the AC steady-state solutions; in the second, the initial values are set using a power-flow calculation to determine the steady-state condition of the generator and loads, and hence use the result to calculate the AC steady-state solutions.

There are four types of XTAP analyses:

- ① Transient analysis
- ② Steady-state initialization → Transient analysis
- ③ Power-flow calculation → Steady-state initialization → Transient analysis
- ④ Frequency-characteristic calculation (frequency scan)

Each type of analysis is described below.

① Transient analysis

This performs a standard analysis of instantaneous values, starting from a state where the power-source voltage or current is zero.

② Steady-state initialization → Transient analysis

The program calculates the AC steady-state solution, considering the unbalanced circuit conditions. Using these results as initial values, the analysis starts from the steady-state operation condition. This type of analysis specifies a negative value for the 'Start time' parameter for the sine and cosine wave voltage/current sources, to include them in the AC steady-state calculation. Power sources with waveforms other than sine or cosine are ignored in the AC steady-state calculation.

③ Power-flow calculation → Steady-state initialization → Transient analysis

The program uses a power-flow calculation (the 'L method') to determine the steady-state solutions, ignoring harmonics or unbalanced circuit conditions. The solutions are used to analyze the AC steady-state solution to find approximations, considering unbalanced circuit conditions. These results are then used as the initial values for analysis away from the steady-state operation condition. 'Power-flow calculation auxiliary parts' are used to specify the power-flow conditions.

④ Frequency-characteristic calculation (frequency scan)

Analysis types ① to ③ are ultimately time-domain calculations (transient analyses). This calculation returns results in the frequency domain. The frequency f is varied within a specified range, and an AC steady-state calculation is performed repeatedly to determine the corresponding change in value. This type of analysis specifies a negative value for the 'Start time' parameter for sine and cosine wave voltage/current sources, so that these power sources may be considered in the AC steady-state calculation. Power sources with waveforms other than sine or cosine are ignored in the AC steady-state calculation.

6.1 Analysis settings

Conditions must be set to perform analyses. To do this, select 'Analysis settings' from the 'Analysis' menu in the main window. The method for setting analysis conditions is described in the procedure relevant to each analysis process.

Analysis Setup

✓ **1 Analysis-file save location**
Analysis-file (.xtf file) PE-01.xtf

✓ **2 Analysis-type selection**
Analysis-type (Analysis) Transient analysis

✓ **3 Analysis conditions**
Case name (name) Case 1
Calculation time increment (Delta_t) 1.0
Units μs
Calculation end time (T_max) 0.04
Units s
BNR switching iteration count (NumBNR) 30
Katzenelson switching iteration count (NumKa) 60
Calculation result output file (BinFile) PE-01.xpf
Output start time (TimeWindowBegin) 0.02
Units s
Output end time (TimeWindowEnd) T_max
Units s
Output frequency (OutputFreq) 1
Output calculation result in text format/CSV Yes (text format)
CSV format output file (CSV_File) PE-01.csv
Debug level (DebugLevel) 0

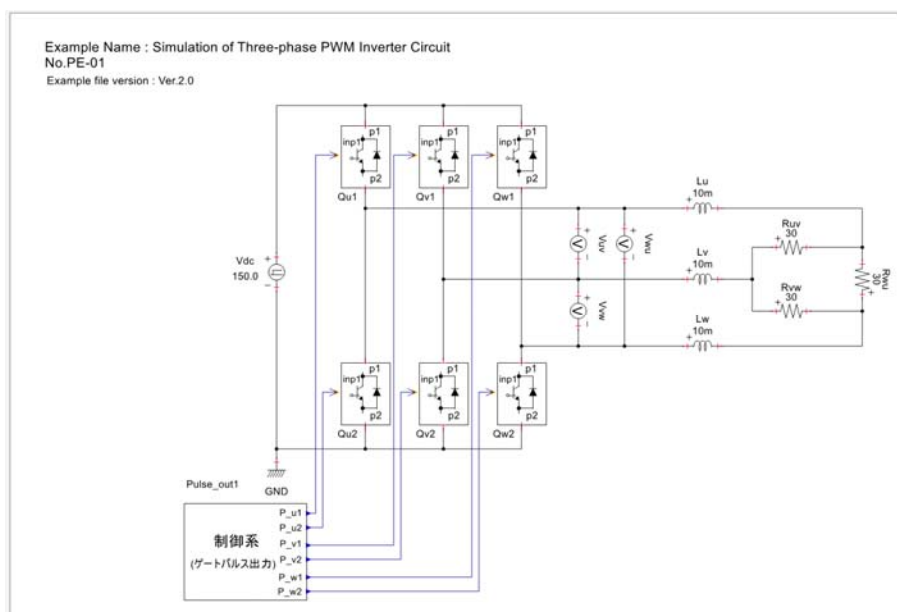
Case name (name)
Specify the case name (name).

Additional OK Cancel

6.2 Analysis procedure (analysis type: transient analysis)

This section describes how to execute a standard transient analysis. As an example, we illustrate the analysis process using the PE-01.xsf file provided with XTAP.

1. Select 'Open' – 'Open circuit diagram' from the 'File' menu in the main window, and select 'PE-01.xsf' from the 'Open file' screen. The PE-01.xsf file is located in work/templates/PE-01, in the folder where XTAP was installed.
2. The following circuit diagram is then displayed.



3. Select 'Analysis settings' from the 'Analysis' menu in the main window, or click the corresponding toolbar button.

4. The analysis settings screen appears.

Analysis Setup	
1 Analysis-file save location	
Analysis-file (.xtf file)	PE-01.xtf
2 Analysis-type selection	
Analysis-type (Analysis)	Transient analysis
3 Analysis conditions	
Case name (name)	Case 1
Calculation time increment (Delta_t)	1.0
Units	μs
Calculation end time (T_max)	0.04
Units	s
BNR switching iteration count (NumBNR)	30
Katzenelson switching iteration count (NumKa)	60
Calculation result output file (BinFile)	PE-01.xpf
Output start time (TimeWindowBegin)	0.02
Units	s
Output end time (TimeWindowEnd)	T_max
Units	s
Output frequency (OutputFreq)	1
Output calculation result in text format/CSV	Yes (text format)
CSV format output file (CSV_File)	PE-01.csv
Debug level (DebugLevel)	0

Case name (name)
Specify the case name (name).

Additional OK Cancel

1 Analysis-file save location

Specify the analysis filename (*.xtf file) and the save location. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

2 Analysis-type selection

Select the analysis type. In this example, 'Transient analysis' is selected.

3 Analysis conditions

Specify the analysis conditions corresponding to the analysis type.

Case name:

Specify the name of the analysis case. This will be used as the title when the analysis is graphed in XPLT.

Calculation time increment:

Specify the calculation time increment (Δt).

Calculation end time:

Specify the calculation end time.

BNR switching iteration count:

Specify the number of iterations for switching the convergence calculation from the standard Newton-Raphson method to the biaxial Newton-Raphson method (this setting usually does not need to be changed).

Katzenelson switching iteration count:

Specify the number of iterations for switching from the biaxial Newton-Raphson method to the Katzenelson method (this setting usually does not need to be changed).

Calculation result output file:

Specify the calculation output file name. By default, the save location is a relative path starting from the location of the circuit diagram file (*.xsf file).

When the save location is instead specified as an absolute path, this is then used as the save location.

Output start time:

Specify the time at which to start collecting the calculation output.

Output end time:

Specify the time at which to end collecting the calculation output.

Output frequency:

Specify the output sampling rate.

Text format/CSV format calculation result output:

Specifying 'Yes (text format)' outputs calculation results in text format (space-separated). Specifying 'Yes (CSV format)' outputs results in CSV format (comma-separated).

Text format output file:

Specify a file name when the calculation output is in text format. The save location is specified relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is then used as the save location.

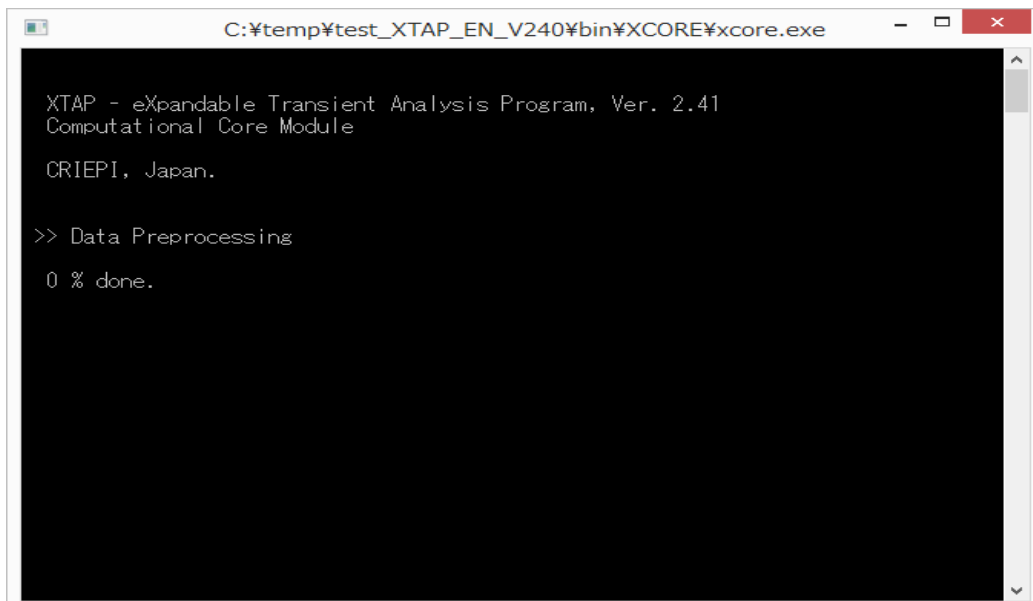
CSV format output file:

Specify a file name when outputting calculation results in CSV format. The save location is a path specified relative to the location of the circuit diagram file (*.xsf file). When the specified save location is an absolute path, this path is used as the save location.

Debugging level:

Specify the debugging level. This setting is not normally used by the user.

5. Click the OK button in the screen.
6. Save the circuit, then perform the analysis by selecting 'Run analysis' from the 'Analysis' menu or by clicking the corresponding toolbar button. The analysis cannot execute if the circuit has been changed. (The 'Run analysis' menu item and corresponding toolbar button are then inactive and cannot be selected).
7. The command prompt is displayed when the analysis is executed.



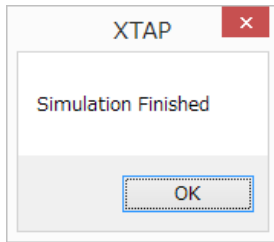
```
C:\temp\test_XTAP_EN_V240\bin\XCORE\xc.exe

XTAP - eXpandable Transient Analysis Program, Ver. 2.41
Computational Core Module

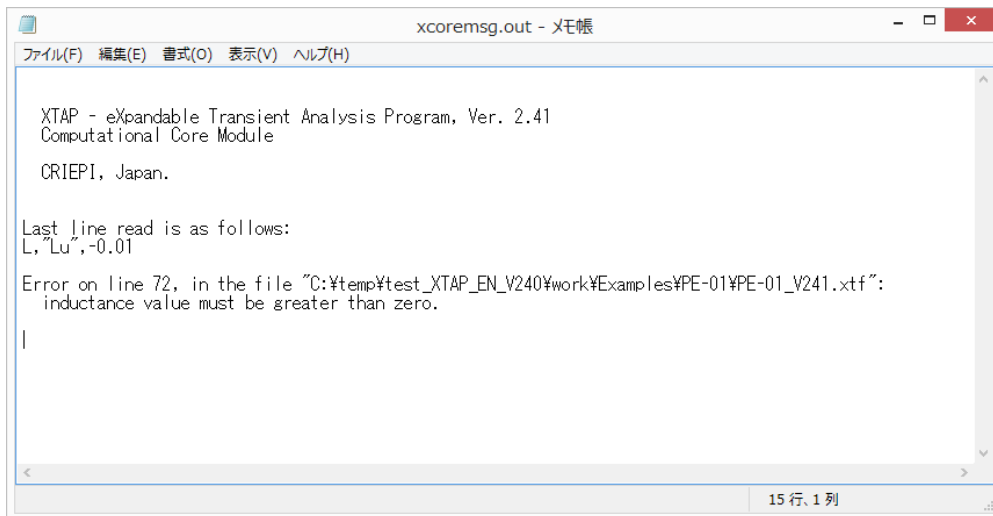
CRIEPI, Japan.

>> Data Preprocessing
0 % done.
```

When the analysis ends normally, the message 'Analysis complete' appears.



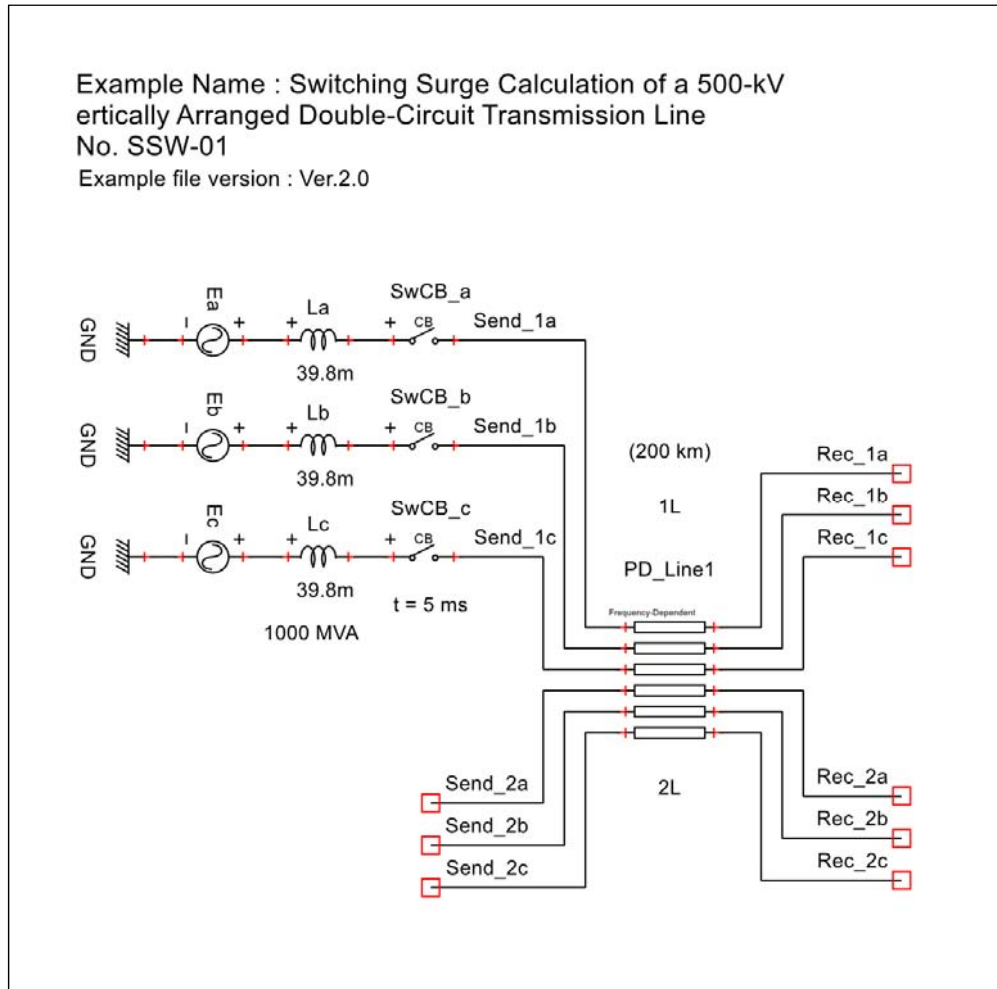
8. An error message is displayed if the analysis process ends with an error. If source of the error is in the circuit, the error message is output and displayed in the file 'xcoremsg.out'. If the error originates from the control block, the error message is output and displayed in the file 'dynamsg.out'. The message file is output to the save location of the analysis file (*.xtf file).



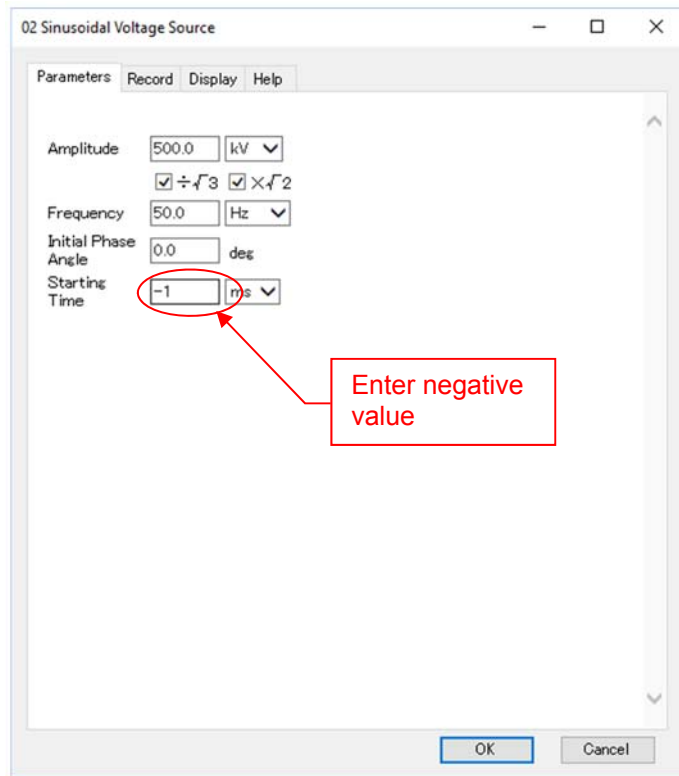
6.3 Analysis procedure (analysis type: steady-state initialization → transient analysis)

This section describes how to perform a steady-state initialization by calculating AC steady-state solutions. As an example, we illustrate the analysis process using the SSW-01.xsf file provided with XTAP.

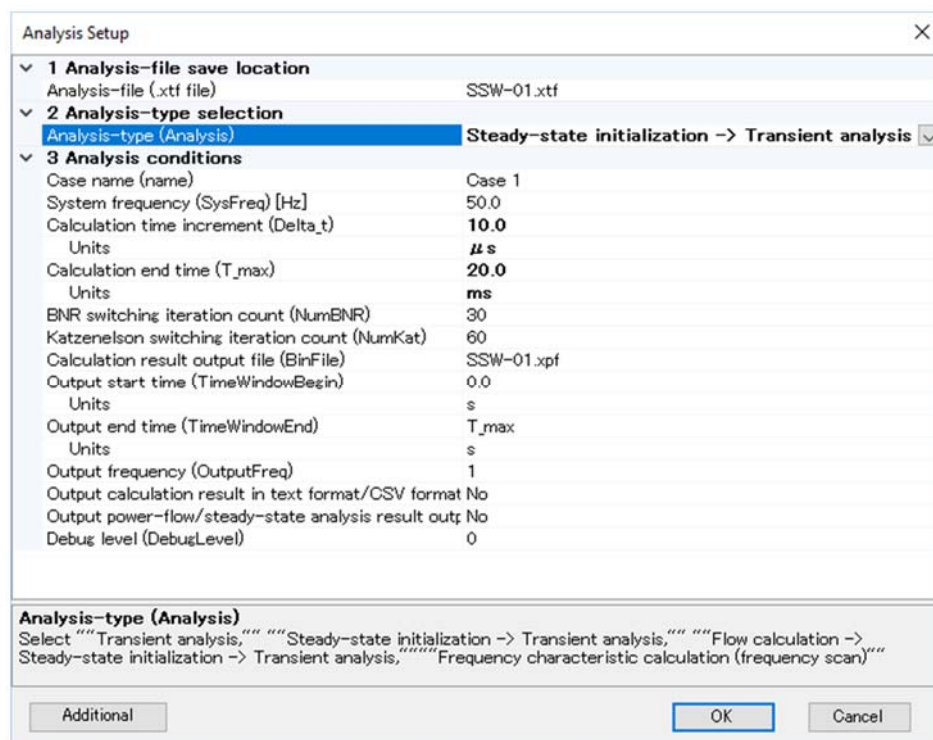
1. Select 'Open' – 'Open circuit diagram' from the 'File' menu in the main window, and select 'SSW-01.xsf' from the 'Open file' screen. The SSW-01.xsf file is located in work/templates/SSW-01, in the folder where XTAP was installed.



2. Sets the parameters for steady-state initialization. Double-click the SIN wave voltage source to open the parameter setting window. To perform steady-state initialization, set “Start time” to a negative value in the parameter settings (any negative values are allowed).



3. Select ‘Analysis settings’ from the ‘Analysis’ menu in the main window.
4. The analysis settings screen then appears.



1 Analysis file save location

Specify the analysis filename (*.xtf file) and the save location. The default setting is [Circuit diagram file name] + .xtf. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

2 Analysis type selection

Select the analysis type. For this example, select 'Steady-state initialization → Transient analysis'.

3 Analysis conditions

Specify the analysis conditions corresponding to the analysis type. All the items other than the three below are the same as for the transient analysis.

System frequency:

Specify the system frequency. This must be the same as the frequency of the power source in the circuit. A warning is output if the circuit contains a sine- or cosine-wave power source with a frequency different from this set frequency.

Output power flow/steady-state analysis result output file:

Setting 'Yes' outputs the steady-state analysis results.

Power flow/steady-state analysis result output file:

Specify a file name when outputting the steady-state analysis results. The default setting is [Circuit diagram file name] + _PFSS.txt. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

5. Click the 'Start calculation' button on the screen.
6. The command prompt is displayed when the analysis process is executed.

```
XTAP - eXpandable Transient Analysis Program, Ver. 2.41
Computational Core Module

CRIEPI, Japan.

>> Data Preprocessing

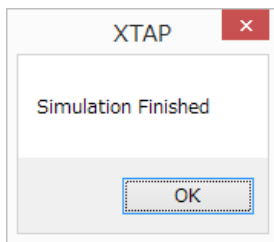
100 % done.

>> Input Data

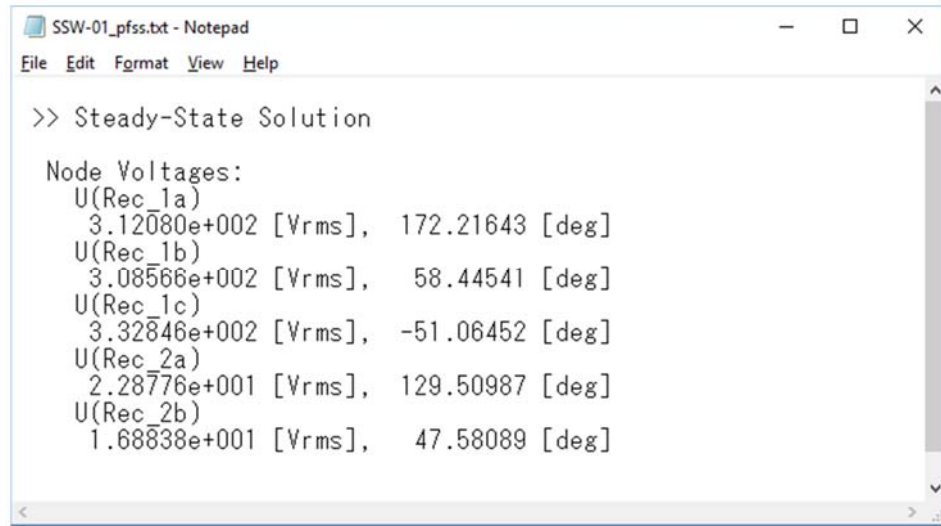
Simulation Options:
  Name = Case 1
  Analysis = Transient simulation
              starting from zero initial conditions.
  Delta_t = 1.000000e-005
  T_max = 2.000000e-002
  NumBNR = 30
  NumKat = 60

Network Info:
  NumNodes = 18
  NumBranches = 10
```

When the analysis process ends normally, the message 'Analysis finished.' appears.



The results of the AC steady-state calculation are output as a text file. The AC steady-state solutions of the variables specified in the 'Recording' tab are output.



6.4 Analysis procedure (analysis type: power flow calculation → steady-state initialization → transient analysis)

This section describes how to perform a steady-state initialization by calculating AC steady-state solutions, with steady-state conditions applied to the generators and loads, as determined by power-flow calculation (the 'L method'). The steady-state initialization workflow is as follows:

- ① Create the circuit diagram (a three-phase bus).
↓
- ② Specify the power-flow conditions using power-flow-calculation auxiliary parts.
↓
- ③ Set information-acquisition flags for the power-flow calculation using macro parts.
↓
- ④ Execute the analysis.

① Create circuit diagram

The circuit diagram is created with the usual procedure, but the nodes and branches used to set initial values (synchronous generators or user-created macro parts) must be connected to a three-phase bus. See Section 5.6 for details of how to connect a three-phase bus.

② Specify the power flow conditions using power-flow-calculation auxiliary parts

Specify the power-flow conditions for each bus using power-flow-calculation auxiliary parts. These parts are located in the '20 Power flow calculation/steady-state initialization auxiliary parts' section of the part category window. Use these parts to enter the initial values specified in the power-flow calculation. A power-flow-calculation auxiliary part should be connected to phase A of a three-phase bus. The reference capacity is specified in Step 4. This capacity is not the reference capacity for individual devices, and therefore care is needed when setting parameters.

③ Use macro parts to set flags for acquiring power flow calculation information

When connected to a three-phase bus, the following parts can use the results of a power-flow calculation (power-flow-calculation information) as parameters:

Parts that can acquire power-flow-calculation information

01 RLC

- 12 RX Parallel load model (single-phase)
- 13 RX Parallel load model

07 Generator

00 Synchronous generator

- 00 Synchronous generator (d1, q1, accounting for magnetic saturation)
- 01 Synchronous generator (d1, q2, accounting for magnetic saturation)
- 02 Synchronous generator (d1, q1)
- 03 Synchronous generator (d1, q2)
- 04 Synchronous generator (d1, q1, accounting for magnetic saturation and axis torque)
- 05 Synchronous generator (d1, q2, accounting for magnetic saturation and axis torque)
- 06 Synchronous generator (d1, q1, accounting for axis torque)
- 07 Synchronous generator (d1, q2, accounting for axis torque)

07 Generator

- 00 Infinite bus generator

99 User's Components

- (Arbitrary macro part)

The part properties for the synchronous generators and infinite bus generators do not have a tab for power-flow-calculation information acquisition; this function is enabled by default. For arbitrary user-created macro parts, the choice to use power-flow-calculation information can be set in the power-flow-calculation information tab.

Parts that acquire power-flow-calculation information operate as follows:

- (1) During steady-state initialization, the part is removed from the three-phase bus.
 - ※1 When the generator bus of an auxiliary component for the power-flow calculation (slack generator bus, PV generator bus, PQ generator bus bar) is connected to the three-phase bus, a three-phase voltage source with the voltage and phase according to the power-flow calculation result of the bus is automatically added.
 - ※2 When the load bus of an auxiliary component for the power-flow calculation (PV load bus, PQ load bus) is connected to the three-phase bus, the three-phase load with the voltage and phase according to the load flow calculation result of the bus is automatically added.
 - ※3 The “13 RX parallel load model” cannot be removed from the three-phase bus in the steady-state initialization calculation. Therefore, when the load bus of the auxiliary component for power-flow calculation is connected to the three-phase bus, both the RX parallel load model and the automatically-added three-phase load are considered in the steady-state initialization calculation. Therefore, if you want to remove the load model from the three-phase bus, please use “14 RX parallel load model (exclude enabled)” instead of “13 RX parallel load model”.
- (2) The following variables (parameters) can be used in the parameter fields and formula fields of part properties. The value is acquired from the three-phase bus to which the part is connected.

Vbus	Bus voltage [V]
theta_bus	Bus voltage phase [rad]
PGbus	Bus generator active power [W]
QGbus	Bus generator reactive power [var]
PLbus	Bus load active power [W]
QLbus	Bus load reactive power [var]
theta_shifted	Phase difference from bus voltage reference phase [deg]

Caution

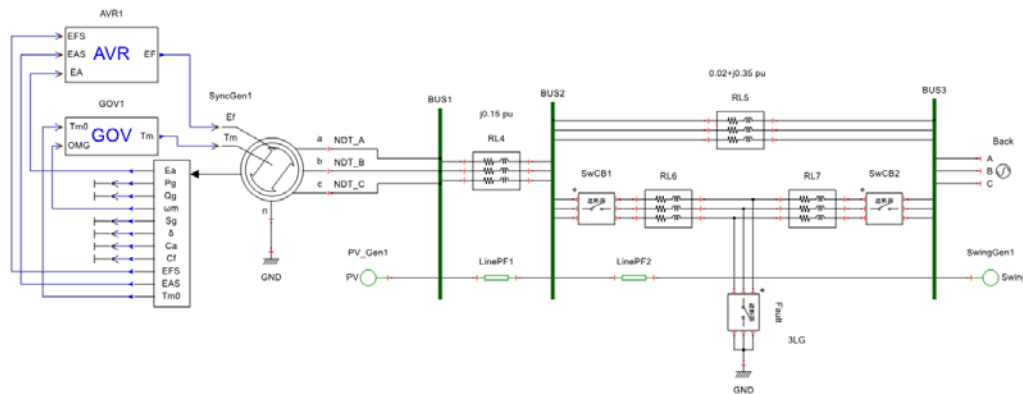
For synchronous generators and infinite bus generators, power-flow-calculation information acquisition is enabled by default (and cannot be disabled). Problems such as information not being considered during steady-state initialization, or initialization not being performed properly, can therefore occur when an analysis is performed without executing a power-flow calculation. When using these parts, the analysis type should therefore be set to the type that includes power-flow calculation.

④ Execute analysis

Execute the analysis once the circuit has been created using Steps 1, 2, and 3.

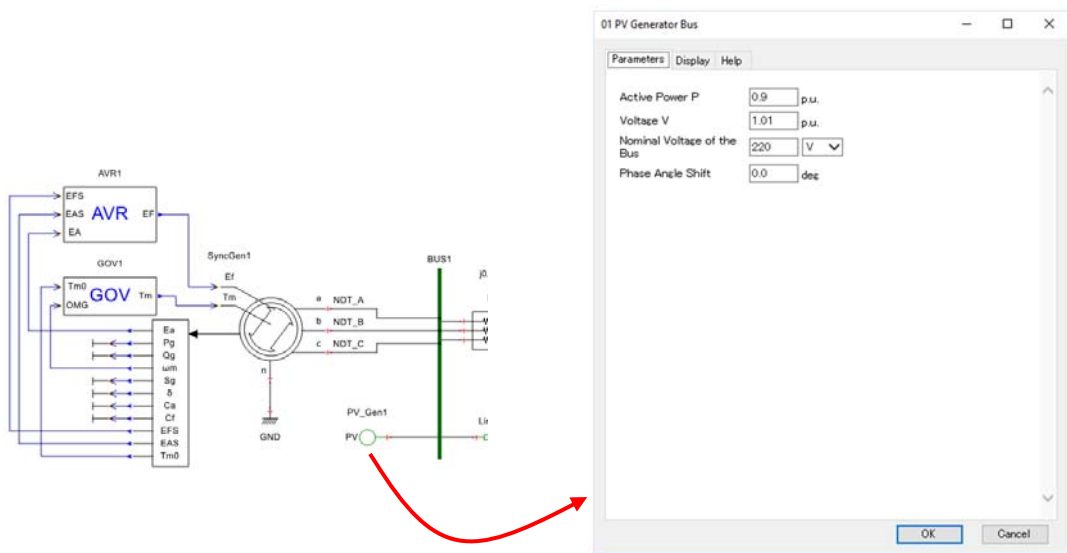
Below is an example of a power-flow calculation, steady-state initialization, and transient analysis using one of the templates (MAC-01) provided with XTAP.

1. From the 'File' menu in the main window, select 'Open circuit'. Then, select 'MAC-01-A.xsf' from the 'Open file' screen. The file MAC-01-A.xsf is located in bin/work/templates/MAC-01, in the folder where XTAP is installed.
2. The circuit diagram below then appears.

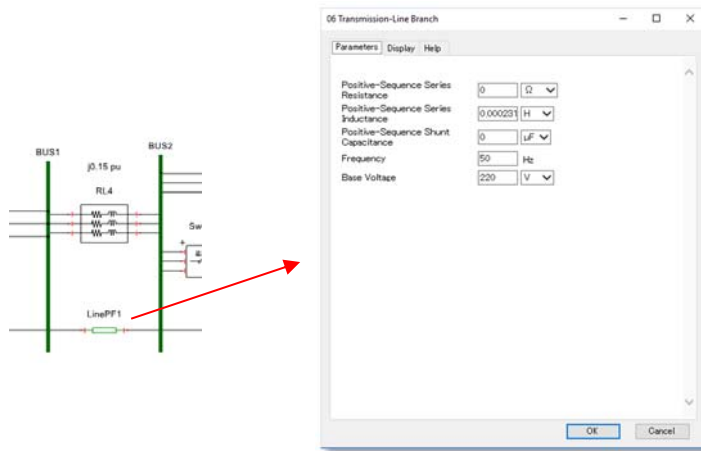


3. Specify the generator power-flow conditions.

Specify the conditions to be applied during the steady-state operation of the generator for the power-flow calculation. From the 'part category' window, select and use the part '07 Generator 00 synchronous generator' in combination with either the part '20 Power flow calculation/steady-state initialization auxiliary parts – 01 PV specification generator bus' or '02 PQ specification generator bus'. The template is designated 'PV', specifying the generator output and generator terminal voltage.

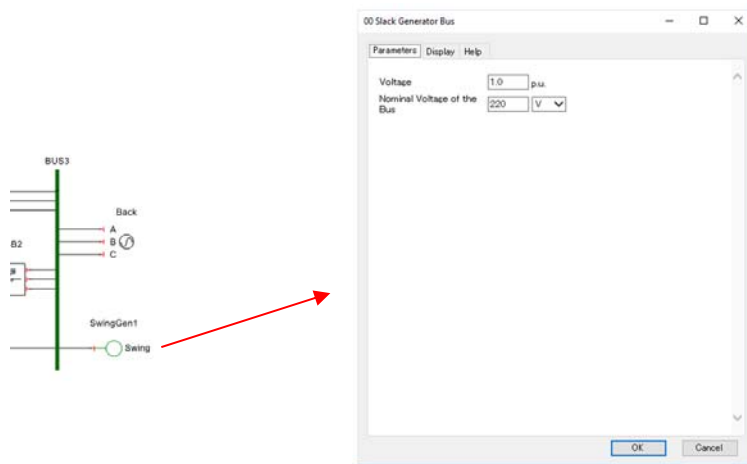


4. Set the power-transmission-line impedance.
Specify the impedance of the power-transmission line to be considered during the power-flow calculation. This calculation considers only the portions with a positive phase, and therefore only these portions can be specified in the parameters.



Use the part '06 Line branch' for a power-transmission line, and '07 Transformer branch' for a transformer.

5. Set the reference generator bus.
One reference generator bus must be set. The part '07 Generator — 02 Infinite bus generator' is normally used, combined with '20 Power flow calculation/steady-state initialization auxiliary part — 00 Reference generator bus'.



- Select 'Analysis settings' from the 'Analysis' menu in the main window. The analysis settings screen appears.

1 Analysis-file save location

Specify the analysis file (*.xtf file) name and the save location. The default setting is [Circuit diagram file name] + .xtf. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

2 Selecting the analysis type

Select the analysis type. For this example, 'Power-flow calculation → Steady-state initialization → Transient analysis' is selected.

3 Analysis conditions

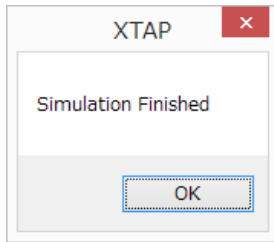
Specify the analysis conditions corresponding to the analysis type. All the items other than the following are the same as for the 'steady-state initialization → transient analysis' process.

Reference capacity:

Specify the reference capacity of the system. Note that this figure is not the reference capacity for each device.

- Click the 'Start calculation' button on the screen.
- The command prompt is displayed when the analysis is executed.

When the analysis ends normally, the message 'Analysis complete' appears.



The results of the power-flow calculation are output as a text file. They can be used to check that the initial values specified in Steps 3 to 5 were used.

A screenshot of a Notepad window titled 'MAC-01-A_pfsst.txt - Notepad'. The text inside shows the results of a power-flow calculation. It includes base values, bus voltages for three buses, active and reactive powers at the buses, and node voltages for three nodes.

```
>> Power-Flow Solution

volt-ampere base = 1.00000e+005 [VA]
system frequency = 50.0 [Hz]

Bus Voltages (magnitude and phase angle):
BUS1
  1.010000 [pu],  16.75089 [deg]
BUS2
  0.998893 [pu],  9.06408 [deg]
BUS3
  1.000000 [pu],  0.00000 [deg]

Active and Reactive Powers at Busses (PG, QG, PL, and QL):
BUS1
  0.900000,  0.135279, -0.000000,  0.000000 [pu]
BUS2
  0.000000,  0.000000,  0.000000,  0.000000 [pu]
BUS3
 -0.891880,  0.128566,  0.000000,  0.000000 [pu]

>> Steady-State Solution

Node Voltage:
U(NDT_A)
  1.28287e+002 [Vrms],  16.75089 [deg]
U(NDT_B)
  1.28287e+002 [Vrms], -103.24911 [deg]
U(NDT_C)
```

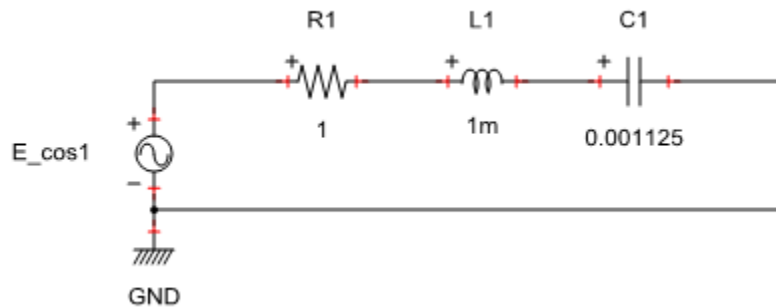
9. An error message will be shown when an error occurs during the execution of the simulation. When the error is related to the circuit part, the details of the error will be displayed in the file "xcoremsg.out". When the error is related to the control-system part, the details of the error will be displayed in the file "dynammsg.out". When the error is related to the power-flow calculation, the details of the error will be displayed in the file "flowmsg.out". All these file are located in the same folder of the simulation .xtf file.

6.5 Analysis procedure (analysis type: frequency characteristic calculation)

The frequency-characteristic calculation is a function that calculates the response to a change in a power-source frequency in a circuit, between user-specified start and end values, in arbitrary increments.

Consider a simple RLC series circuit as an example.

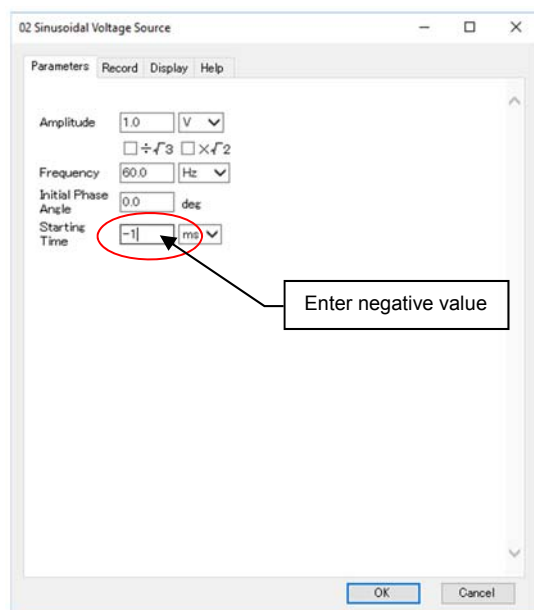
1. Create a series circuit consisting of a power source, resistance, inductance, and capacitance as shown in the figure below.



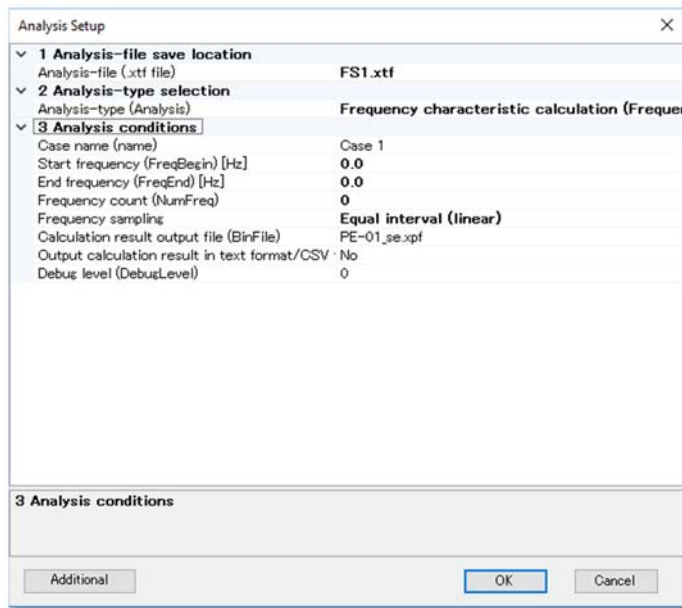
The calculated frequency characteristic is that produced by the power source 'E_cos1'. Usable power sources may be of one of the following four types. Other power sources are treated as short-circuited (voltage sources) or open (current sources) in the context of the calculation. The usable power sources will also be treated as short-circuited (voltage sources) or open (current sources) if the setting described in Step 2 (entering a negative start time) is not entered.

Sine wave voltage source Sine wave current source
Cosine wave voltage source Cosine wave current source

2. Set the parameters for the frequency-characteristic calculation. Double-click the power source to open the 'parameter setting' screen. Set 'Start time' to a negative value (any negative value is acceptable) to cause this power source to become recognized as the power source for implementing frequency variation.



3. Select 'Analysis settings' from the 'Analysis' menu in the main window. The analysis settings screen appears.



1 Analysis file save location

Specify the analysis filename (*.xtf file) and the save location. The default setting is [Circuit diagram file name] + .xtf. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

2 Analysis type selection

Select the analysis type. For this example, select 'Frequency characteristic calculation (frequency scan)'.

3 Analysis conditions

Specify the analysis conditions corresponding to the analysis type.

Case name:

Specify the analysis case name. This name is used as the title when the analysis is graphed in XPLT.

Start frequency:

Specify the starting frequency for the frequency-characteristic calculation.

End frequency:

Specify the end frequency.

Frequency count:

Specify the number of frequencies at which to perform the calculation between the start and end frequencies.

Frequency sampling:

Specify whether to use equal (linear) or log-scale sampling intervals.

Calculation result output file:

Specify the output file name. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

Text format/CSV format calculation result output:

Specifying 'Yes (text format)' outputs the calculation results in text format (space-separated). Specifying 'Yes (CSV format)' outputs them in CSV format (comma-separated).

Text format output file:

Specify a file name when outputting calculation results in text format. When the save location is specified as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

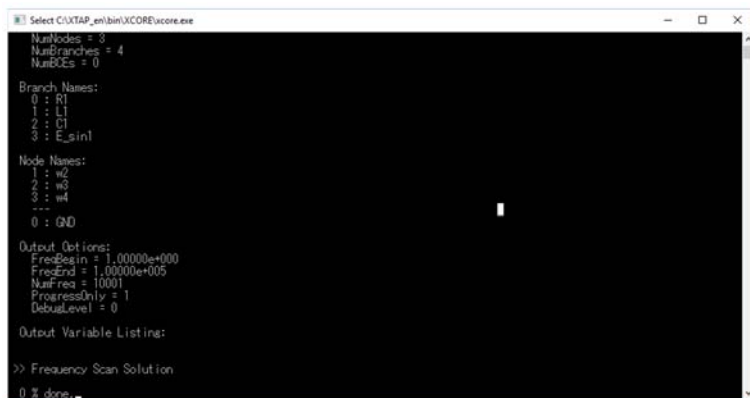
CSV format output file:

Specify a file name when outputting calculation results in CSV format. When the save location is expressed as a relative path, this path is assumed to be expressed relative to the location of the circuit diagram file (*.xsf file). When the save location is specified as an absolute path, this path is used as the save location.

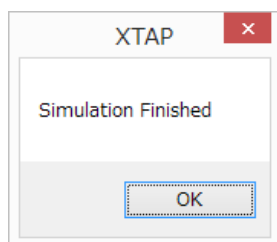
Debugging level:

Specify the debugging level. This setting is not normally used by the user.

4. Click the 'Start calculation' button on the screen.
5. The command prompt is displayed when the analysis process is executed.



When the analysis process ends normally, the message 'Analysis finished' appears.



6.6 Files related to analysis processes

This section describes the files used for analyses (*.xsf files) and those generated by analysis processes.

***.xsf (XTAP schematic file) files**

These binary files store circuit diagrams to be analyzed.

***.xpf (XTAP plot file) files**

These binary files store the analysis results. They can be used to display graphs with XPLT.

xcorem.sg.out

This file contains errors and warning messages, generated during analysis, that relate to circuit areas. Generated errors are displayed automatically on the screen, but warnings unaccompanied by errors are not displayed automatically, and therefore require caution.

dynam.sg.out

This file contains errors and warning messages, generated during analysis, that relate to control-block areas. Generated errors are displayed automatically on the screen, but warnings unaccompanied by errors are not displayed automatically, and therefore require caution.

flowmsg.out

The details of errors and warnings related to the power-flow calculation will be saved in this file. Errors will be automatically displayed in the screen when executing the simulation, however warning will not be automatically displayed.

***.xtf (XTAP text file) files**

Text-format input data files that are generated from *.xsf files by the XTAP graphical user interface, and are read by the XTAP analysis program.[#]

***.xmf (XTAP model file) files**

These files specify models (macro parts) included by *.xtf files. Like *.xtf files, *.xmf files are text files generated from *.xsf files by the graphical user interface.[#]

***.xtf.pp (XTAP text file preprocessed) files**

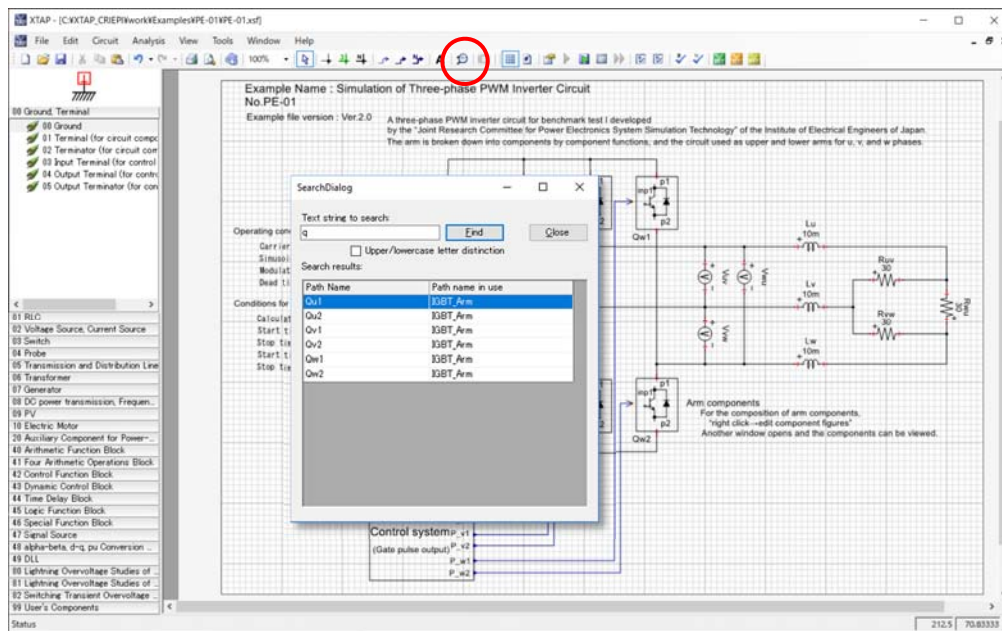
These files are generated by the XTAP analysis program as intermediate process outputs.[#]

Normally .xpf files, .html files, .xtf files, .xmf files and xtf.pp files are located in the same folder as the .xsf schematic file. However, the location of .xpf files and .xtf files can be changed by changing options in the simulation setup.

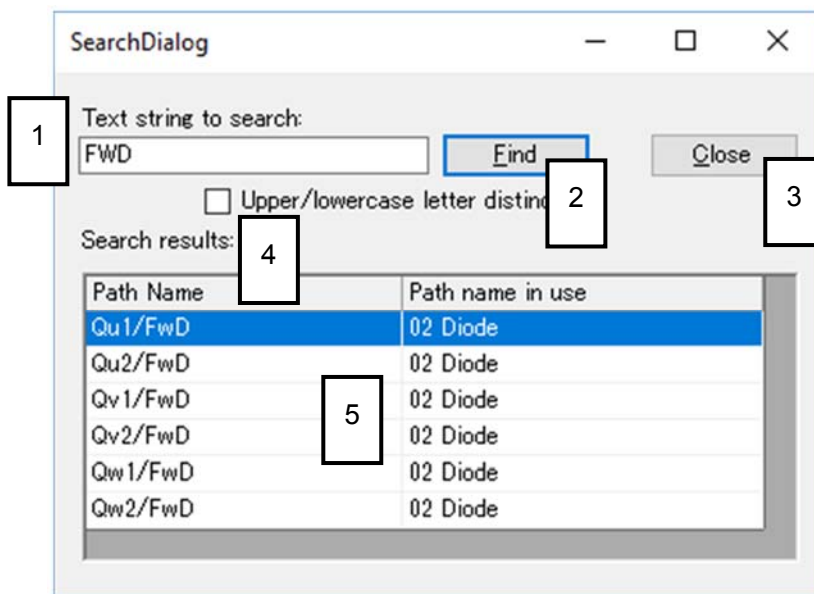
[#]Users do not need to reference these files.

6.7 Search function

The search function can be used to find parts or other elements in a circuit diagram.



Select 'Search' from the 'Circuit' menu or click the corresponding toolbar button to open the search screen.



Search-screen items:

- 1 is used to enter the target text string;
- 2 starts the search;
- 3 closes the window;
- 4 makes the search case-sensitive when this checkbox is selected;
- 5 lists the search results. Clicking a 'Path name' entry selects that part or element. If the selected search result is located within a macro part, the circuit diagram of that macro part opens to show the result.

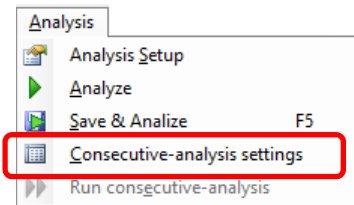
7 Consecutive-Analysis Function

7.1 Overview of consecutive analysis function

This function uses parameter settings derived from values such as analysis-circuit element values. The function calculation involves consecutively varying the parameter values. In addition, XTAP also has another function for calculating statistical switching surges (described in Chapter 8).

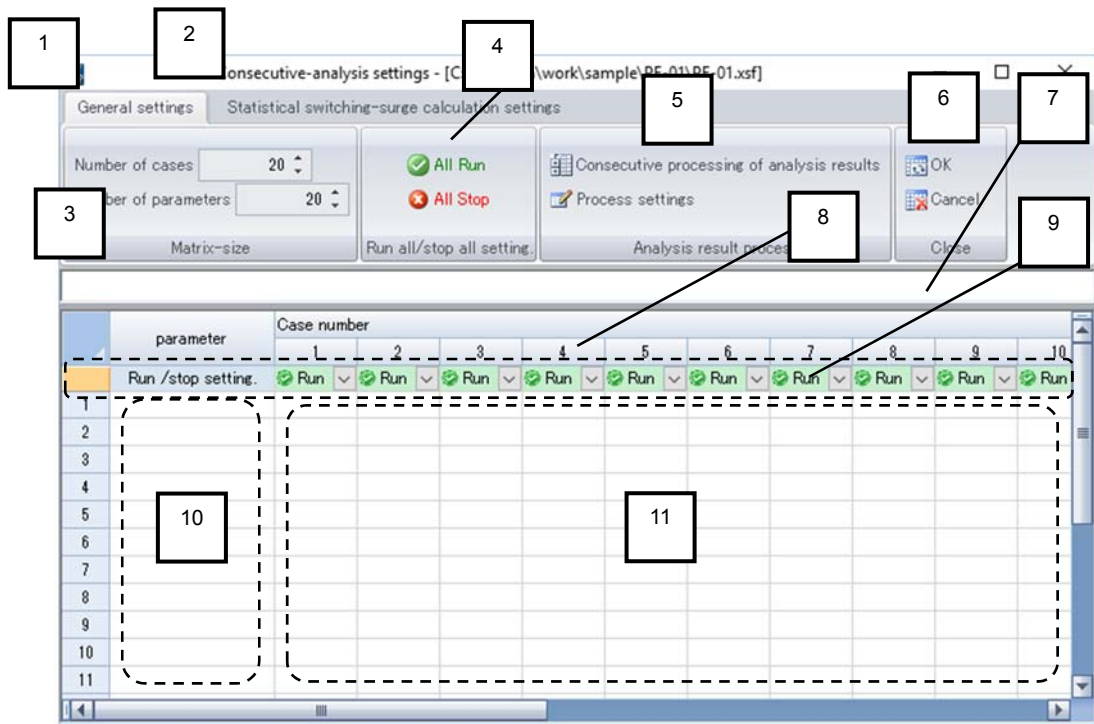
7.2 Opening the consecutive-analysis settings window

The setting for the consecutive-analysis can be made in the consecutive-analysis settings window. Select “Consecutive-Analysis Settings” from “Analysis” in the main menu. Or click the “Consecutive-Analysis Settings” icon in the toolbar.



7.3 Consecutive-analysis settings window

This section describes the consecutive-analysis settings window. An example is shown in the following screenshot.



1. General settings tab

- used to set consecutive analysis.

2. Statistical switching-surge calculation settings tab

- used to set the statistical switching-surge calculation function (described in Chapter 8).

3. Matrix-size menu

- used to specify the number of cases to be executed consecutively and the number of parameters to set. The data input screen changes according to the values specified here. Up to 1,000 cases and up to 100 parameters can be specified. The analysis can still proceed even if some of the specified cases and parameters are left blank.

4. Run all/stop all menu

- It is possible to select at once cases that will be executed and cases that will be stopped. By clicking on “Run all” all cases will be set to be executed. By clicking on “Stop all” all cases will be set to be stopped. Moreover, when selecting some case numbers (refer the item 10 shown below), this menu will be automatically changed to “Run Selected Cases” and “Stop Selected Cases”

5. Analysis result process menu

- contains the buttons used to run and set statistical processes (operations acquiring values such as the waveform maximum or average values), executed on multiple-analysis results (described in Section 7.8).

6. Close menu

- closes the window. Clicking ‘OK’ closes the window with the entered settings applied. Clicking ‘Cancel’ closes the window without applying the changes.

7. Formula area

- displays the parameter input cell information. This area can also be used to input numerical values and formulas to be provided as parameters directly.

8. Case number button

- This button is used to select several cases at once. It can be used to setting to run or to stop several cases at once, for example. It is also possible to selected cases by using the Ctrl or Shift keys.

9. Run/stop selection button

- determines whether or not each case will be executed in a consecutive analysis.

10. Parameter name input cells

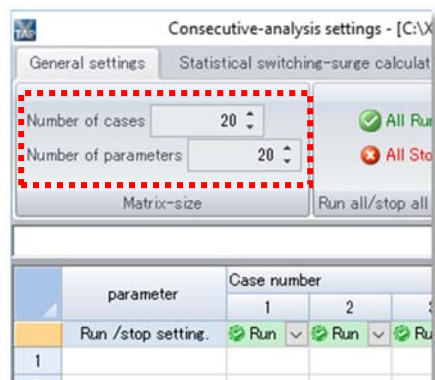
- used to enter parameter names (variable names).

11. Parameter input cells

- used to enter the numbers and formulas required as parameters.

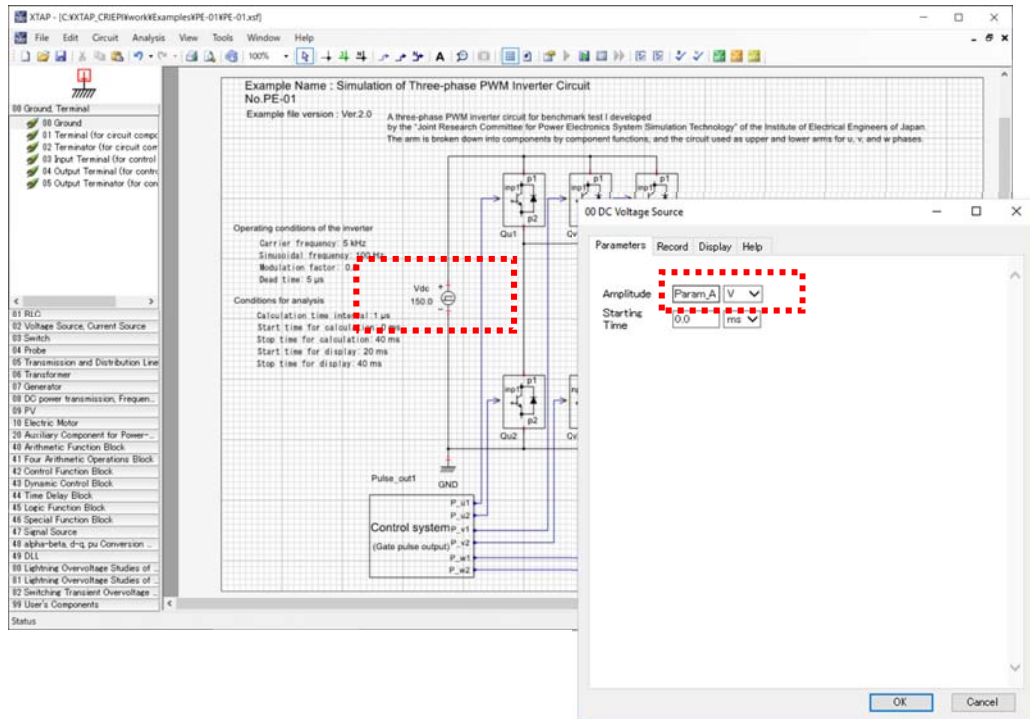
7.4 Setting the number of cases and of parameters

Use the 'matrix size' menu to specify the number of cases and of parameters to execute consecutively. The number of parameter-setting cells then changes to match the specified number of cases and parameters.



7.5 Parameter input method

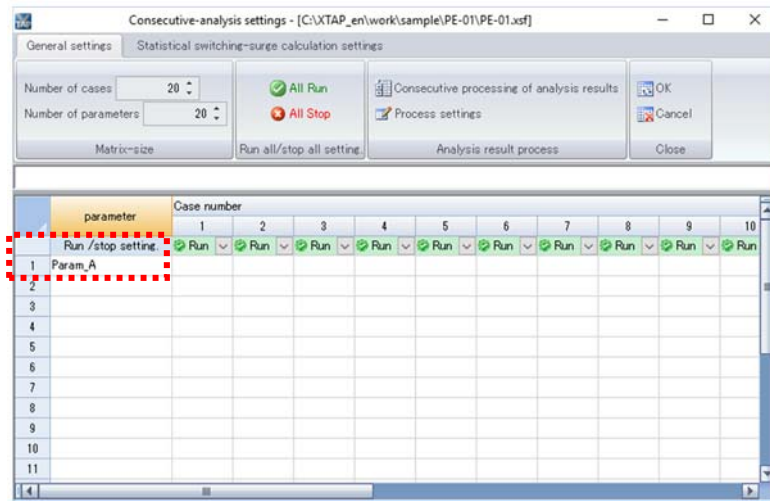
1. The properties normally entered as values are entered as variables. In the example below, the voltage-source voltage is set as the parameter 'Param_A'.



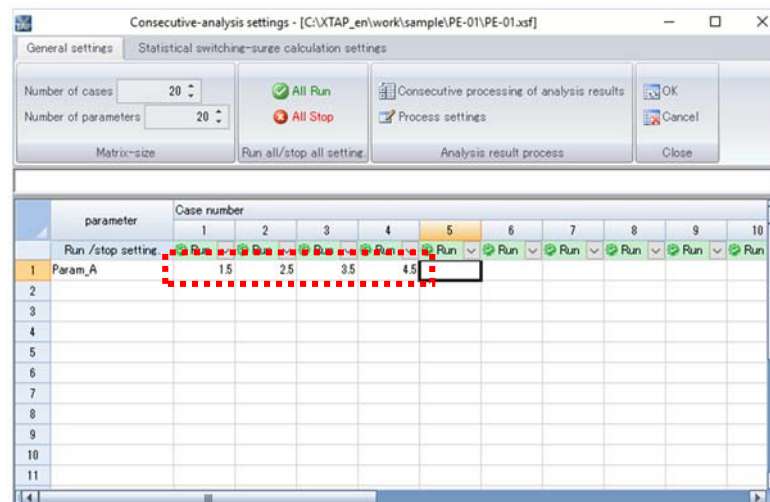
Most of the properties (e.g., voltage, current, frequency, resistance, inductance, and the 'switch ON' time) of parts (e.g., power sources, resistors, and switches) can be set as parameters. However, note that variable names cannot be used for some properties, such as initial values (e.g., the initial current in an inductor or the initial voltage of a capacitor), file names of parts that reference external files, and properties of power-flow-calculation auxiliary parts.

- All properties that can be used as macro-part parameters are permitted.
- Only alphanumeric characters (uppercase letters, lowercase letters or numbers) and underscores “_” can be used for the parameter names.
- Note that auxiliary units (prefixes for units of measurement such as M, k, m, or μ) for values set as parameters in the properties input screen are ignored.
- Note that a normal analysis is no longer possible if part-property parameters are set in order to run consecutive analysis (since values are not input for variables entered as parameters).

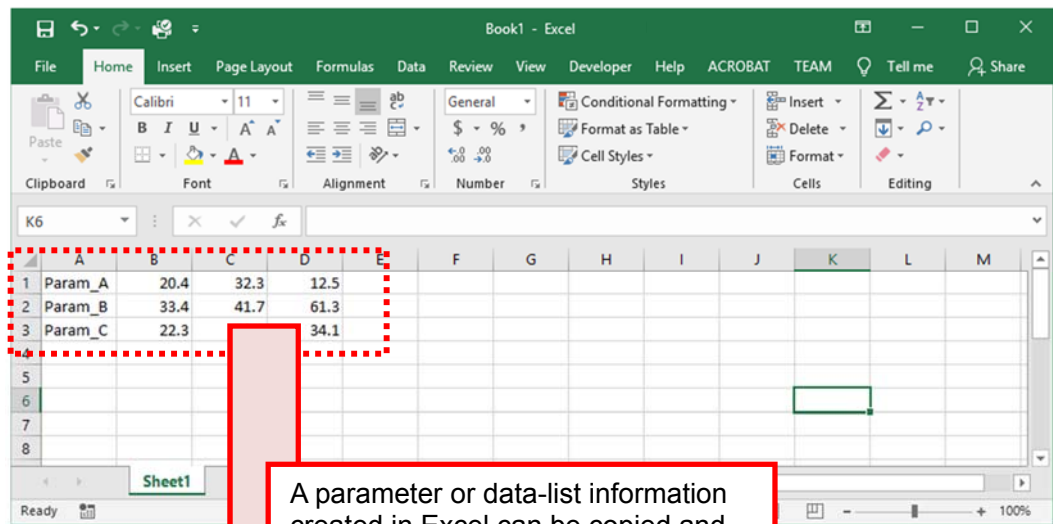
2. Enter the parameter name in the 'Parameter name' field.



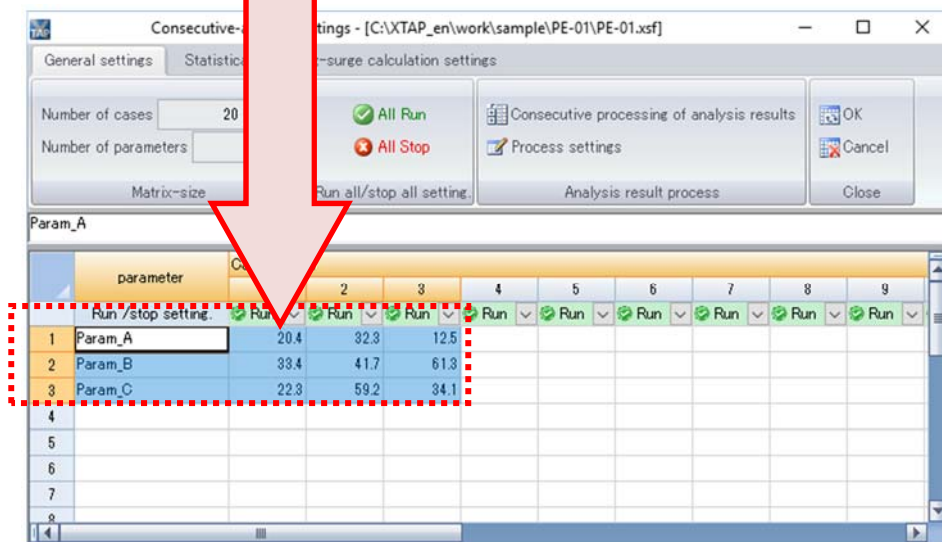
3. Enter the parameter value for each case.



Entering parameter values can be facilitated by copying the data sets for multiple cases from a spreadsheet program such as Excel, and pasting them into XTAP.

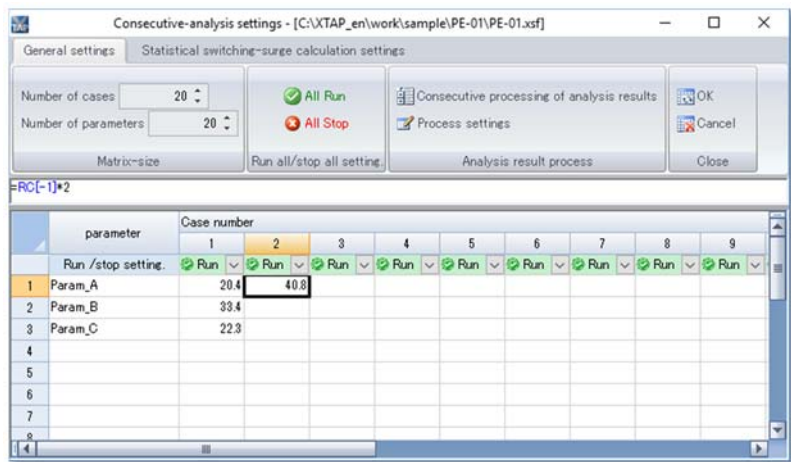
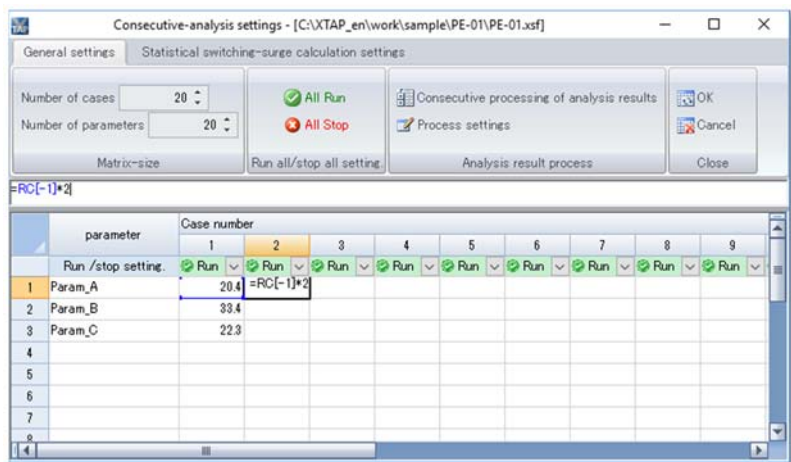


A parameter or data-list information created in Excel can be copied and pasted as a block of multiple cells.

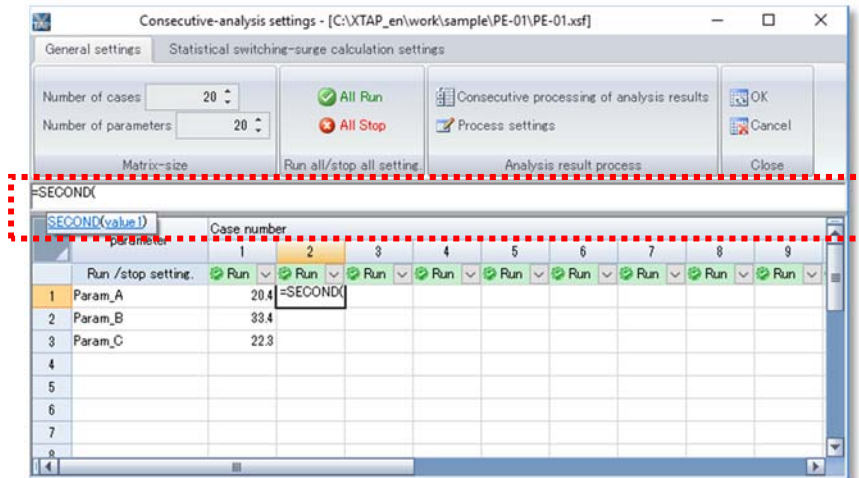


Parameter values can be entered using a formula syntax. The information in the formula area or a cell is interpreted as a formula when its first character is an 'equals' sign (=). Cells are referenced in R1C1 format.

A reference in R1C1 format is entered as follows. A relative cell reference to the cell that is located one cell to the left of the current cell is written 'RC[-1]', where 'R' stands for 'row' and 'C' for 'column'. The brackets [] specify the number of steps separating the referenced cell from the current cell, in the horizontal or vertical direction. Specifying 'R' (row) without a number in brackets means that the referenced cell is in the same row as the current cell. 'R[-1]' signifies 'one row up', and 'R[1]' means 'one row down'. Similarly, 'C' (the column), specified without a number in brackets, signifies that the referenced cell is in the same column as the current cell, 'C[-1]' signifies 'one column to the left', and 'C[1]' means one column to the right. Absolute cell references are specified without brackets. For example, 'R1C1' denotes the cell of the first row and first column.

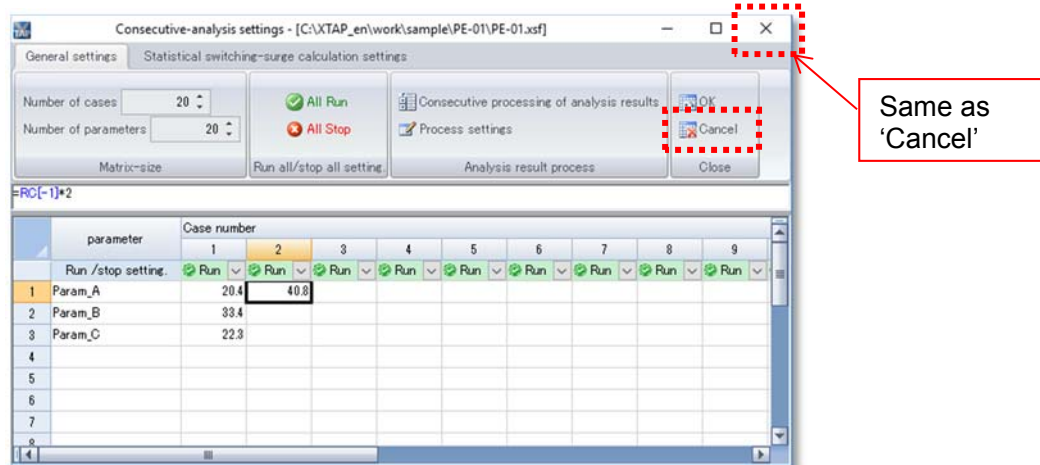


Entering an 'equals' sign (=) in the formula area followed by a letter or text string displays a list of functions starting with the entered text. (Hovering the mouse cursor over the name of a function displays a tooltip with brief help information for that function.)



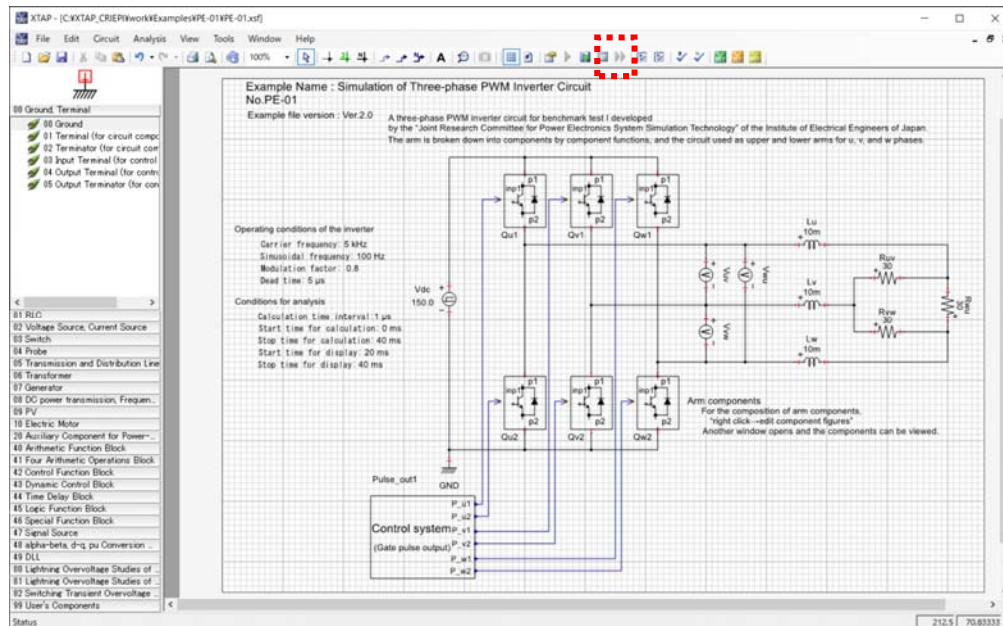
7.6 Ending/canceling parameter settings

When the parameter settings have been entered, click 'OK' in the 'Close' menu to apply the changes to the analysis circuit. (An analysis cannot be executed without first saving the circuit.) To cancel the editing of parameter input cells, click 'Cancel'. The 'X' button at the top-right corner of the window performs the same operation as the 'Cancel' button. When analysis is executed (described later), the process stops in the error status if a parameter cell is missing a value. However, rows and columns left completely blank are automatically ignored; no error is generated if the specified number of cases and parameters exceeds the actual number of cases and parameters provided.

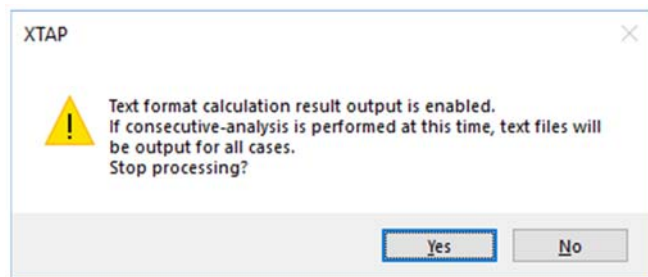


7.7 Executing a consecutive analysis

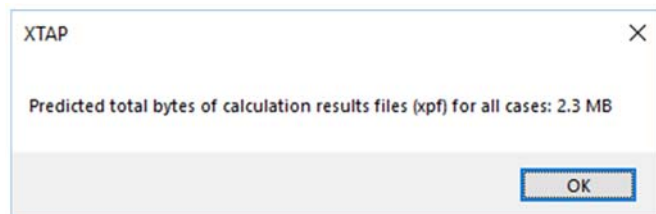
After entering the analysis settings and saving the circuit, press the toolbar button indicated below to execute the consecutive analysis.



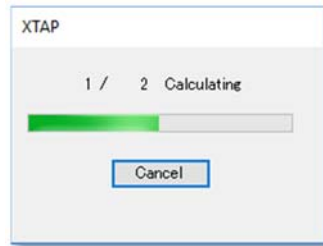
The following message appears if a text-format calculation output has been selected in the analysis settings. The file size may be large for such an output. To cancel an analysis, select 'Yes'. To output the calculation results in text format using the set analysis settings, select 'No'. If there is no problem, select 'No' to proceed with the calculation.



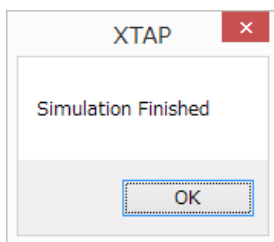
The expected size of the calculation output file will be displayed. Check if there is sufficient hard-disk capacity before proceeding with the simulation.



When the calculation starts, a message is displayed indicating the case being currently executed. Cases for which the run/stop setting has been entered as 'Stop' are not executed.



The following message appears when the analysis of all the cases has finished.



The following files are then generated:

Analysis file name_(case number).xtf

- input data files (one for each case) in text format.

Analysis file name_(case number).xtf.pp

- files generated by the XTAP analysis program as intermediate process outputs.

AUTOSUB.xmf

- a data file containing the created circuit data in the form of macro parts (since macroing is required for using parameters).

Analysis file name_(case number).xpf

- binary files storing the analysis results (one file for each case).

Analysis file name_all.csv

- a file that contains the analysis results, and the file name and time for each case. These analysis results are used when performing statistical processes.

7.8 Statistical processes for analysis results

The recorded waveform results of a consecutive analysis carried out on a set of cases can be loaded into XTAP to determine the following quantities:

- Maximum (MAX)
 - Minimum (MIN)
 - Maximum absolute value (MAXABS)
 - Average (AVE)
 - Integral (INTEGRAL)
 - Effective value (RMS) (Frequencies are set in the PostProcessing.ini file.)
- *The PostProcessing.ini file is stored in the folder containing xgui.exe (normally C:\XTAP\bin).*

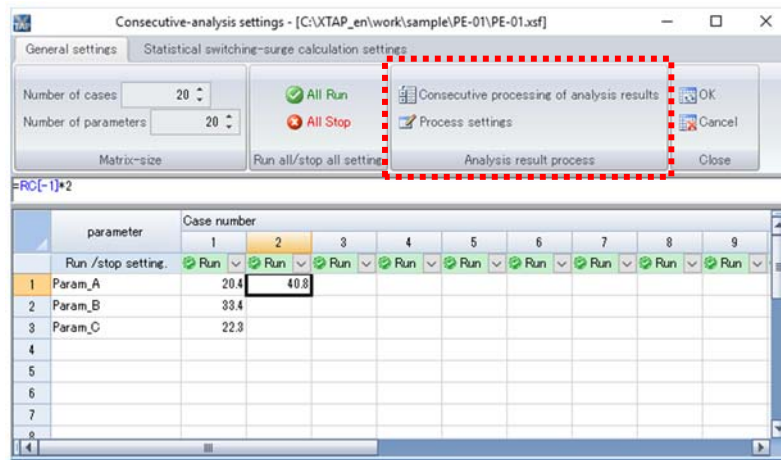
The format used for writing values is as follows:

<pre>[PostProcessingSection] HZ=50</pre>
--

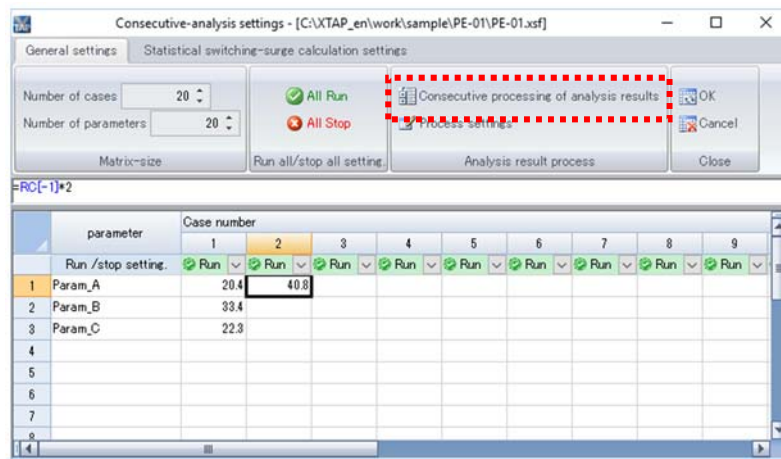
In addition to the six above quantities, operations on the recorded waveforms can be implemented using DLL files, as required by the user. For details, see Appendix 3 in this manual, “*Creating and Using DLL Files for Statistical Processes.*”

7.9 Executing statistical processes

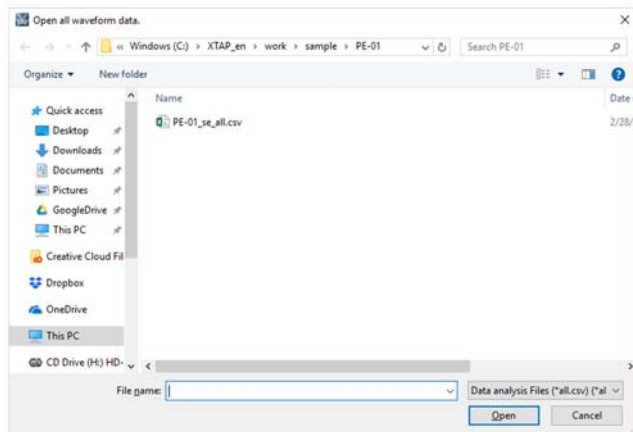
Statistical processes are executed from the 'analysis-result process' menu in the 'consecutive-analysis settings' window.



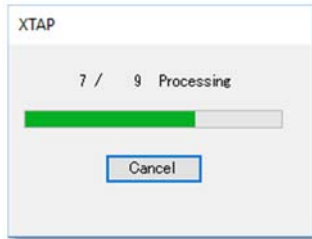
(1) Consecutive processing of analysis results



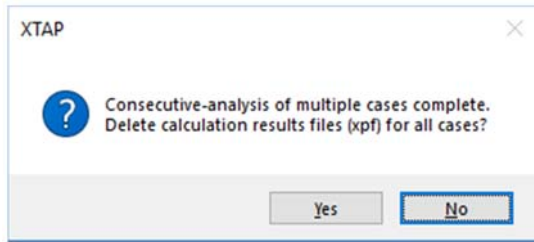
Press the 'analysis-result consecutive processing' button to open the following screen. Select the file ([File name] + 'all.csv') containing the file names for all the waveform data, and click the 'Open' button. This operation can also be performed by double-clicking on the file name.



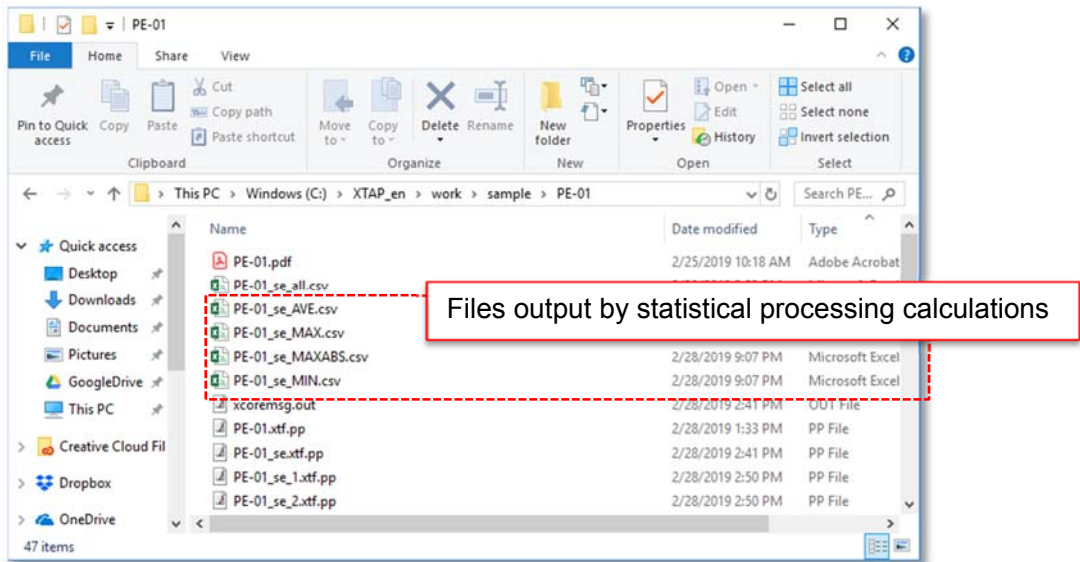
Statistical processing is executed on each waveform file.



The following window then appears, asking whether or not the calculation result files are to be deleted. Click 'Yes' to delete the files, or 'No' to retain the files, for example to create graphs.



The statistical-process result files (*.csv) are output to the folder where the calculation was executed. The file containing the maximum-value (MAX) results is named [Analysis file name] + '_MAX.csv'. The files for the other results are named in a similar fashion.



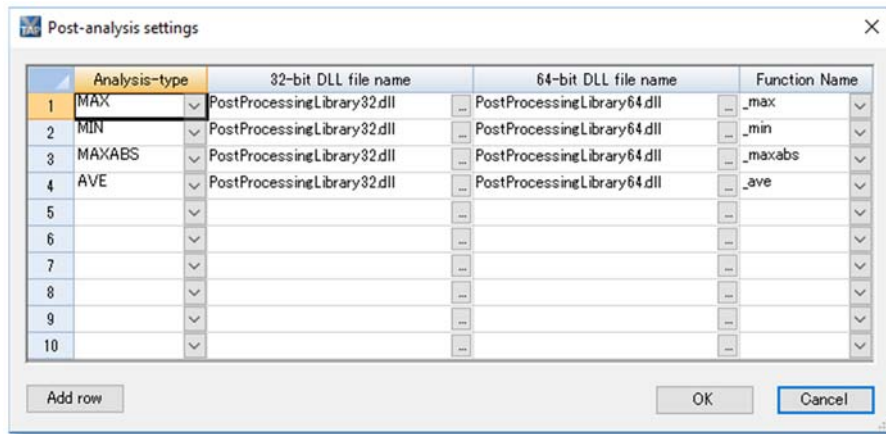
The information contained in the file is shown below. This comprises the calculation results (one data item per waveform) for the analyzed waveforms, i.e., the waveforms that were selected for recording by ticking the appropriate checkbox and that were output to the *.xpf files.

Case Num	Node_u	Node_v	Node_w	Lu	Lv
1	50	50	50	6.84E-31	5.92E-31
2	100	100	100	1.37E-30	1.18E-30
3	150	150	150	-4.07E-16	-4.07E-16

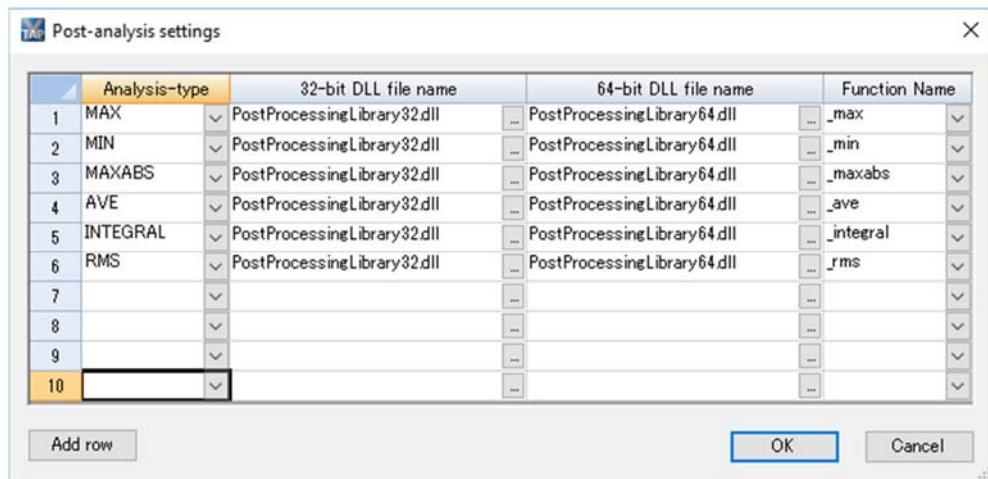
(2) Process settings

parameter	Case number	1	2	3	4	5	6	7	8	9
1 Param_A		20.4	40.8							
2 Param_B		33.4								
3 Param_C		22.3								

Pressing the ‘process settings’ button opens the following window. Enter the process type, 32-bit DLL file name, 64-bit DLL file name, and function name. Particular DLL file names can be selected via the ‘Browse’ button.



In XTAP, the two dll files “PostProcessingLibrary32.dll” and “PostProcessingLibrary64.dll” are installed by default. They contain the six operations listed previously: maximum (MAX), minimum (MIN), maximum absolute value (MAXABS), average (AVE), integral (INTEGRAL), and effective value (RMS). Accordingly, to perform any of these six operations in statistical processing, they must be specified as shown below, to be executed in the consecutive-processing of analysis results.

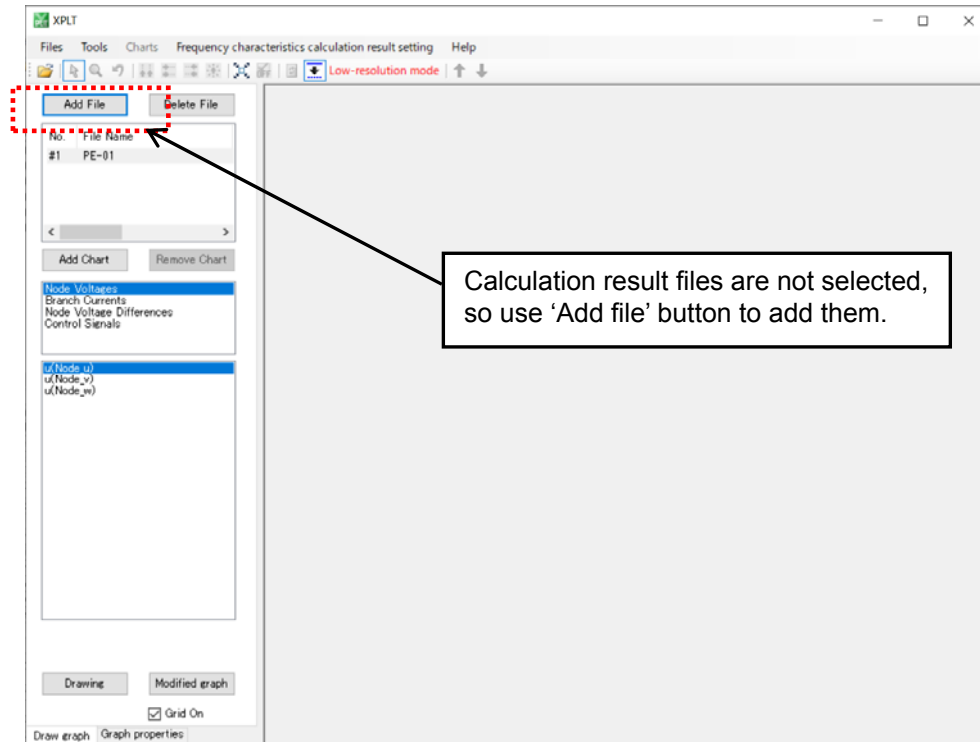


Rows, columns, and cells can be copied and pasted.

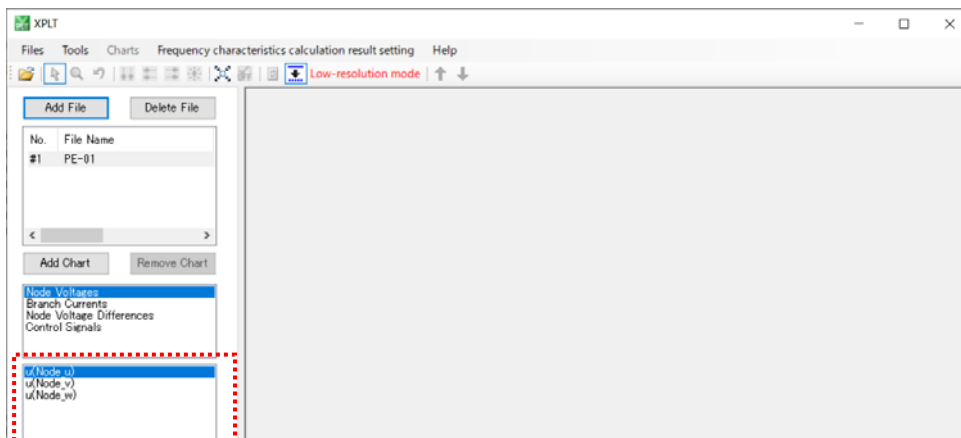
When a user-created function has been created, it can be executed by: ① entering the analysis type name, ② specifying the created DLL, and ③ entering the function name defined in the created DLL.

7.10 Displaying analysis results in XPLT

When XPLT is started after finishing statistical calculations, the calculation-result files are not automatically selected. The calculation-result files to be displayed can be entered in XPLT using the 'Add file' button.



When displaying calculation results, produced by the consecutive-analysis function, in XPLT, the specifier 'AUTOSUB/' is added to the element name.



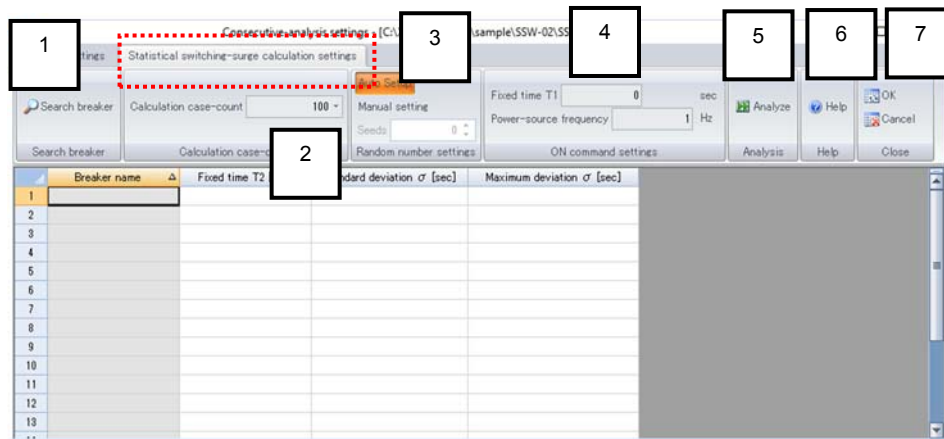
8 Statistical Switching-Surge Calculation Function

8.1 Overview of statistical switching-surge calculation function

The statistical switching-surge calculation function statistically analyzes the magnitude of an input surge overvoltage by randomly varying the breaker ON timing and performing many input-surge calculations. This analysis function automatically performs all these calculations. Specifically, it lets the user set parameter values that are related to the ON timing, perform many calculations automatically, and perform statistical processes on the analysis results.

8.2 Opening the settings window for a switching-surge calculation

Open the settings window for a switching-surge calculation by selecting the 'Statistical switching-surge calculation settings' tab in the 'case setting' window.



1. Breaker-search menu

- used to search for breakers (switch parts with breaker logic) in the circuit.

2. Calculation case-count menu

- used to specify the number of calculations to perform.

3. Random number setting menu

- selects the method of determining the breaker ON command timing for each case. The ON timing is determined from uniformly distributed random values. Either of the following two methods can be selected for generating these uniformly distributed random numbers:

- Automatic: The seed value used to generate the random numbers is changed each time the analysis is executed.
- Manual: The random numbers are calculated using a user-set seed value.

4. ON command setting menu

Using the timing from $t = 0$ until the fixed time T1 set here has elapsed as the reference, the ON command timing uniformly distributes the range of cycles found from the set power-source frequency. The random numbers in the distribution are the uniformly distributed random numbers set in the 'random number setting' menu (Item 3 above).

- Fixed time T1: Specify the reference time for the breaker ON time.
- Frequency: Specify the range of values.

5. Analysis menu

- executes the analysis.

6. Help menu

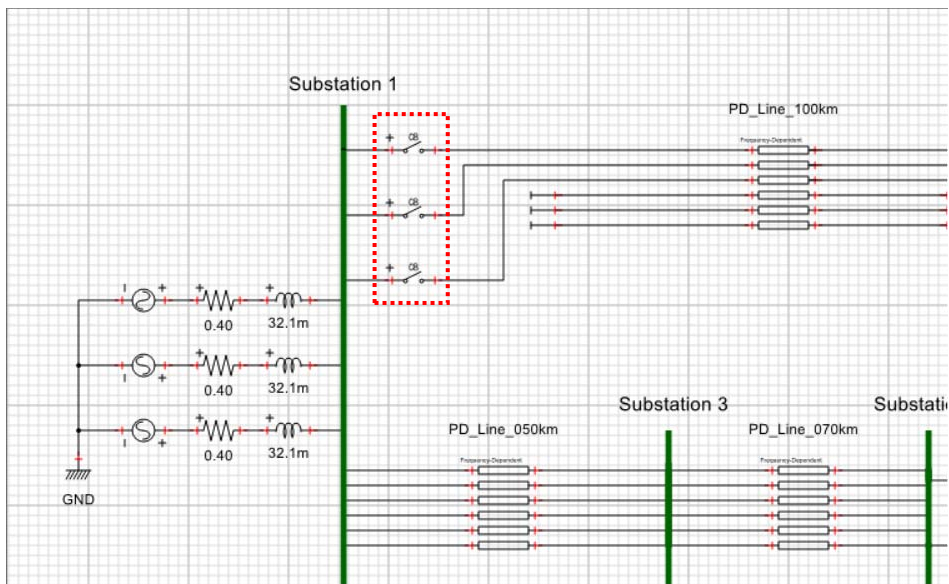
- displays the help information (user manual).

7. Close menu

- saves the settings entered in this window upon clicking 'OK', or cancels them upon clicking 'Cancel'. The 'X' button at the top-right corner of the window performs the same operation as the 'Cancel' button.

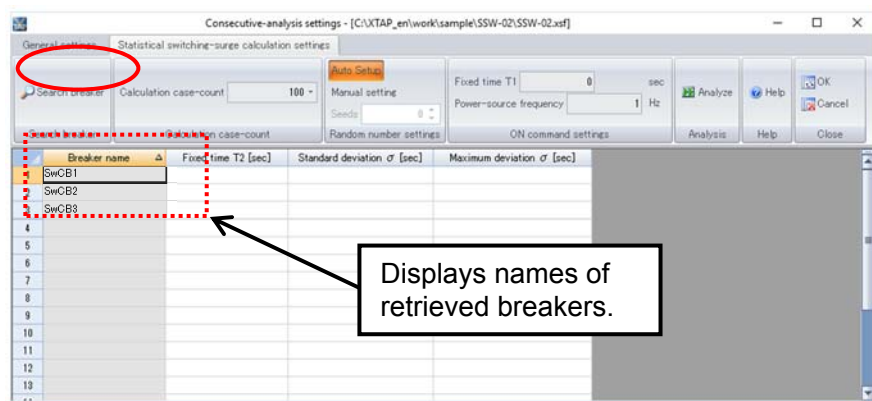
8.3 Analysis example

This section provides an example of a statistical surge-switching calculation for the following circuit consisting of three breakers.



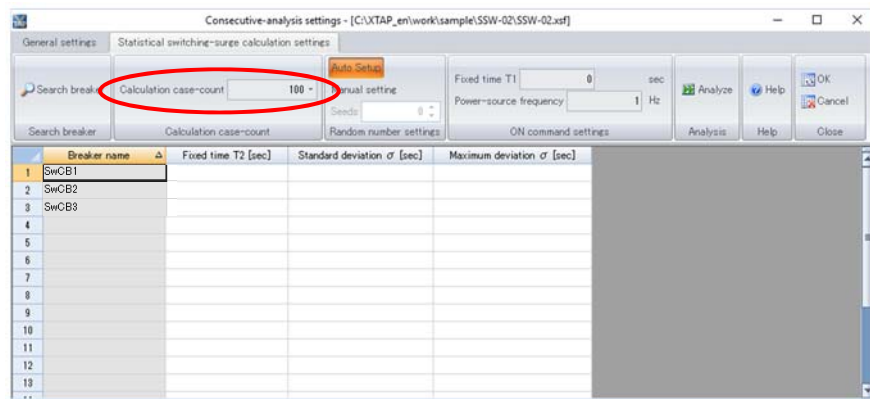
(1) Breaker search

Click the 'Breaker search' button in the case setting window to search for breakers in the circuit.



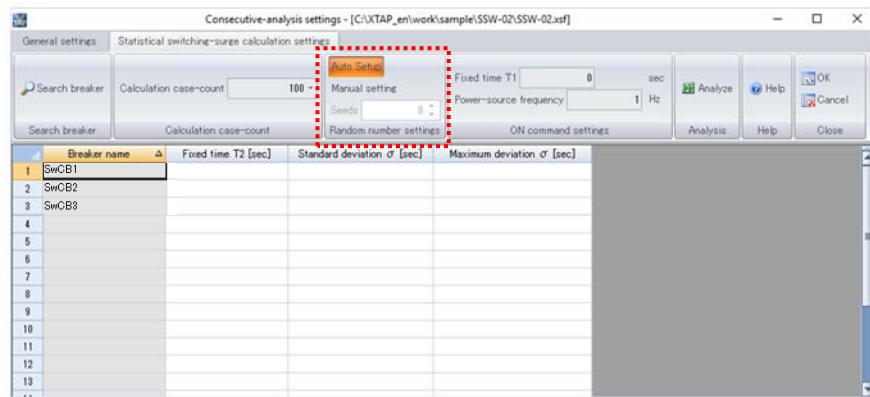
(2) Case number specification

Specify the number of cases to calculate.



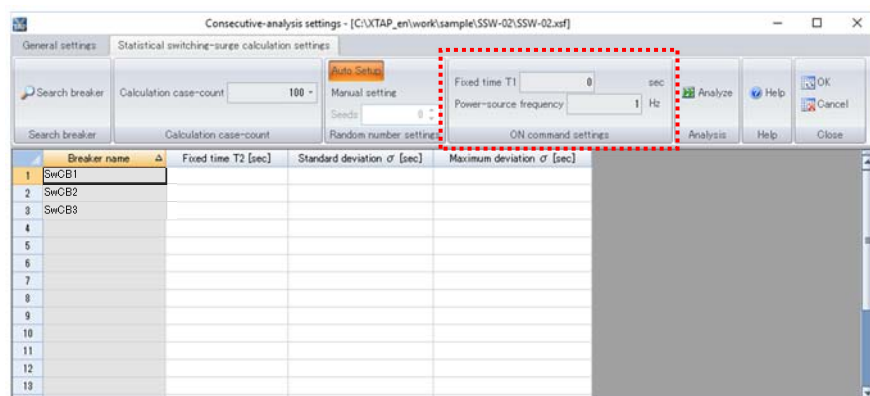
(3) Random number setting

Specify the method for calculating random numbers. When selecting the manual method, enter a seed value.



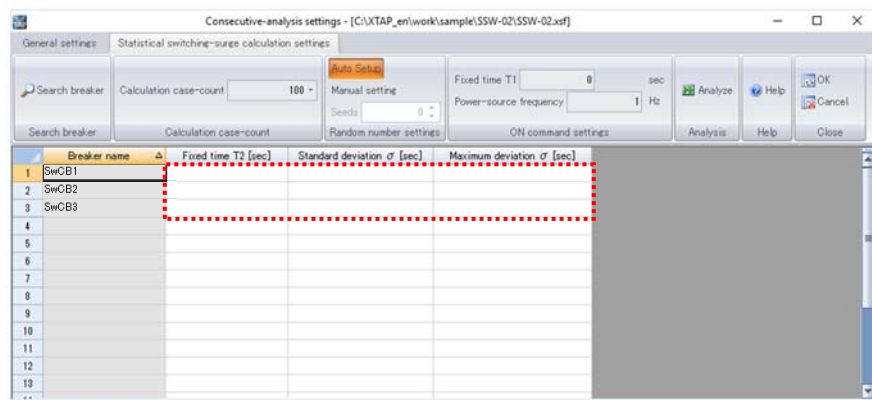
(4) ON command settings

Enter the values for the parameters that specify the breaker ON time.



(5) Breaker parameter input

Enter the values for the parameters that determine the breaker ON timing.



Fixed time T2:

Specify the time lapse from the ON command until the breaker operates.

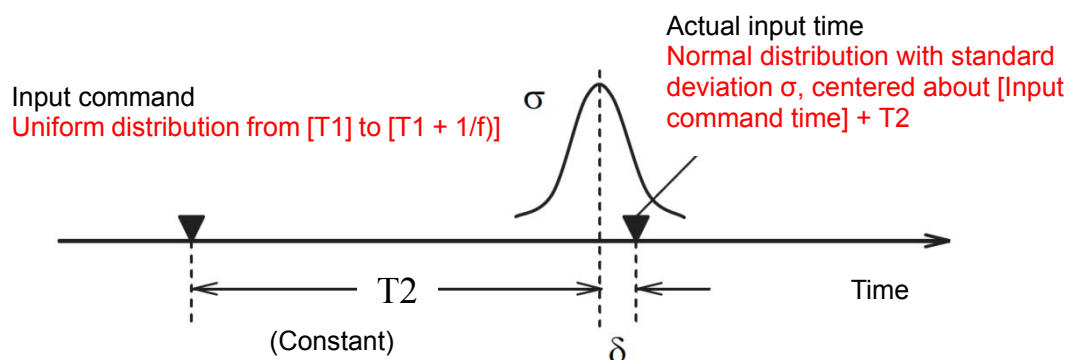
Standard deviation σ :

Specify the standard variation in the breaker operation time.

Maximum deviation σ_{\max} :

Specify the maximum standard variation.

The breaker operates according to the following timing.

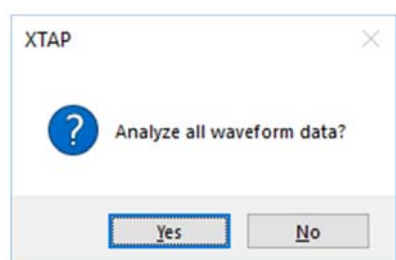


(6) Analysis execution

Click the 'analysis execution' button to execute the analysis.

(7) Statistical processing of analysis results

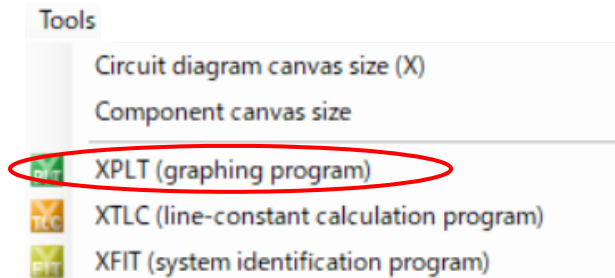
Statistical analysis starts automatically when the analysis ends. (For the consecutive analysis described in Chapter 7, the statistical analysis must be done manually. For the statistical switching-surge calculation, the program proceeds to statistical processing after the analysis has finished.) Click 'Yes' to start the statistical processing. The analysis and output files for this processing are the same as those described in Chapter 7. See Chapter 7 for further information.



9 XPLT (Graphing Program)

9.1 Starting XPLT

To start XPLT from XTAP, select 'XPLT (graphing program)' from the 'Tools' menu in the main window.



The same operation can be performed by clicking the toolbar button indicated below.

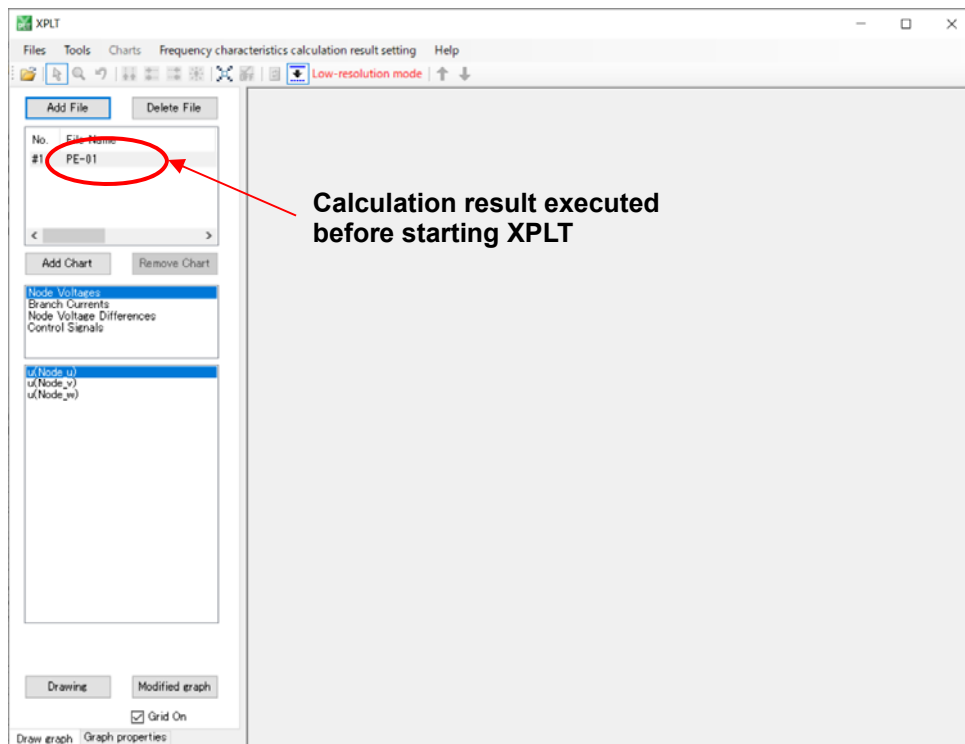


9.2 Selecting the analysis result files

When XPLT starts, an analysis result file (*.xpf file) is selected if the analysis was executed before starting XPLT. Otherwise, or to select a different analysis result file or to add files, click 'Add file' and select the desired file from the 'Open file' window.

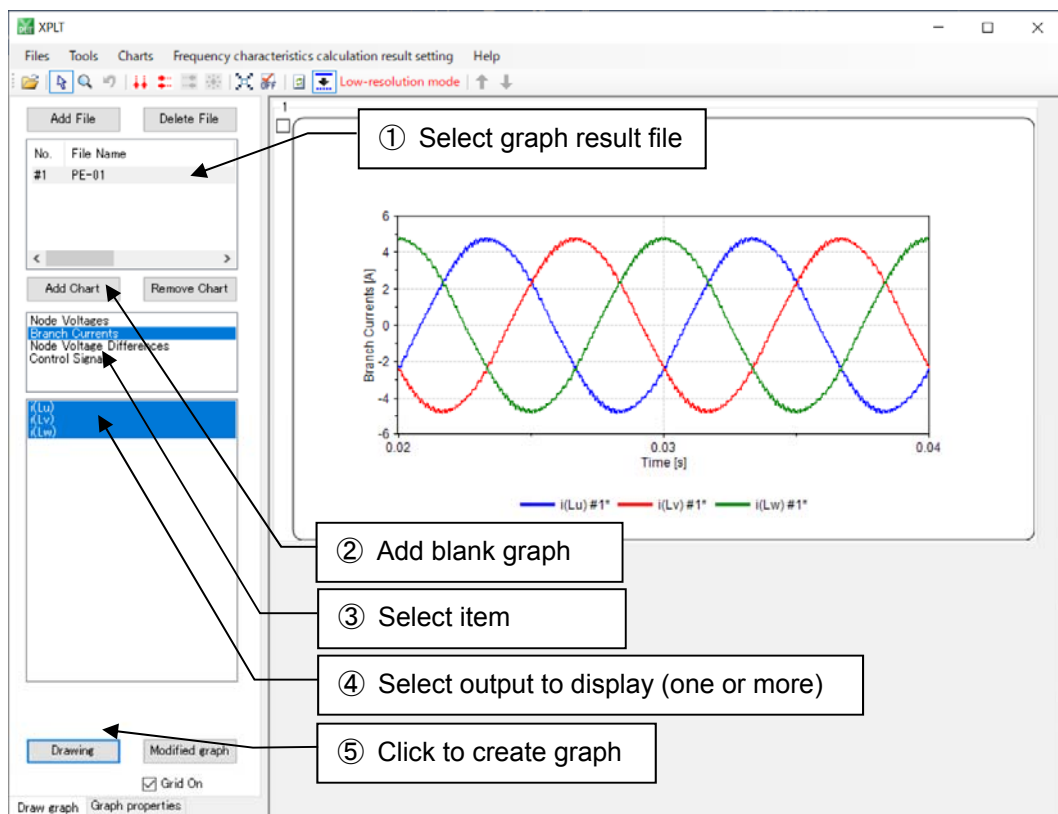
Note that when attempting to select an analysis-result file for which the analysis was done without checking the 'Recording' option for any part, there will be no data to output. The message 'No data to output' therefore appears, and the analysis result file is not selected.

Multiple analysis-result files can be selected, enabling the data from each file to be plotted onto a single graph.



9.3 Displaying graphs

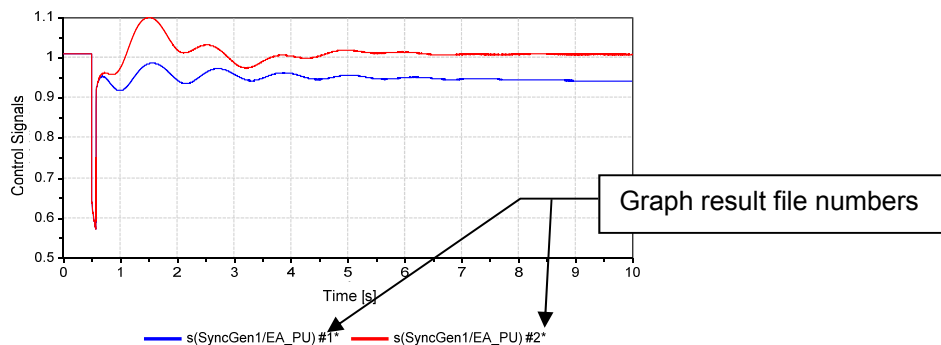
There are two types of graph displays: normal (plotting unmodified outputs) and modified (plotting modified outputs). The normal graph displays are themselves of two types: a transient-analysis graph display (with time on the horizontal axis), and a frequency-characteristic calculation graph display (with frequency on the horizontal axis).



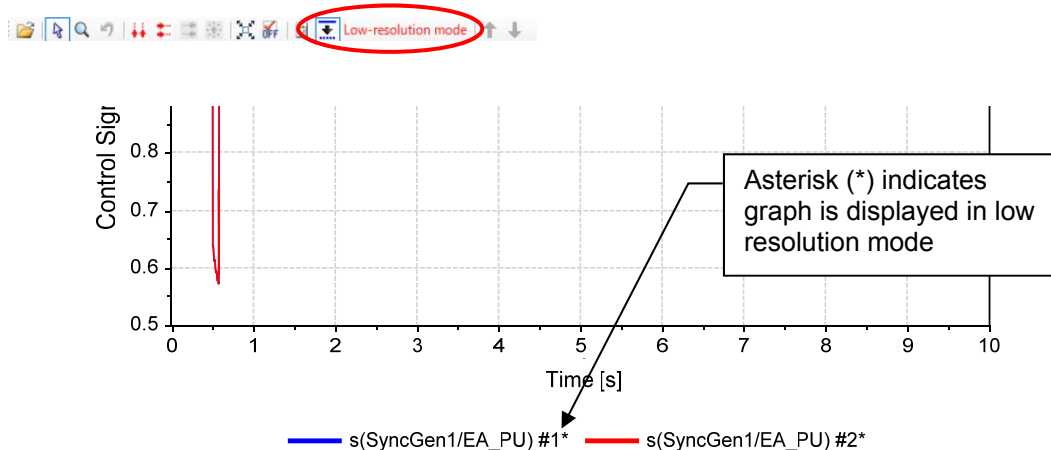
(1) Normal graph display

- ① Select a graph result file. Only one graph file can be selected.
- ② Click the 'Add graph' button to add a graph area. Do not click this button when adding to a graph area that is already displayed.
- ③ Select the item to display.
- ④ Select the output to display. Multiple outputs can be selected by pressing the Control or Shift key.
- ⑤ Click the 'Graph' button.

The output from multiple result files can be displayed onto a single graph. To display a second output, select a different graph result file in Step 1 above, and plot it in the same graph area. The numerical labels shown in the legend below the graph (i.e., #1 and #2) refer to the numbers written to the left of the file names in the window.



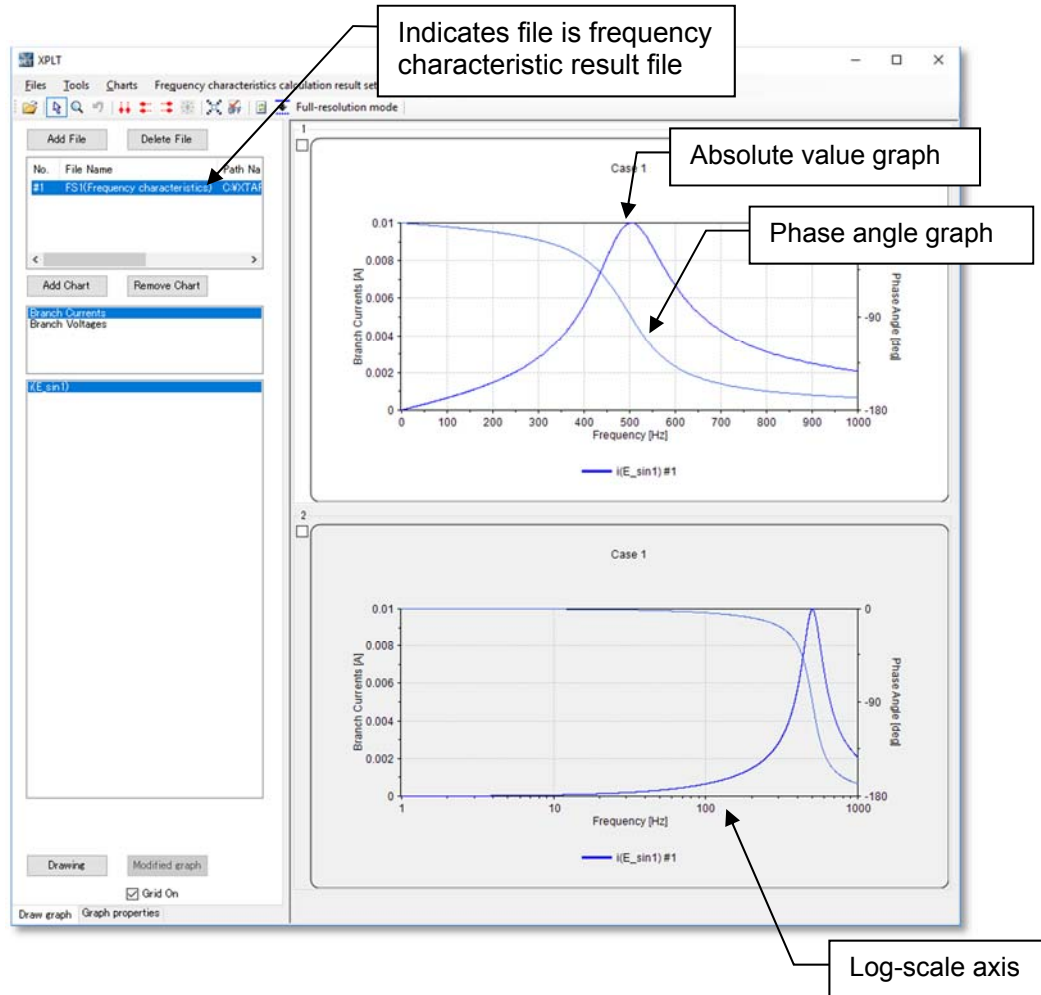
Loading and plotting large amounts of data are a slow process. When the number of data points exceeds the specified number, it is possible to omit some data points from the display (low-resolution mode), or to display all the data points (full-resolution mode). Low-resolution graphs are denoted by an asterisk after the file number in the label below the graph.



The default setting is to cull points if there are 10,000 or more. To switch between low- and full-resolution modes, select 'Tools' from the 'File' menu, or click the corresponding toolbar button. The low-resolution mode is also described in Section 6.5.

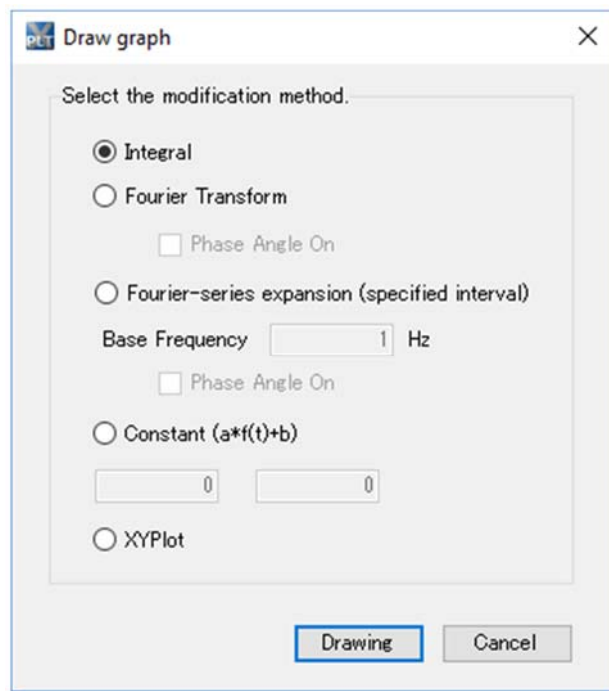
(2) Graph display for the frequency-characteristic calculation

When a frequency-characteristic calculation result file is selected, the words 'frequency characteristic' are appended to the file name and the graph is created with frequency on the horizontal axis. The absolute value and phase are displayed on the vertical axis by default. The horizontal axis has a logarithmic scale if that option was chosen for frequency sampling. Modified graph displays are produced in the same way as normal graph displays.



(3) Modified graphing

Modified graphs involve the 5 following actions.



① Integration

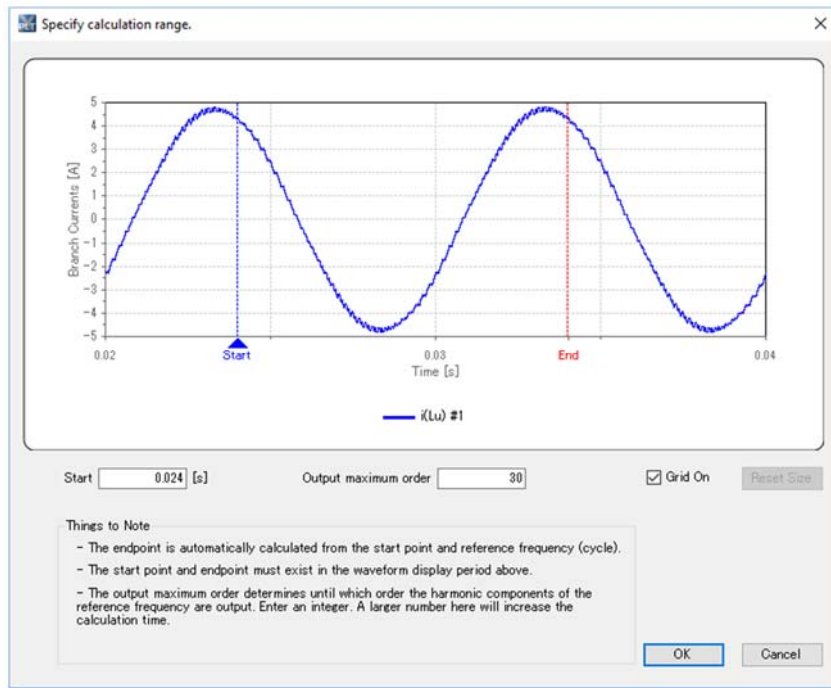
- displays the integrated result of the selected output.

② Fourier transform

- displays the Fourier transform of the selected output. The phase angle is also displayed when the 'Phase angle display' checkbox is checked.

③ Fourier-series expansion (specified interval)

- performs a Fourier-series expansion over a specified range of the selected data. The phase angle is also displayed when the 'Phase angle display' checkbox is checked. The calculation range is set with the endpoint determined automatically from the specified reference frequency once the starting point has been set (moved left or right).



④ Constants ($a \cdot f(t) + b$)

- multiplies the selected output by a specified constant a and adds a specified offset b .

Draw graph

Select the modification method.

☐ Integral

☐ Fourier Transform

☐ Phase Angle On

☐ Fourier-series expansion (specified interval)

Base Frequency 1 Hz

☐ Phase Angle On

☒ Constant ($a \cdot f(t) + b$)

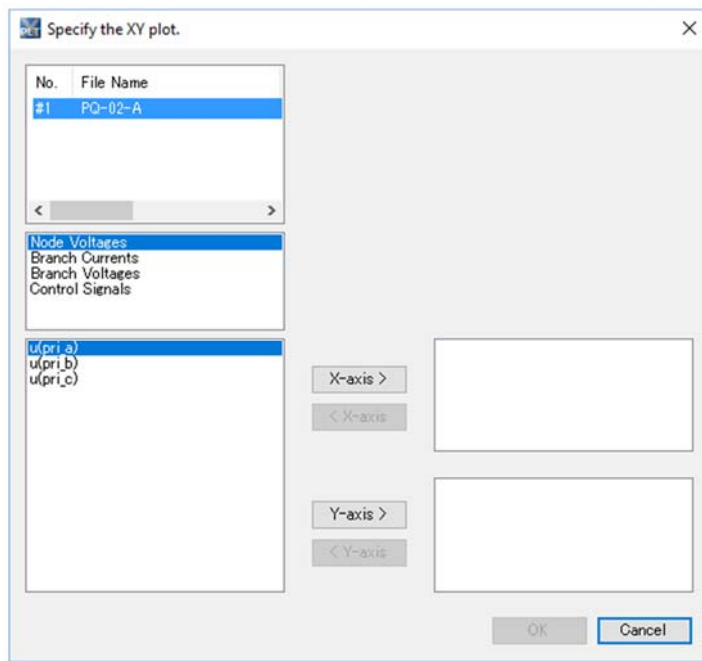
5 3

☐ XYPlot

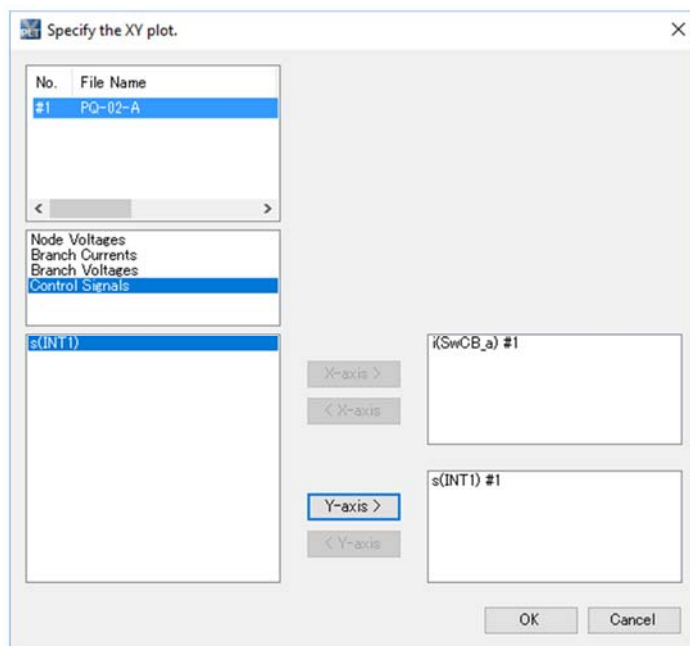
Drawing Cancel

⑤ XY plot

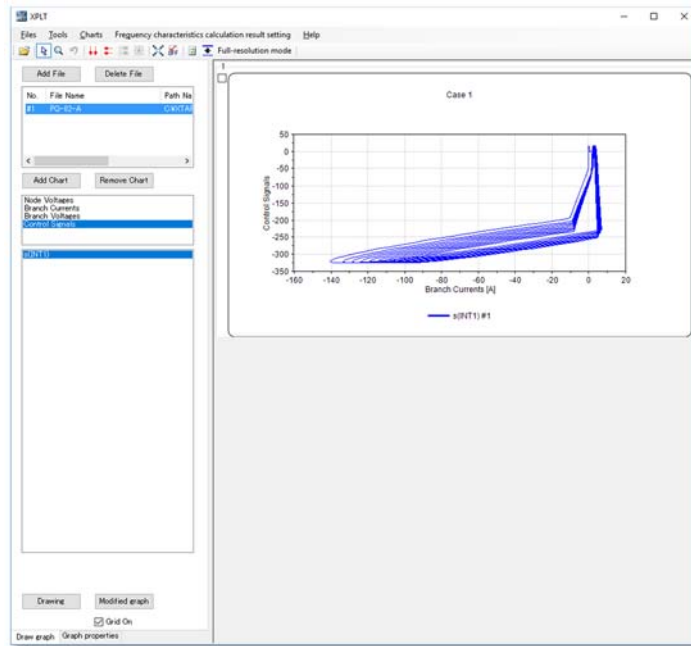
- selects two signals and plots a graph by using one as the horizontal axis and the other as the vertical axis (for instance, this can be used to render the $i-\Phi$ characteristics from the current and magnetic flux). Click the 'Plot' button to display the following screen. Select the signals here in the same way as on the main XPLT screen, and set them for use as the X and Y axes.



Only one signal may be selected for the X axis (once it is selected, the 'X axis >' button will be deactivated). Multiple signals may be selected for the Y axis.



Click the OK button to display an XY plot as shown below.

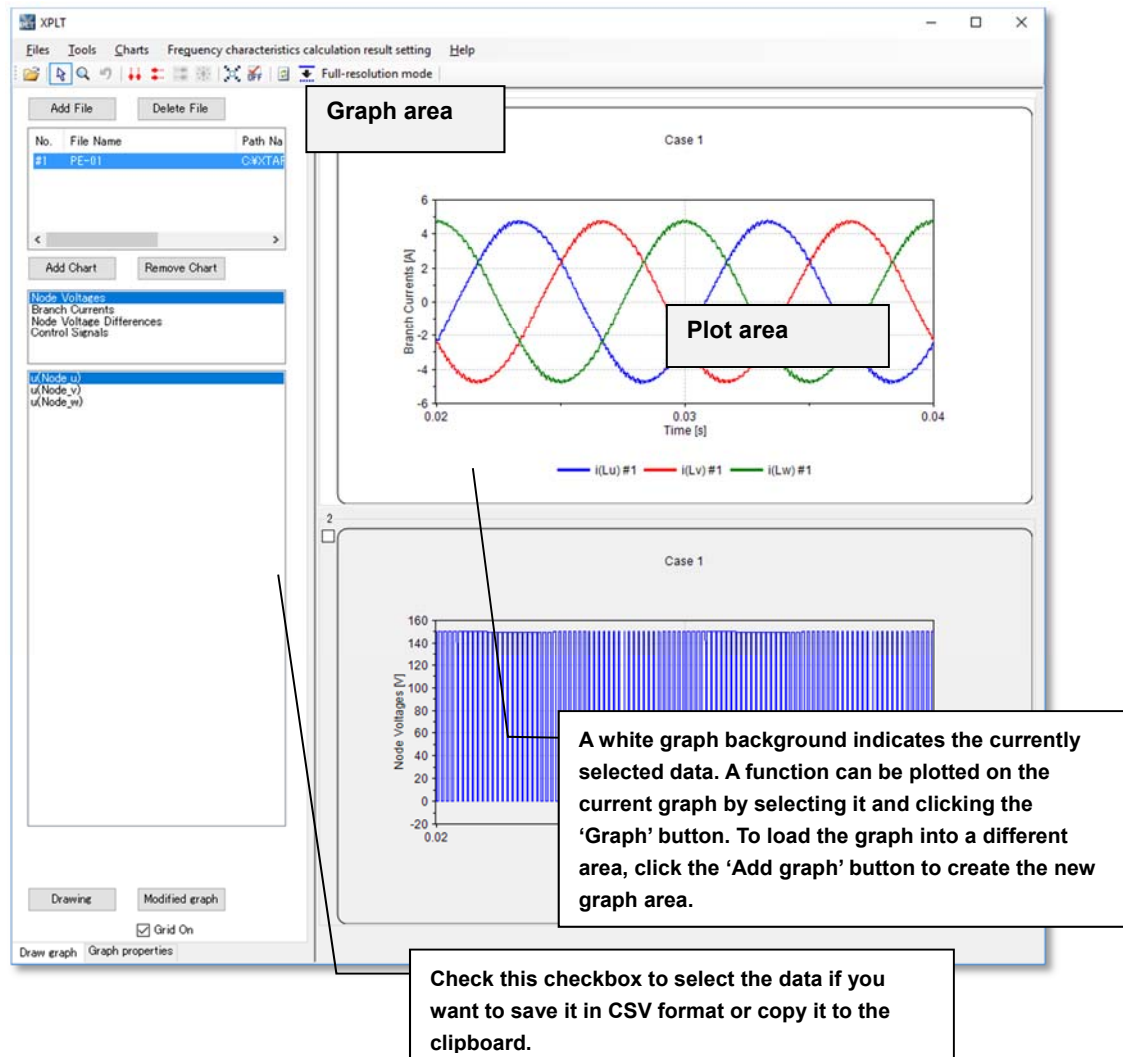


(4) Panel button operations

The screenshot shows a software interface with the following components and callouts:

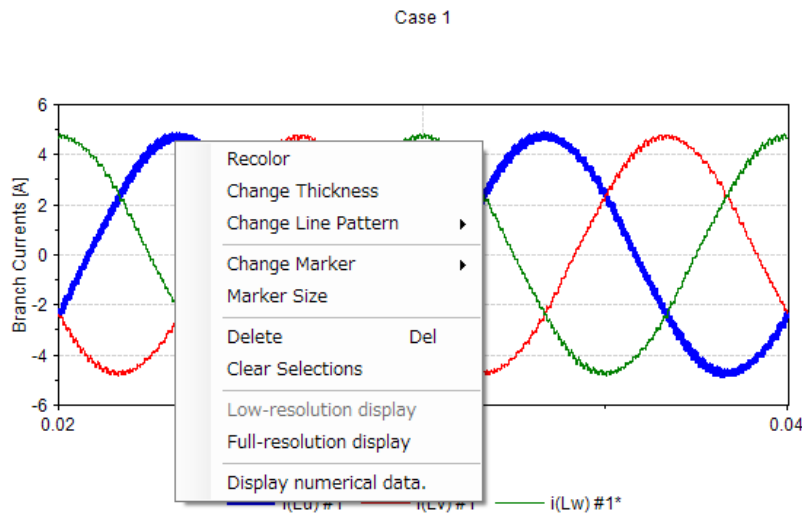
- Add File**: Adds analysis-result files to the list.
- Delete File**: Deletes the selected analysis-result file from the list.
- File List**: A table with columns 'No.', 'File Name', and 'Path Na'. It contains one entry: '#1 PQ-02-A C:\EXTA...'. Callout: Displays the added analysis-result files. This is used to select the file to graph when multiple files have been added. Only one file can be selected.
- Add Chart**: Adds one graph screen.
- Remove Chart**: Deletes one graph screen.
- Graph Type List**: A list containing 'Node Voltages', 'Branch Currents', 'Branch Voltages', and 'Control Signals'. Callout: Selects the item to be plotted. Only one item can be selected.
- Plot Labels**: A list containing 'u(pri_a)', 'u(pri_b)', and 'u(pri_c)'. Callout: Selects the results to output. Hold down the Shift or Control key on the keyboard to select multiple results.
- Vertical Boundary Line**: A vertical black line. Callout: Drag an item with the mouse to move the boundary line.
- Drawing**: A button. Callout: After selecting items, click this button to create the graph.
- Modified graph**: A button. Callout: Click this button to display the result of integration or a Fourier transform of the displayed graph.
- Grid On**: A checkbox. Callout: Check/uncheck to enable/disable grid line display.
- Draw graph**: A button.
- Graph properties**: A button.

(5) Graph display area



9.4 Graph editing

Select any waveform or the legend of the graph and right-click with the mouse to display a menu. The plot line then displayed with a thick line is the selected waveform.



Change color

The color of the selected curve can be changed. Select 'Change color' from the menu to open the 'Color settings' dialog. Select the desired color and click 'OK'. The curve changes color accordingly.

Change thickness

You can change the line thickness of the selected curve. Select 'Change thickness' from the menu and select the desired thickness magnification.

Change Line Pattern

It is possible to change the line type of the plotted graph. Choose one line type (solid line, dash line, etc) from the menu options.

Change Marker

It is possible to add a marker to a waveform (the marker is set to 'None' by default).

Marker Size

You can set the marker size when adding markers to the waveform.

Delete

Selecting a curve in the graph by clicking on it gives it a thicker line width. Right-click and select 'Delete' from the menu to delete the selected curve. The graph area still remains even if all the curves have been deleted. To delete the graph area also, click the 'Delete graph' button in the panel on the right.

Clear Selections

This menu item deselects a selected curve, reverting its line width back to the normal.

Low-resolution display

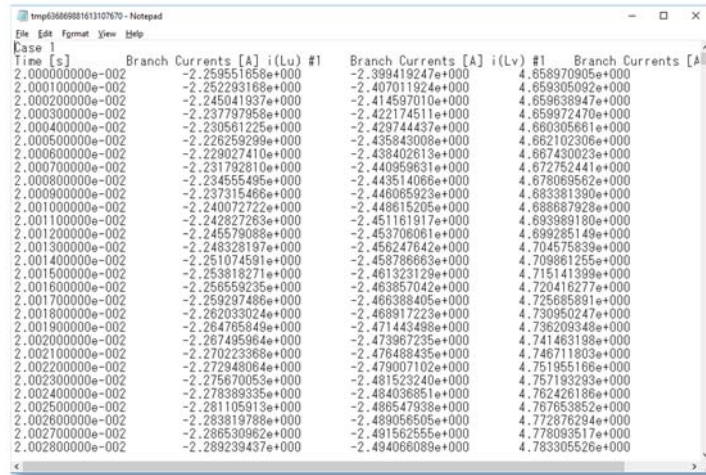
This option switches the display of a curve from full to low resolution. This item cannot be selected when the curve is already displayed in low-resolution display mode.

Full-resolution display

This option switches the display of a curve from low- to full-resolution. This item cannot be selected when the curve is already displayed in full-resolution display mode.

Display numerical data

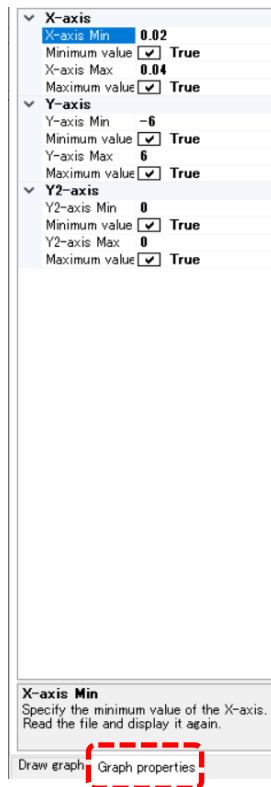
This option displays the numerical values for the selected curve in a text editor (Windows Notepad (notepad.exe) by default). The numerical data are displayed in tab-separated format, allowing copying and pasting to Excel or similar spreadsheet software. You can change the choice of text editor. (Windows Notepad is slow when displaying a large volume of data, so using a faster text editor is recommended.)



Right-clicking away from a curve but still in the graph area displays a menu containing only the 'Display numerical data' item. Selecting this item displays the numerical data for all the curves in the graph.

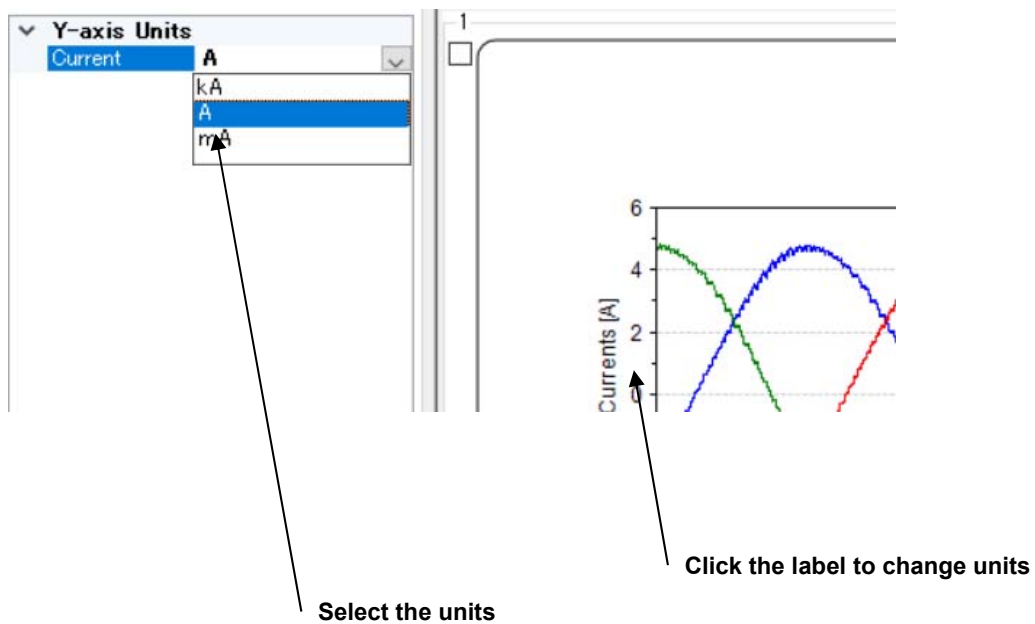
Editing the properties list

Clicking the 'Graph properties' tab at the bottom of the screen's left panel activates the properties-list mode, which displays the properties list for the current graph.



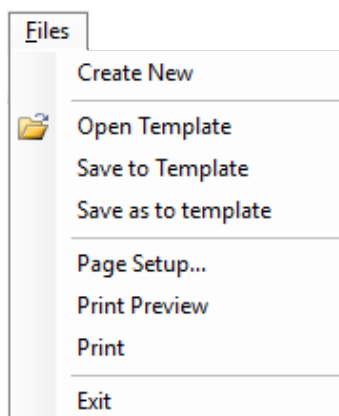
Maximum and minimum values can be set for the X- and Y-axes in the properties list. If different quantities (such as voltage and current) are plotted simultaneously on a single graph, the properties list for all the corresponding Y-axes are also shown.

Clicking on the X- or Y-axis graph label while in properties-list mode allows a change of the corresponding measurement units. It is also possible to change the units of the X- or Y-axis by clicking the legend of the X- or Y-axis, respectively.



9.5 Menus

(1) File menu

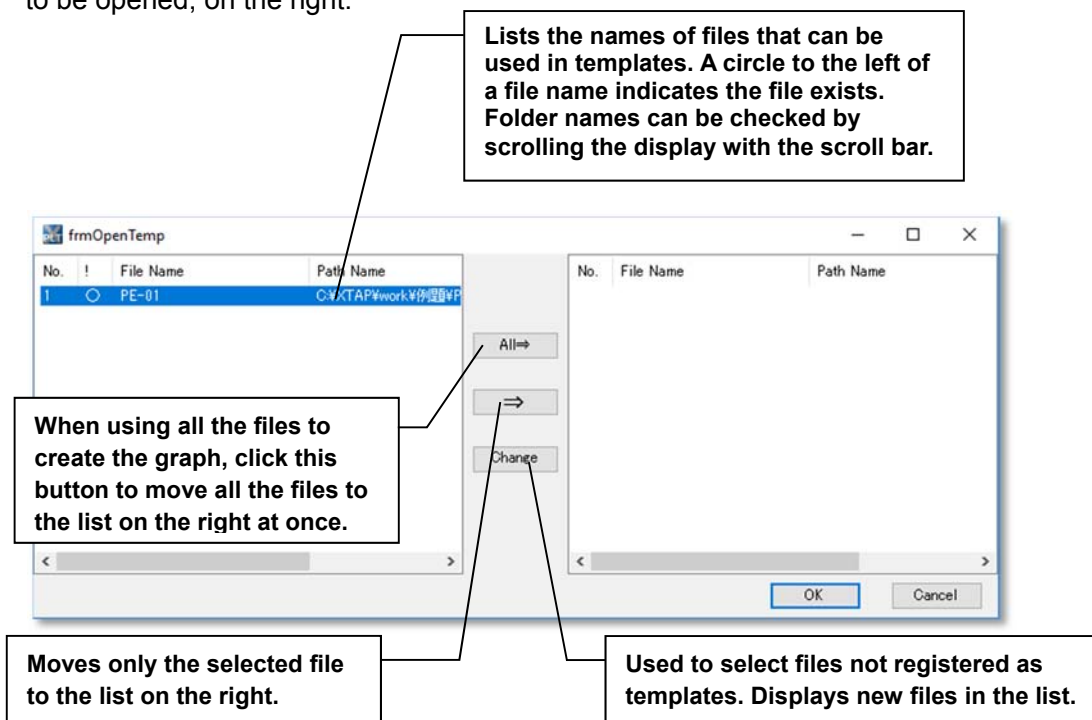


New

This option erases all the currently displayed graph data and displays a new graph.

Open template

A template is a group of files containing generic patterns of presentation that can be used as models for creating graphs of the same type. A template must be created from individual template files before it can be used. To open template files, select 'Open template' from the 'File' menu in the main window. The following file-selection screen appears. The arrow button is used to move the file selected on the the left to the list of files to be opened, on the right.

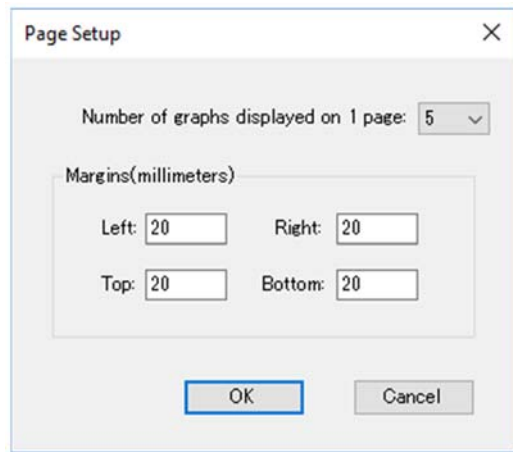


Save to template/Save as to template

After displaying the graphs, select one of these menu items to save the currently displayed graph items.

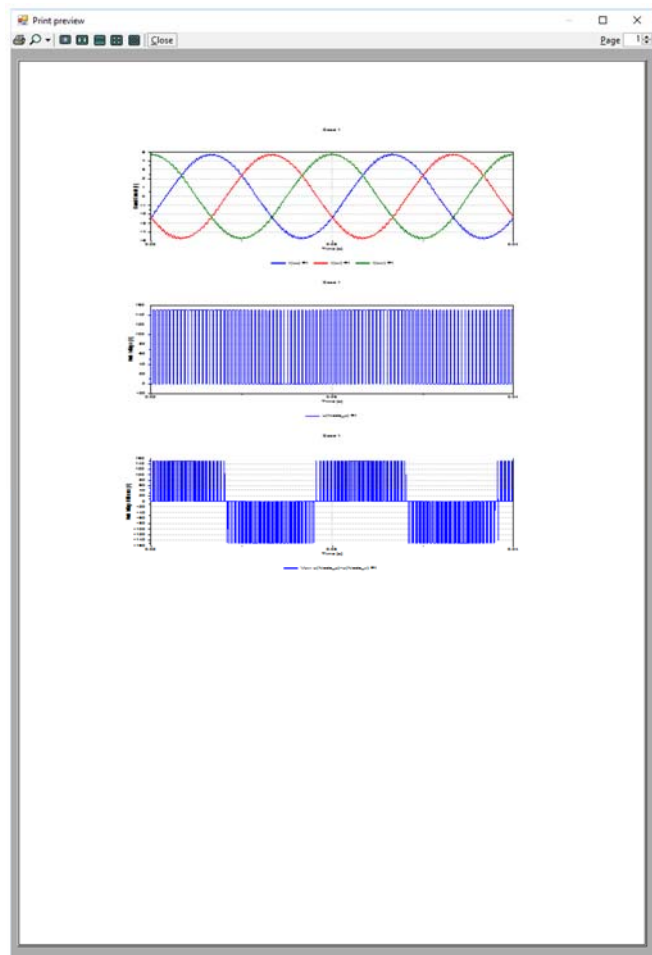
Page settings

This option is used to specify the number of graphs printed per page (i.e., on one sheet of paper) and the page margins.



Print preview

A preview is shown of how the currently displayed graphs will appear when printed.



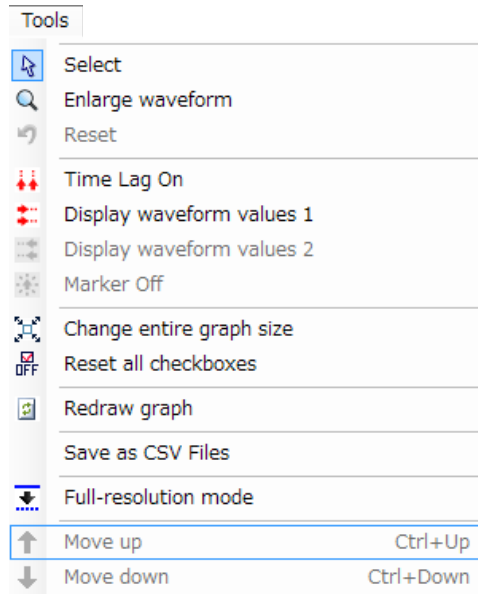
Print

The displayed graphs are printed. If any of the graphs have been checked, only the checked graphs are printed.

Exit

The application terminates.

(2) Tools menu



Selection

- enters the normal mode for selecting a waveform. Click this button to switch from the 'Enlarge waveform' mode described below to the normal mode.

Enlarge waveform

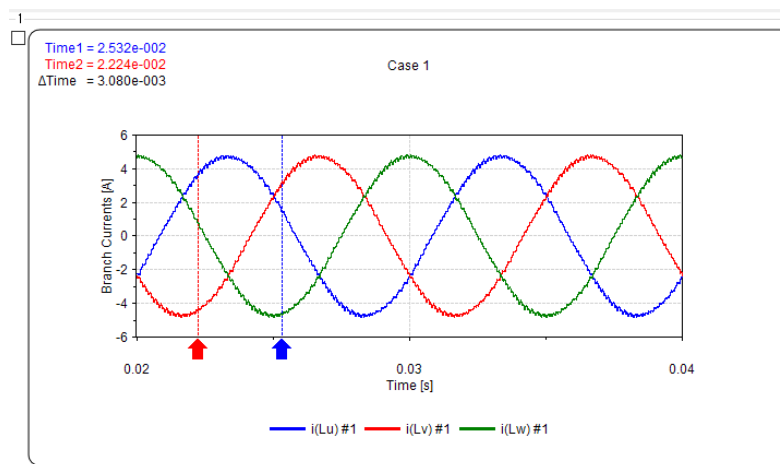
- enters the mode for enlarging a waveform. This mode causes the mouse cursor to change to a magnifying glass. Drag any part of the graph to zoom in to that region. Right-click to return to the normal selection mode.

Restore

- restores an enlarged waveform display to the normal display.

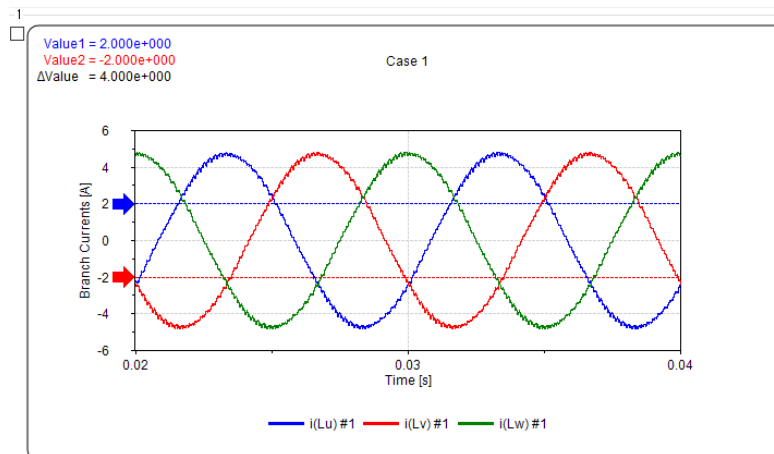
Display time difference

- displays two markers on the graph (red and blue). Use the mouse to drag the markers to the waveforms you would like to focus on. The times and differences indicated by the two markers are displayed in the upper left corner of the graph. Select [Delete marker] to delete that marker.



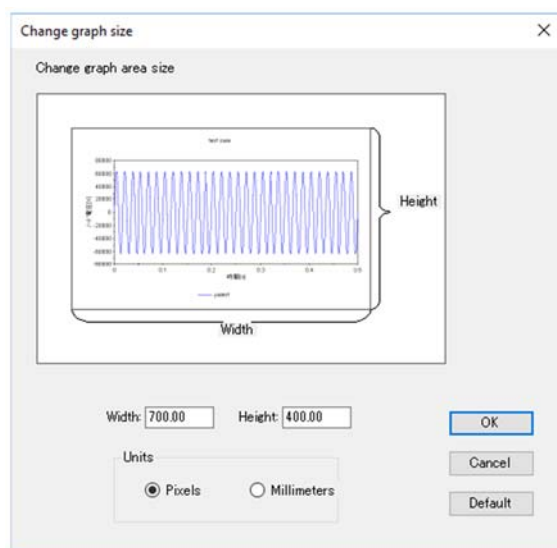
Display waveform values

Two ordinate value markers are displayed, showing the corresponding values and their difference.



Change entire graph size

- changes the display size of the entire graph.



Reset all checkboxes

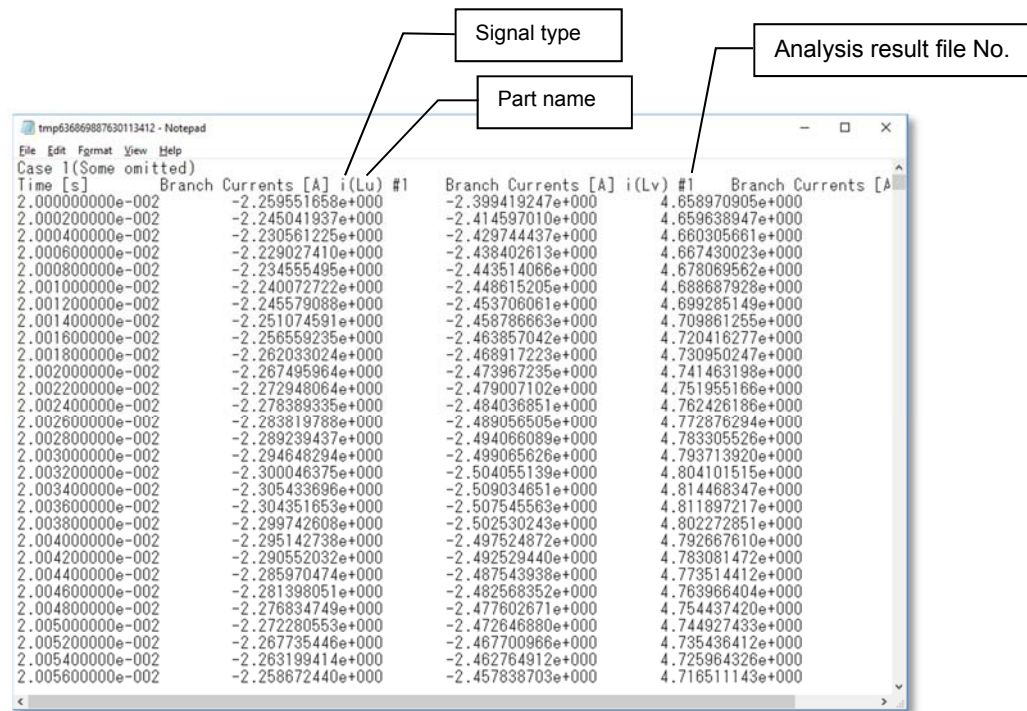
- resets all checkbox settings for the graph selection states.

Redraw graph

- reloads the data files of the displayed graph and redraws the graph. If the data in the analysis result files (*.xpf files) have changed because a calculation was redone, the currently displayed graph contents can be replaced with the new analysis results.

Save in CSV format

- saves the selected graph in CSV format. Selecting this item displays the file-saving dialog, where the user can specify a filename. When saving multiple graphs, a CSV file is created for each graph. The output for a graph saved in CSV format is shown on the next page. Note that saving a low-resolution graph in csv format causes that data to be culled. In this case, "culled" will appear on the first line of that file, and "*" will be appended to the item name of the culled data.



Low-resolution display mode/Full-resolution display mode

- switch between the low- and full-resolution display modes. The graph display toggles between these modes each time these items are selected. The current mode is displayed on the toolbar.

Toolbar in low-resolution mode:



Toolbar in full-resolution mode:



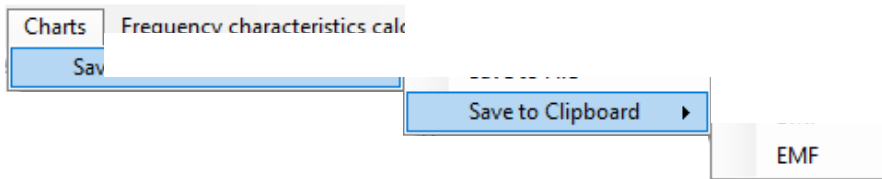
Move up

- Moves the currently active graph up one position.

Move down

- Moves the currently active graph down one position.

(3) Graph menu



Save graph image – Save to file

- saves the current graph as an image file such as a metafile or bitmap file.

Save graph image – Save to clipboard

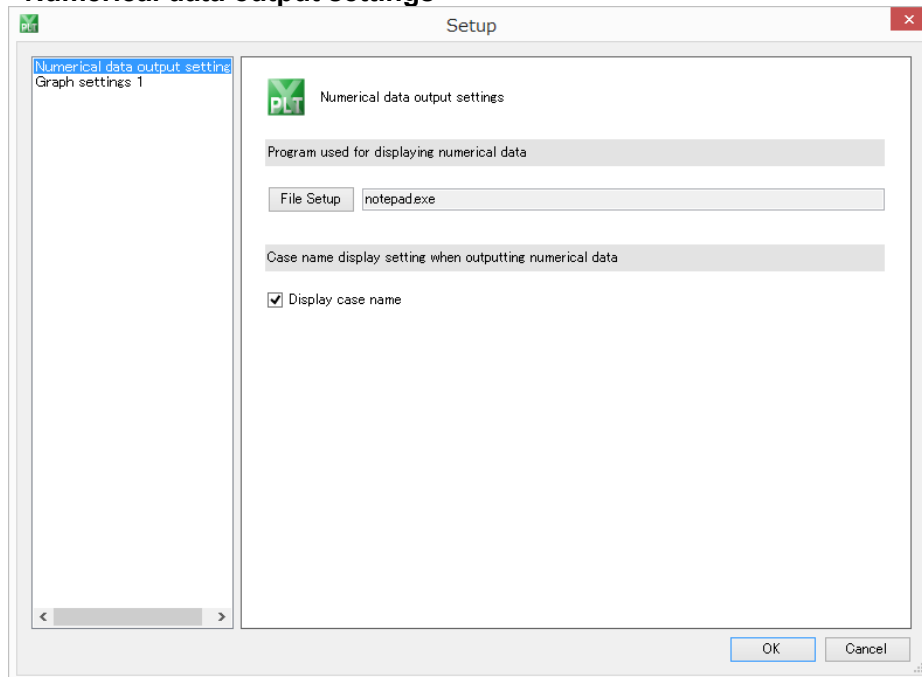
- copies the current graph to the clipboard in bitmap format or metafile format, allowing it to be pasted into, e.g., Microsoft Word or Excel.

(4) Setup menu

Setup

- Opens the setup window, where it is possible to change the settings of XPLT.

<Numerical data output settings>



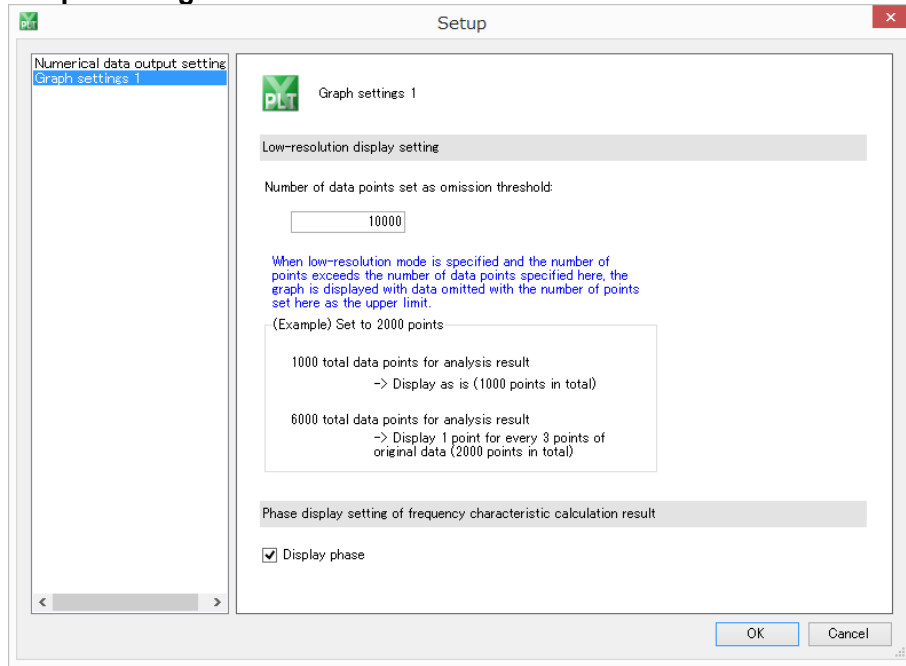
Program used for displaying numerical data

In this option it is possible to choose the program used for displaying numerical data. By default, the program used for displaying numerical data is the notepad (notepad.exe), which is installed by default in Windows. However, when opening large files in notepad it takes a long time to show its contents. Therefore, it is recommended to use a lightweight text editor.

Case name display setting when outputting numerical data

In this option it is possible to choose to show the case name of the graph in the first row of the csv file, when outputting numerical data to a csv file. When the checkbox is checked, the case name will be shown. When the checkbox is not checked, the case name will not be shown (in this case, the first row will be the data label and the data will be outputted from the second row)

<Graph settings 1>



Low-resolution display setting

In this option it is possible to specify the total number of points that triggers the low-resolution display setting. By default, it is set to 10000 points.

Phase display setting of frequency characteristic calculation result

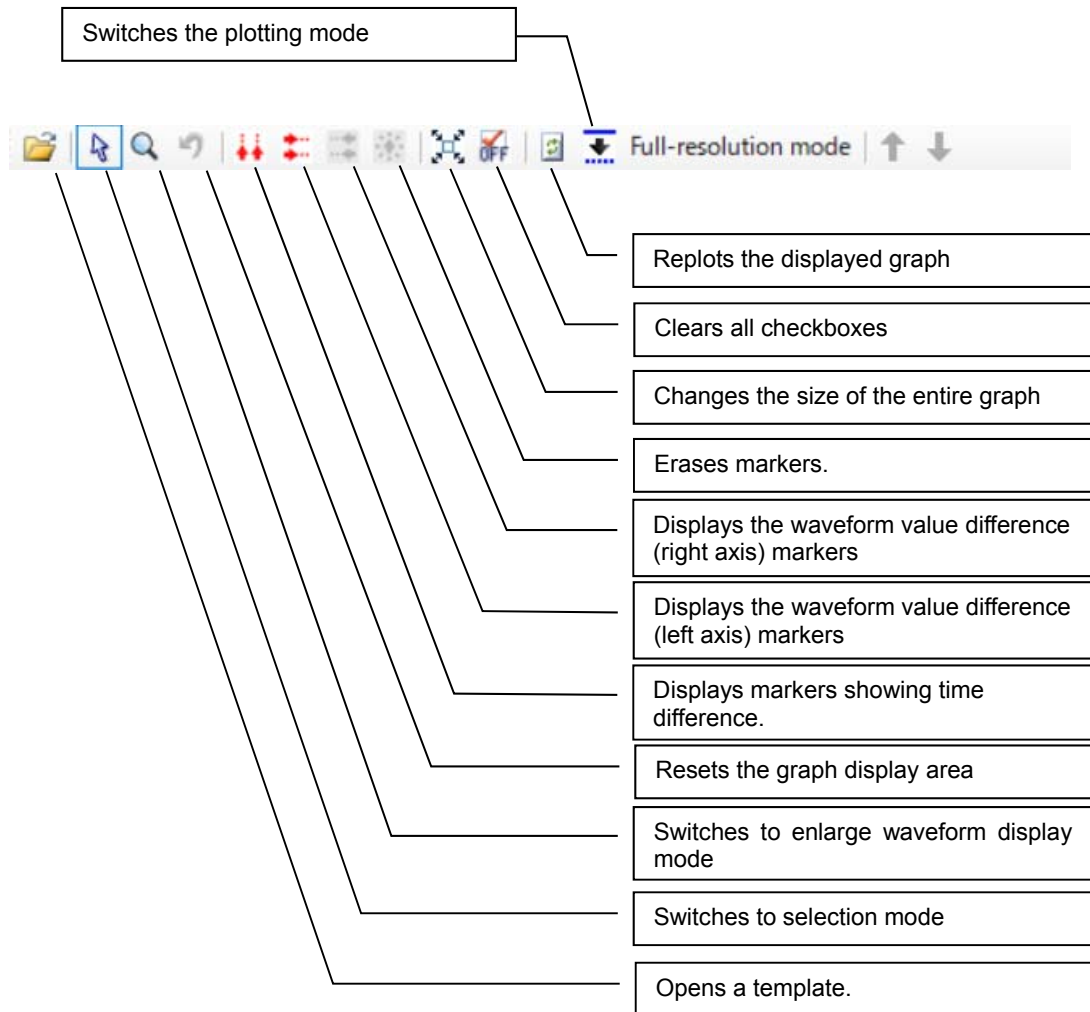
In this option it is possible to choose to display the phase information when plotting a frequency characteristic waveform. If the checkbox is checked, the phase will be displayed in the plot. If the checkbox is not checked, it will not be displayed.

(5) Help menu

About...

- Displays the XTAP version information

(6) Toolbar



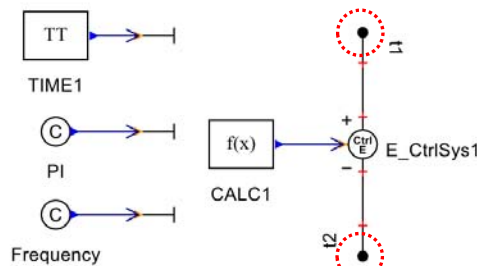
10 Creating and Editing Macro Parts

XTAP allows the creation (registering) of multiple circuit parts or control-block parts together as a single new part. Once a part is registered in this way, it can be reused as often as needed. When reused, its parameter values can also be changed.

10.1 Packaging procedure (creating macro parts)

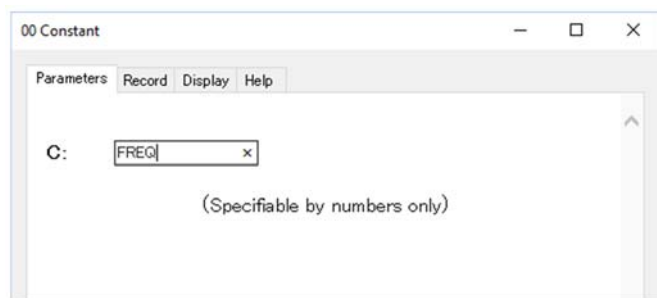
A created circuit can be 'packaged' to create a new part (a 'macro part').

1. Create a circuit diagram. Create appropriate terminals to connect the parts to each other by selecting the terminals from the '00 Installation/terminals' part category, and placing them in the diagram. In the example circuit diagram shown below, a value calculated in the control system is passed to the electrical circuit as a voltage source. The '01 Terminal (circuit)' part can be used to connect the terminals of electrical-circuit parts (circled in red below) to other circuits. The control-system terminals TIME1, PI, and 'Frequency' (circled in blue below) do not need to be connected to other parts; hence, use the '05 Terminating output terminal (control)' part to prevent terminals from being automatically appended to these parts when the circuit is packaged. If the circuit contains a terminal that is not connected to an output terminal or terminating terminal, the corresponding terminal is automatically appended to that element when the circuit is packaged.

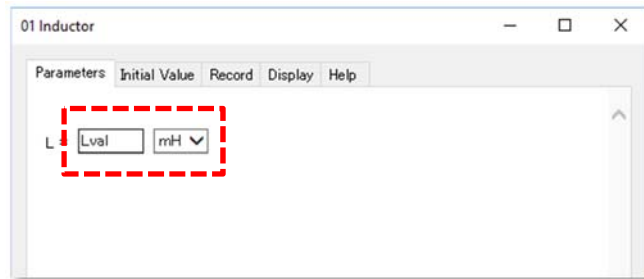


Specifying part parameters with text strings enables them to be specified as macro-part parameters once the circuit has been made into a macro part. In this example, a parameter is specified for the 'Frequency' terminal using the parameter name FREQ.

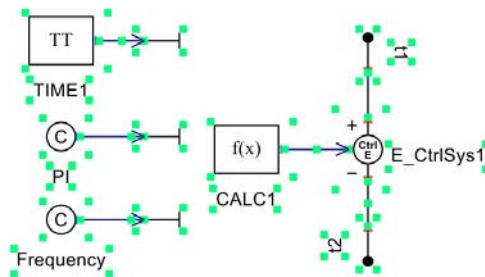
It should be noted that only alphanumeric characters (uppercase letters, lowercase letters or numbers) and underscores “_” can be used for the parameter names.



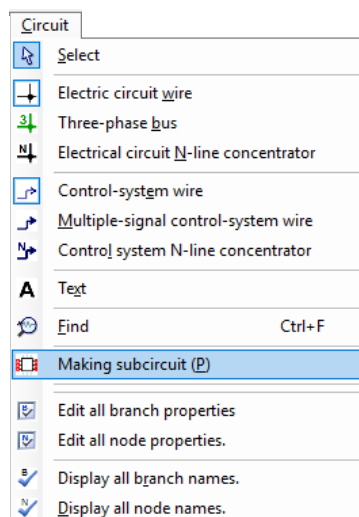
The following example screen shot shows a variable named 'Lval' being specified for the parameter of an inductor. It is important to note that *when a variable is specified for a parameter, the auxiliary units of that parameter (i.e., unit prefixes such as M, k, m or μ) are ignored*. The value of Lval can be set from the macro part's properties as described later, but for the present case, if Lval is set to 10.0E-3, the inductor value used in calculations will be 10.0E-3 [H] and not 10.0E-3 [mH].



2. Select the circuit area to be packaged. In this example, 'Select all' is selected from the 'Edit' menu to select the entire circuit diagram. The entire circuit can also be selected using the mouse.



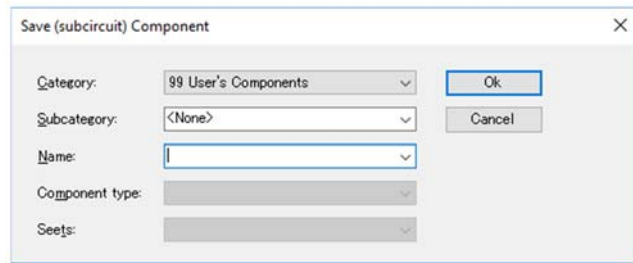
3. Select 'Package' from the 'Circuit' menu.



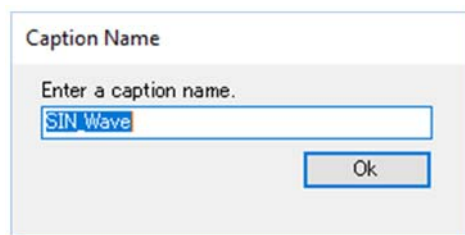
This operation can also be performed by clicking the toolbar button indicated below.



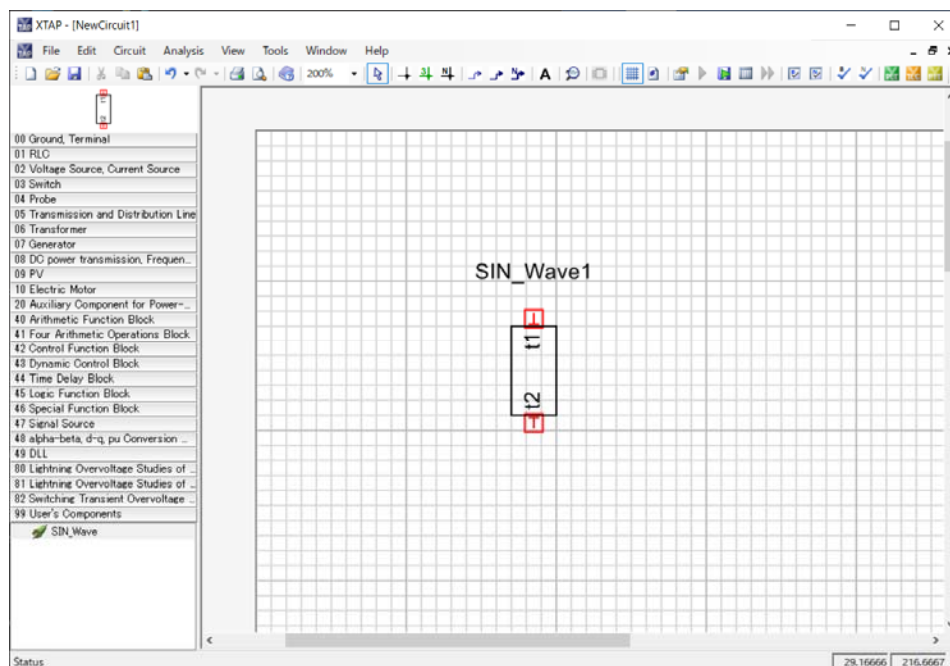
4. The following screen appears, allowing the macro part to be registered in the part-category window. **Only the category '99 User's Components' can be selected.** A subcategory can be selected only if it has already been created. See Section 10.6 for details of how to create and delete categories in the part-category window. Select the part name with which the macro part is to be registered, then click 'OK'. In this example, the part name 'Control SIN power source' has been selected.



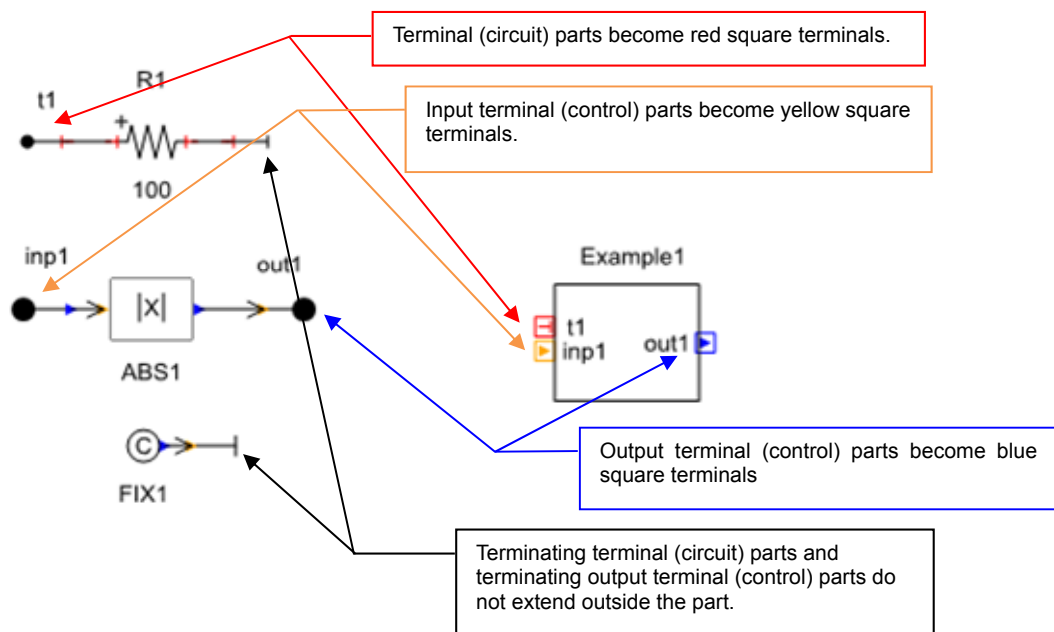
5. Specify the caption name that will be used to label the macro part on the canvas. This may include upper- and lowercase letters, the underscore character (_), and numbers. Caption names may not start with a number and **cannot contain Japanese characters or other Asian-width characters.**



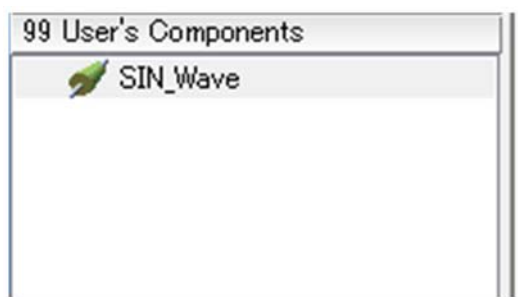
Click 'OK' to create the macro part on the canvas.



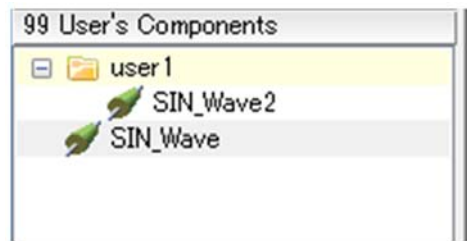
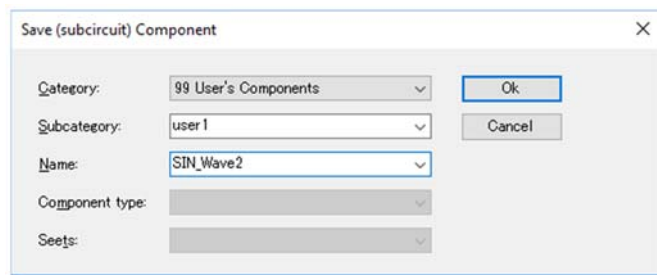
The original part type determines how the terminals of the created part are handled, as shown below.



6. The part created in the part-category window is registered.



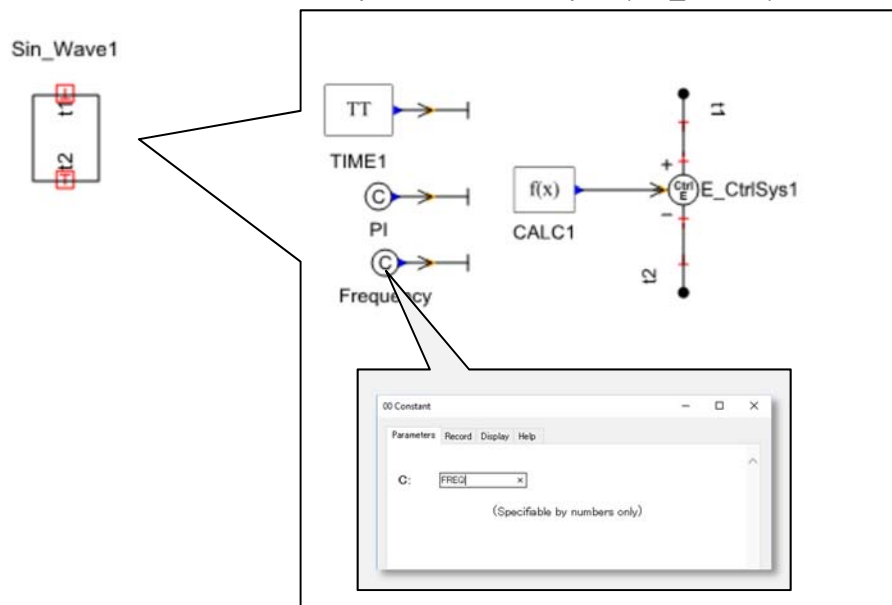
If the part was created in a previously specified subcategory, it is registered in that subcategory.



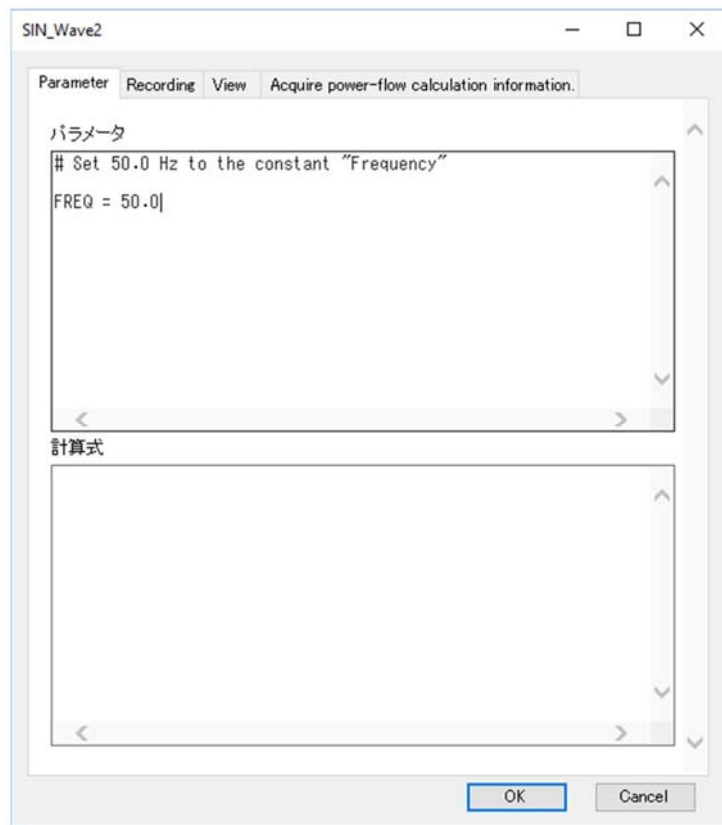
7 Method for setting parameters

In Step 1, the parameter for the 'Frequency' part was entered using the text string FREQ, making it a parameter for the macro part. We here describe how to specify this parameter when using the macro part.

Composition of macro part (Sin_Wave1)



Double-click on the macro part 'Sin_Wave1' to open its 'parameter setting' screen. Specify the value of the 'Frequency' parameter in Sin_Wave1 as 50.0, as shown below. As explained above, ***if the original circuit part enabled the selection of auxiliary units (prefixes for the unit of measurement such as M, k, m, etc.) prior to packaging, those auxiliary units are ignored when the parameter value is set in the macro part.***

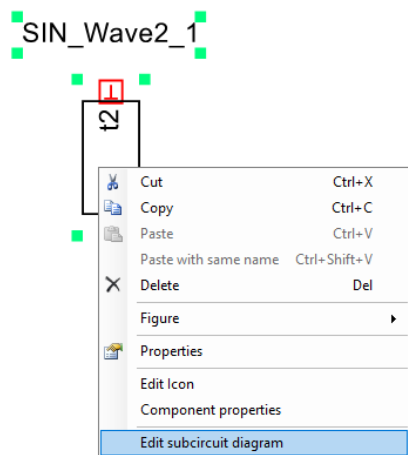


10.2 Changing macro parts

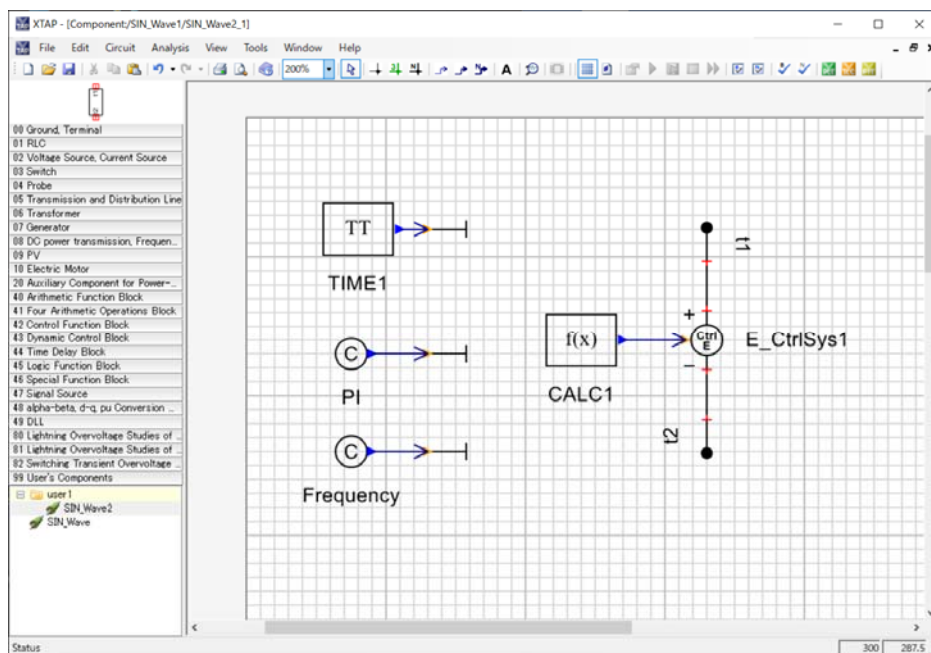
(1) Changing the circuits of macro parts placed in circuit diagrams

This section describes how to change the circuit within a macro part that is placed in a circuit diagram. Changes made to macro-part elements are applied to all the parts contained in the circuit diagram (or macroed circuit diagram) that were created from that part. Changes are applied even to a new part that is introduced into the circuit diagram via the 'part category' window, provided that this change was made in the circuit diagram. (In the circuit diagram, the same macro parts all have the same configuration, and the configuration in the circuit diagram has priority). However, part configurations that are modified in the 'part category' window do not affect the original part configuration; thus, a part placed on another circuit canvas will retain the original configuration.

1. Right-click the macro part you want to edit. The 'macro part' menu appears. Select 'Edit packaged diagram'.



2. The circuit diagram that the macro part was packaged from is displayed. The background color of the circuit-diagram canvas is gray, as when editing icons.

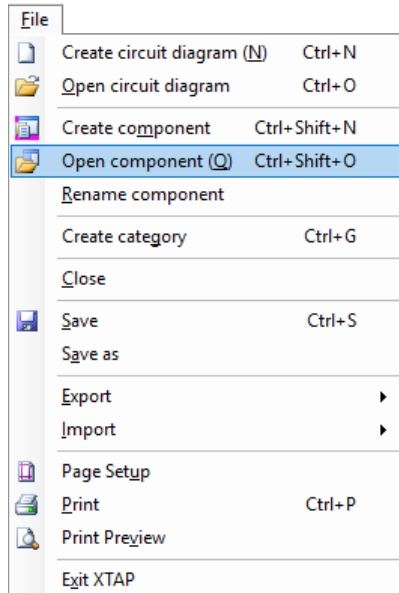


3. Edit and save the circuit diagram.

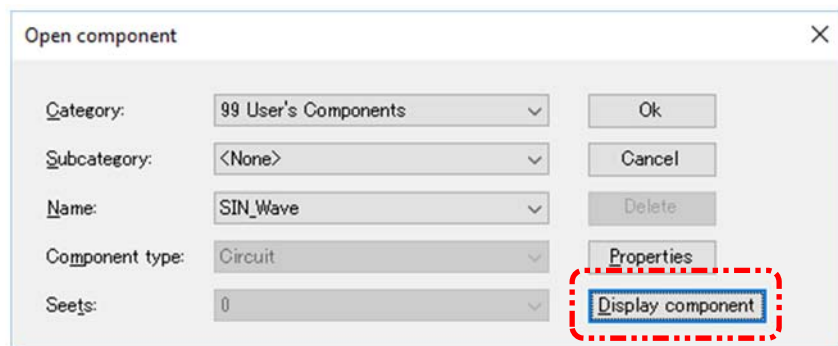
(2) Changing circuits of macro parts registered in the 'part category' window

The circuit of a macro part registered in the 'part category' window is edited as follows.

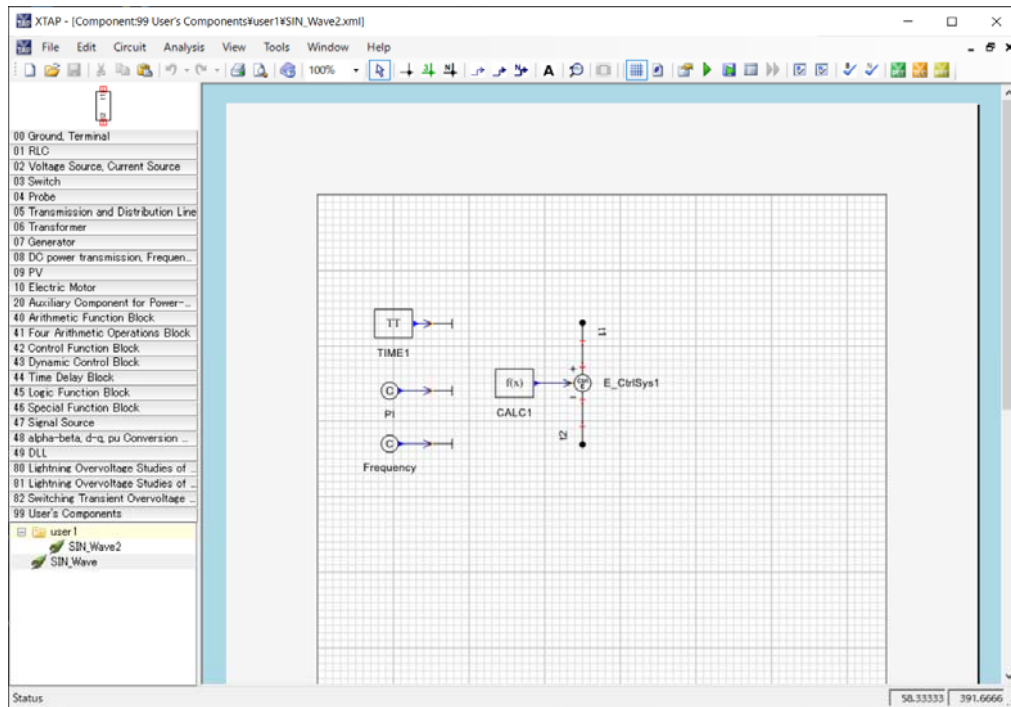
1. Select 'Open' – 'Open component' from the 'File' menu, or click the toolbar button used to open files and select 'Open component'.



2. Select the macro part you want to edit, then click 'Display part'.



3. The circuit diagram from which the macro part was packaged is displayed on a canvas with a blue background.

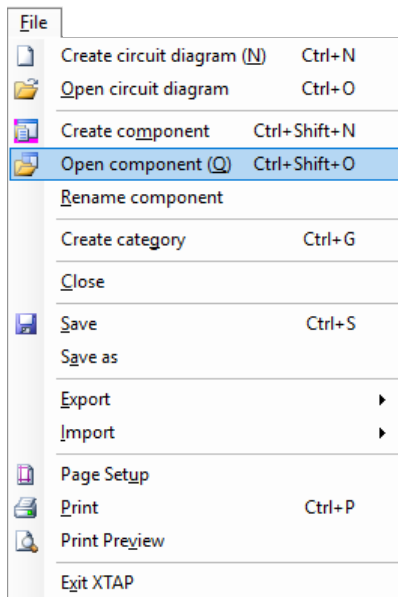


4. Edit and save the circuit diagram.

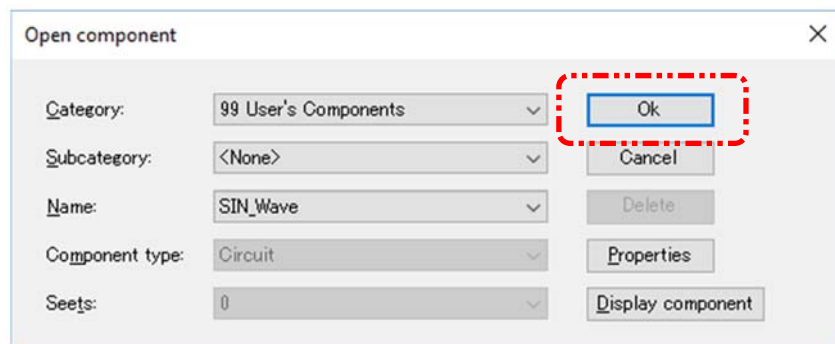
10.3 Editing macro-part icons

The icon of a created macro part is edited as follows.

1. Select 'Open' – 'Open component' from the 'File' menu, or click the toolbar button used to open files and select 'Open component'.



2. Select the macro part you want to edit, then click 'Display part'.



The button operations are described below.

Ok

- opens the editing screen for the icon of the macro part.

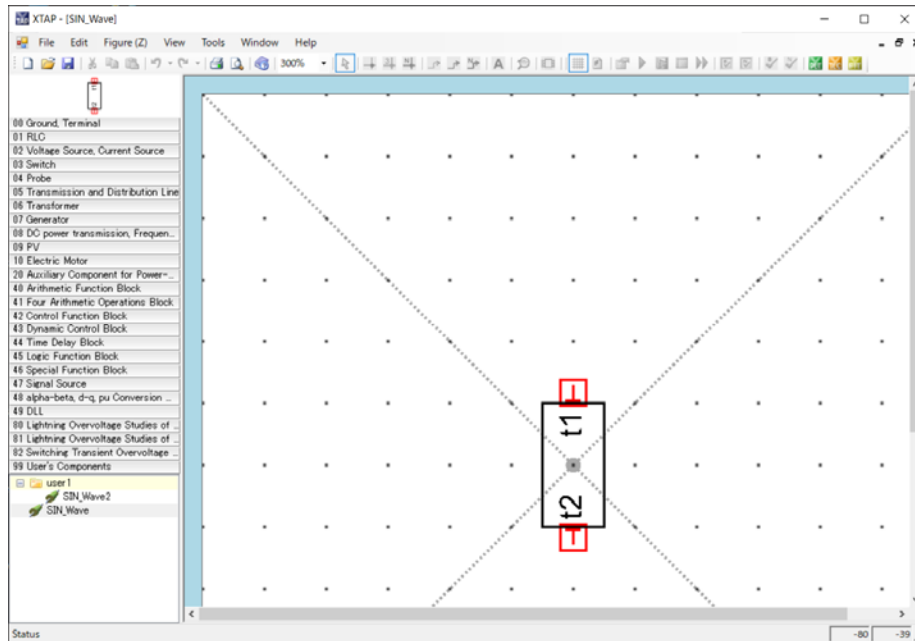
Property

- used to register parameters/formulas, or change part caption names.

Display part

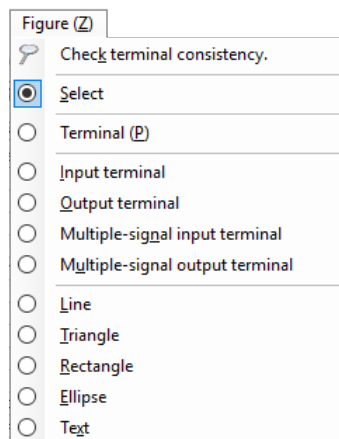
- displays the parts that compose the macro part (the original circuit diagram). This is used to change the circuit configuration or part parameters.

3. The icon-editing screen is displayed on a canvas with a blue background.



The following operations can be performed in the icon-editing screen:

Form (Z) menu (drawing objects)



Select

- selects a drawing object placed on the component canvas.

Straight line

- creates a straight line.

Triangle

- creates a triangle.

Rectangle

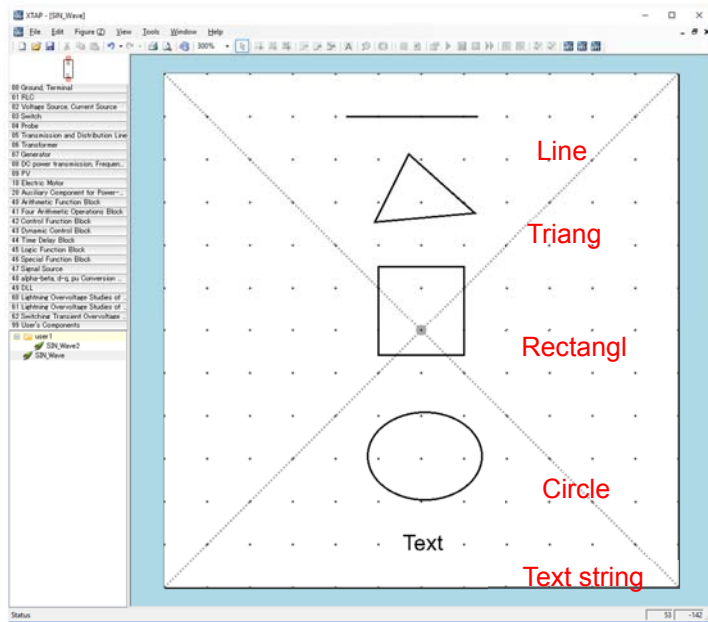
- creates a rectangle.

Circle

- creates a circle.

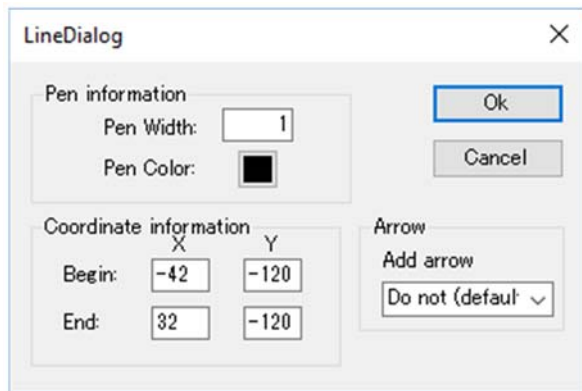
Text string

- creates a text string.



Line

- creates a line.
1. Select 'Line' from the 'Form' menu.
 2. Move the mouse cursor to the component canvas, and click the starting location.
 3. Drag the mouse to draw the line to the desired end location.
 4. Select 'Select' from the 'Form' menu.
 5. Double-click the created line to display its properties screen. Set the required property values to adjust the line features.



Pen information

Pen Width: sets the line thickness.

Pen Color: sets the line color.

Coordinate information

Start point: sets the coordinates of the starting location.

Endpoint: sets the coordinates of the end location.

Arrows

None (default): leaves the line without arrow ends.

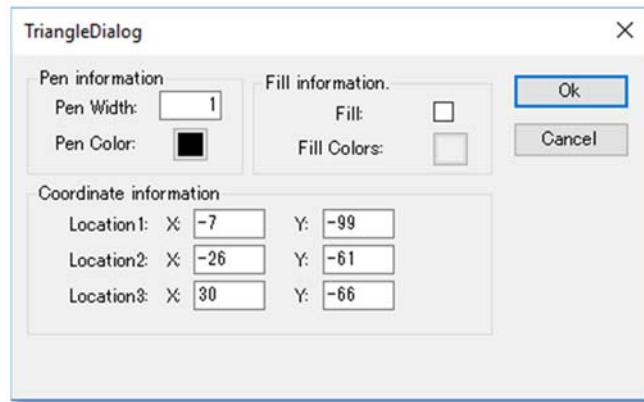
Start point: adds an arrow to the starting end of the line.

Endpoint: adds an arrow to the endpoint of the line.

Both: adds arrows to both ends of the line.

Triangle

- creates a triangle.
1. Select 'triangle' from the 'Form' menu.
 2. Move the mouse cursor to the component canvas, and click the location at which to create the triangle. A triangle is displayed.
 3. Select 'Select' from the 'Form' menu.
 4. Double-click the created triangle to display its properties screen. Set the required property values to adjust the triangle features.



The screenshot shows a 'TriangleDialog' window with a close button (X) in the top right corner. The window is divided into three main sections: 'Pen information', 'Fill information', and 'Coordinate information'. The 'Pen information' section includes a 'Pen Width' text box with the value '1' and a 'Pen Color' color picker showing black. The 'Fill information' section includes a 'Fill' checkbox (unchecked) and a 'Fill Colors' color picker showing white. The 'Coordinate information' section contains three rows of coordinate pairs: 'Location1' with X: -7 and Y: -99, 'Location2' with X: -26 and Y: -61, and 'Location3' with X: 30 and Y: -66. At the bottom right of the dialog are 'Ok' and 'Cancel' buttons.

Pen information

Pen Width: sets the line thickness.

Pen Color: sets the line color.

Fill information

Fill: check this box to fill the triangle with the selected color.

Fill color: sets the fill color.

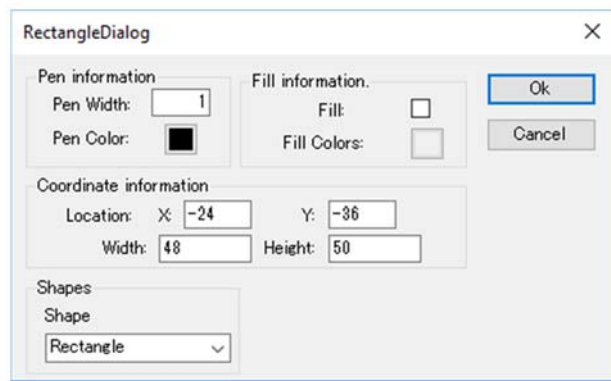
Coordinate information

- sets the triangle corner coordinates.

Rectangle

- creates a rectangle.

1. Select 'Rectangle' from the 'Form' menu.
2. Move the mouse cursor to the component canvas, and left-click at the desired location of the top-left corner of the rectangle.
3. Drag the mouse to define the size of the rectangle.
4. The rectangle is created when you stop dragging the mouse.
5. Select 'Select' from the 'Form' menu.
6. Double-click the created rectangle to display its properties screen. Set the required property values to adjust the rectangle features.



The screenshot shows a 'RectangleDialog' window with the following sections:

- Pen information:** Pen Width: 1, Pen Color: black.
- Fill information:** Fill: ☐, Fill Colors: (empty color box).
- Coordinate information:** Location: X: -24, Y: -36; Width: 48, Height: 50.
- Shapes:** Shape: Rectangle (selected from a dropdown menu).

Buttons for 'Ok' and 'Cancel' are located on the right side of the dialog.

Pen information

Pen Width: sets the line thickness.

Pen Color: sets the line color.

Fill information

Fill: check this box to fill the rectangle with the selected color.

Fill color: sets the fill color.

Coordinate information

Top-left coordinates: sets the coordinates of the top-left corner.

Width: sets the width.

Height: sets the height.

Rectangle shape

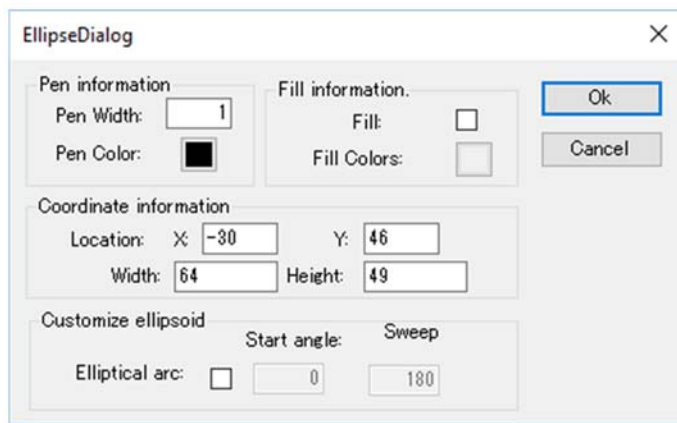
Rectangle: defines a normal rectangle (square corners).

Rounded rectangle: gives the rectangle rounded corners.

Circle

- creates a circle.

1. Select 'Circle' from the 'Form' menu.
2. Move the mouse cursor to the component canvas, and left-click at the desired location of the top-left of the circle frame.
3. Drag the mouse to define the size of the circle.
4. A circle is created when you stop dragging the mouse.
5. Select 'Select' from the 'Form' menu.
6. Double-click the created circle to display its properties screen. Set the required property values to adjust the circle features.



Pen information

Pen Width: sets the line thickness.

Pen Color: sets the line color.

Fill information

Fill: check this box to fill the circle with the selected color.

Fill color: sets the fill color.

Coordinate information

Top-left coordinates: sets the coordinates of the top-left corner of the circle frame.

Width: sets the width of the circle frame.

Height: sets the height of the circle frame.

Customize circle

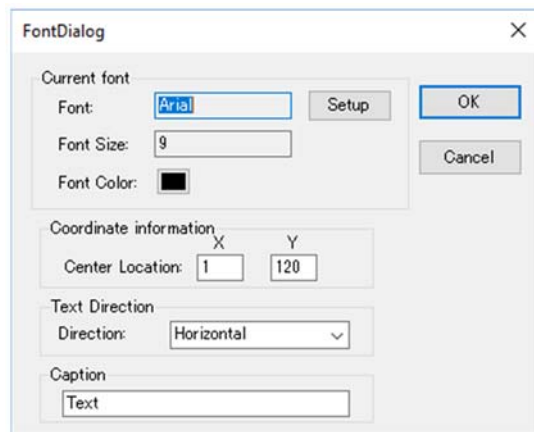
Make into arc: turns the circle into an arc when this box is checked.

Start angle: sets the starting angle of the arc.

Sweep angle: sets the sweep angle of the arc.

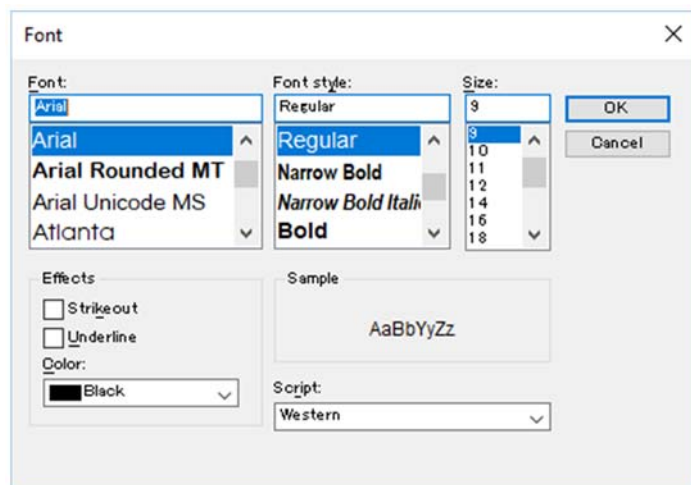
Text string

- creates a text string.
1. Select 'Text string' from the 'Form' menu.
 2. Move the mouse cursor to the component canvas, and click at the desired location of the text string.
 3. The cursor appears for entering text.
 4. Enter the desired text string.
 5. Select 'Select' from the 'Form' menu.
 6. Select the created text string and right-click to display a menu. Select 'Properties' from this menu to display the properties screen for the text string. Set the required property values to adjust the text-string features.



Current font

Click 'Set' to set the font.



Coordinate information

- specifies the coordinates of the center of the text string.

Display orientation

Normal: displays the text string normally.

Right 90°: rotates the text string clockwise so that its left side is at the top.

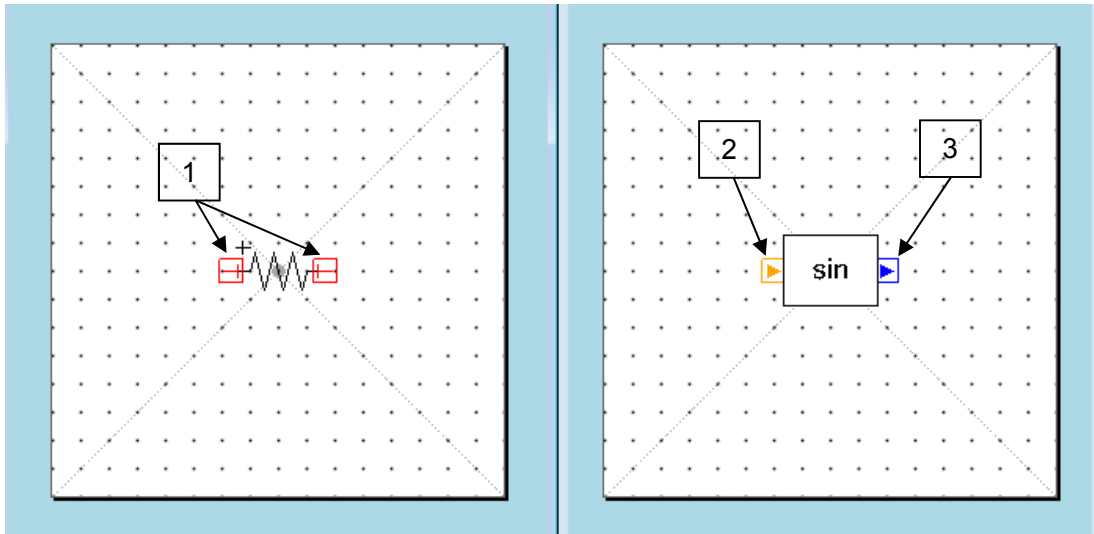
Left 90°: rotates the text string counterclockwise so that its left side is at the bottom.

Caption

- specifies the content of the text string.

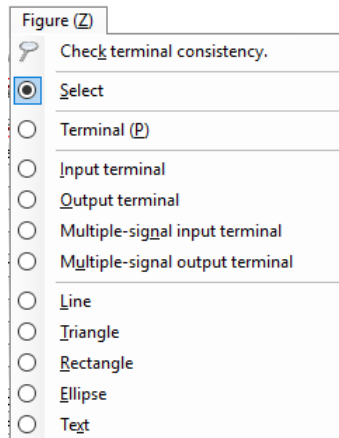
10.4 Connection-terminal settings

This section describes the screen for defining the settings of the part terminals. Terminals can be placed on the component canvas in the same way as when editing icons. Two parts are shown as examples in the screens below..



1. Terminals (default color: red) of an electrical-circuit part are used to connect one electrical-circuit part to another.
2. Input terminals (default color: orange) of a control-system block part connect an input signal to the part.
3. Output terminals (default color: blue) of a control-system block part connect to an output signal produced within the part.

Form menu (terminals)



Terminal

- creates a terminal for an electrical-circuit part.

Input terminal

- creates an input terminal for a control-system block part.

Output terminal

- creates an output terminal for a control-system block part.

Multiple-signal input terminal

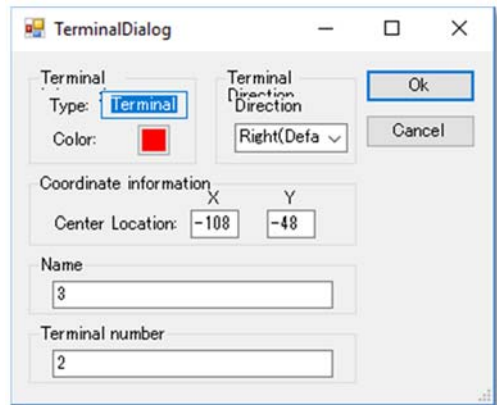
- creates a control system block part input terminal for handling vector signals

Multiple-signal output terminal

- creates a control system block part output terminal for handling vector signals

Terminal properties screen

Double-click on a terminal or right-click on it and select 'Properties' from the menu that appears, to display the terminal's properties screen.



Terminal information

Type: displays the terminal type (terminal, input terminal, output terminal).

Terminal color: used to select the color of the terminal.

Display orientation

- selects the orientation of the terminal (horizontal or vertical).

Coordinate information

- specifies the coordinates of the terminal's center. This must be entered in multiples of 4 (i.e., 0, 4, 8, etc.) since terminals are always placed on grid lines.

Name

- specifies the name used to manage the connection line (e.g., electric-circuit wire or control-system wire) connected to the terminal.

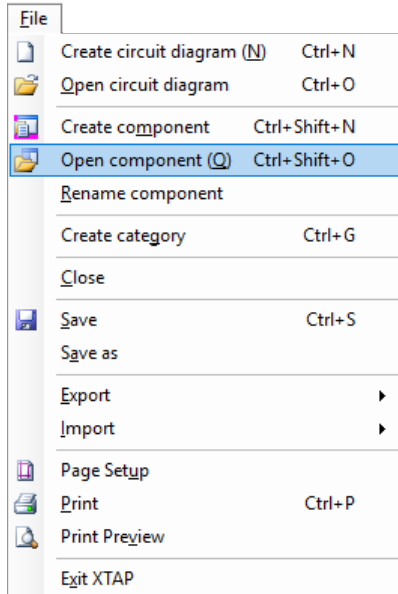
Terminal number

- specifies the output sequence of the terminal information output to *.xtf files (XTAP Text Files). Terminals numbers must be unique in parts.

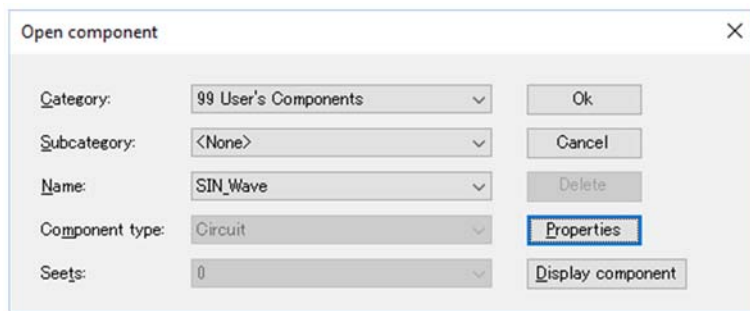
10.5 Registering parameters/formulas

Parameters and formulas can be registered in created macro parts. Once registered, a parameter or formula is entered when the macro part is set in the circuit diagram.

1. Select 'Open' – 'Open component' from the 'File' menu, or click the toolbar button used to open files and select 'Open component'.



2. Select the macro part you want to edit, then click 'Properties'.



3. The part's properties editing screen appears.

AddPageDlg
✕

Type name

Page Setup

	Tab Title	File Name	
▶	Parameter	XmfParams.html	...
	Recording	RecordOptionsSub.html	...
	View	DisplayOptionsCaption.html	...
	Acquire power-flow calculatio...	PowerFlowInfo.html	...
	Encryption	Sub_EncPassword	...

Data settings

Search tag

Tag ID
▶ SubParams
SubExpression
Caption
captiondisp
XFIXID_POWERFLOWINFO
SubOutput
*

Initial value
Value

Label
Direction
Display reference labe
Editing possible
New

Script
Input script
Output script

Text output
Output
Output sequence

Units
Unit conversion label

Label

CDP_NONE
False
False
True
2

Ok

Cancel

- To register a parameter, select 'SubParams' under 'Tag ID' and enter the parameter value next to 'Value' under 'Initial value'. This is demonstrated below for the parameter 'b = 100'.

AddPageDlg

Type name:

Page Setup

Tab Title	File Name
Parameter	XmfParams.html
Recording	RecordOptionsSub.html
Display	DisplayOptionsCaption.html
Acquire power-flow calculatio...	PowerFlowInfo.html
Encryption	Sub_EncPassword

Data settings

Search tag:

Tag ID
SubParams
SubExpression
Caption
captiondisp
XFIXID_POWERFLOWINFO
SubOutput
*

Initial value

Value: **b=100**

Label

Direction: **CDP_NONE**

Display reference label: **False**

Editing possible: **False**

New: **False**

Script

Input script: **False**

Output script: **False**

Text output

Output: **True**

Output sequence: **2**

Unit

Unit conversion label: **False**

Value

Specify the initial value of the data.

Ok Cancel

- To register a formula, select 'SubExpression' under 'Tag ID' and enter the formula next to 'Value' under 'Initial value'. This is shown below for the formula 'a = 5 * b'.

AddPageDlg

Type name:

Page Setup

Tab Title	File Name
Parameter	XmfParams.html
Recording	RecordOptionsSub.html
Display	DisplayOptionsCaption.html
Acquire power-flow calculatio...	PowerFlowInfo.html
Encryption	Sub_EncPassword

Data settings

Search tag:

Tag ID
SubParams
SubExpression
Caption
captiondisp
XFIXID_POWERFLOWINFO
SubOutput
*

Initial value

Value: **a = 5 * b**

Label

Direction: **CDP_NONE**

Display reference label: **False**

Editing possible: **False**

New: **False**

Script

Input script: **False**

Output script: **False**

Text output

Output: **True**

Output sequence: **3**

Unit

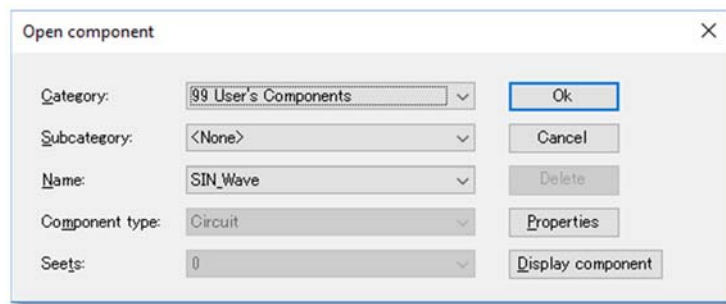
Unit conversion label: **False**

Value

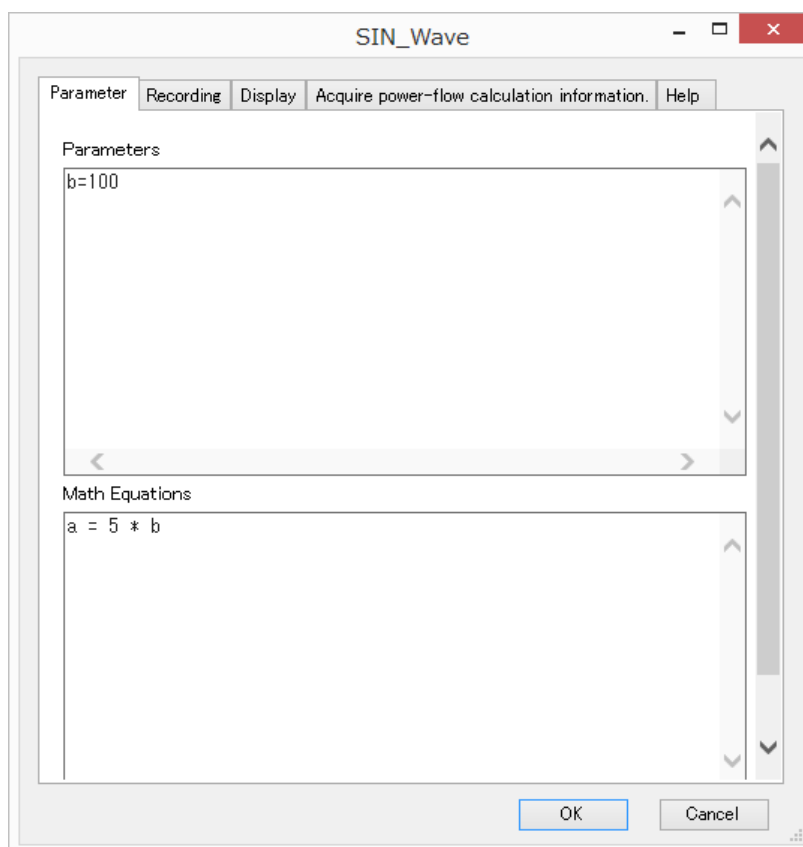
Specify the initial value of the data.

Ok Cancel

- Click 'Ok' to close the properties screen. Click 'Ok' in the 'Open component' screen to close the component.



- When the macro part is placed in the circuit diagram and its properties are viewed, the registered parameter and formula are entered.



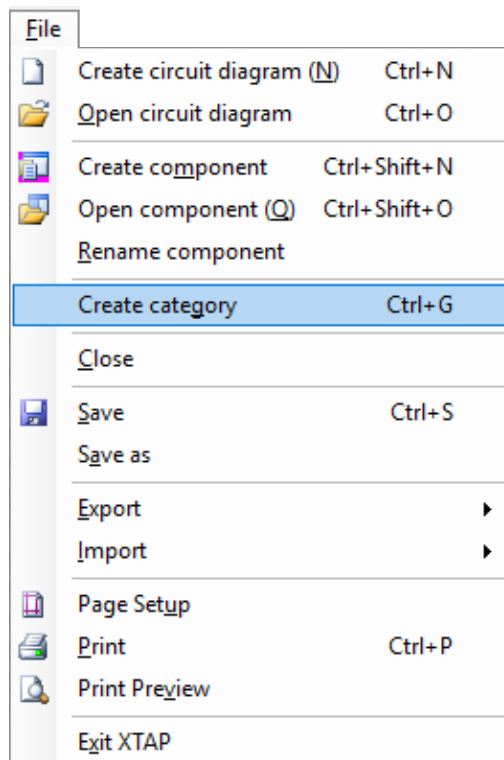
10.6 Part-category window composition; creating/deleting categories

Categories can be created within the '99 User's Components' category in the 'part category' window. Categories created within the '99 User's Components' category or parts registered there can also be deleted.

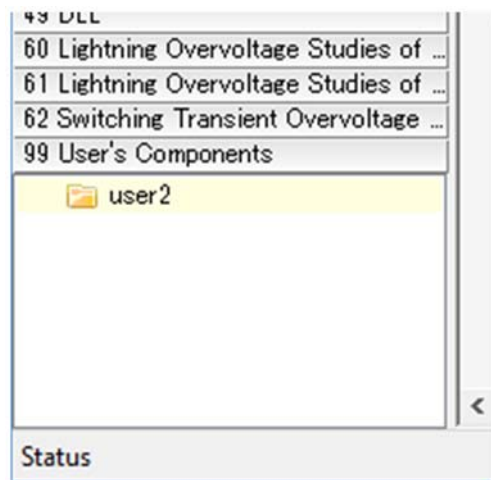
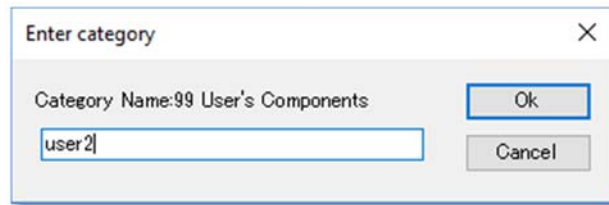
(1) Creating subcategories

This section describes how to create categories in the part-category window. All user-created categories are created as subcategories within the '99 User's Components' category.

1. Select 'Create category' from the 'File' menu.



2. The 'category input' screen appears. Enter the desired category name in the 'Category name' field. To create another category within this category, specify the higher-level category name, followed by the lower-level category name, with the names separated by '¥'. The screen shot below shows the category 'User1' being created.



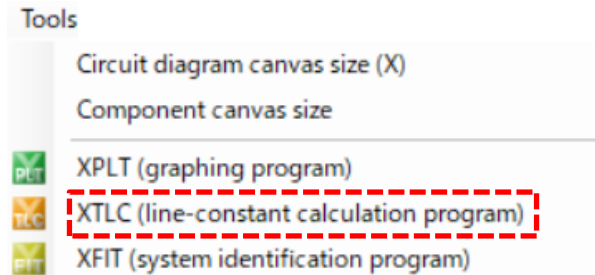
(2) Deleting categories/macro parts

To delete a created category, right-click on its name and select 'Delete' from the right-click menu. If any parts have been registered in a category, they must all first be deleted before the category itself can be deleted. Parts are deleted by the same method as the one used for categories.

11 Other

11.1 Starting XTLC (line constants calculation program)

To start XTLC from XTAP, select 'XTLC (line constants calculation program)' from the 'Tools' menu in the main window.



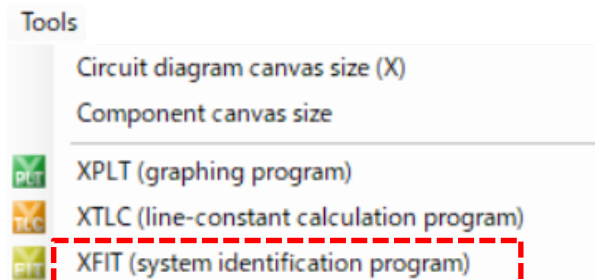
The same operation can also be performed by clicking the toolbar button indicated below.



Appendix 1 describes how to use XTLC.

11.2 Starting XFIT (system identification program)

To start XFIT from XTAP, select 'XFIT (system identification program)' from the 'Tools' menu in the main window.



The same operation can also be performed by clicking the toolbar button indicated below.



Appendix 2 describes how to use XFIT.

XTLC (Line Constants Calculation Program)

Version 2.4

Instruction Manual

Central Research Institute of Electric Power Industry (CRIEPI)

December 2022

Contents

1	Introduction.....	145
1.1Overview	145
1.2Main window.....	146
1.3Menu bar and toolbar	146
2	Creating New Line-Placement Data	149
2.1Selecting the line type	149
2.2Entering line-placement data	149
2.3Line-constant calculation	155
2.4Displaying line-constant calculation results	156
3	Referencing Existing Line-Placement Data	162
3.1Referencing line placement data	162
3.2Revising line-type information	162
3.3Calculating line constants and displaying results	162
4	Using Line Model Data Created in XTLC.....	163
4.1How to specify the data for the distributed-parameter line model.....	163
4.2How to specify the data for matching resistance	167
5	Registering Information in Line/Cable Databases.....	171
5.1Databases	171
5.2Overhead transmission lines	172
5.3Overhead distribution lines	173
5.4Cables	173
5.5Line/cable database functions	174

1 Introduction

1.1 Overview

The line-constants calculation program (XTLC) calculates the line constants referenced by the XTAP distributed-constant line model, by placing line conductors and using physical constants. It can also be used independently and specifically for line-constant calculations. Figure 1 outlines the basic operation. Further details of how to use each function are provided in the sections indicated.

□ Line-constants calculation [XTLC]

Creating new data: see Section 2.

Using existing data: see Section 3.

Line-type selection

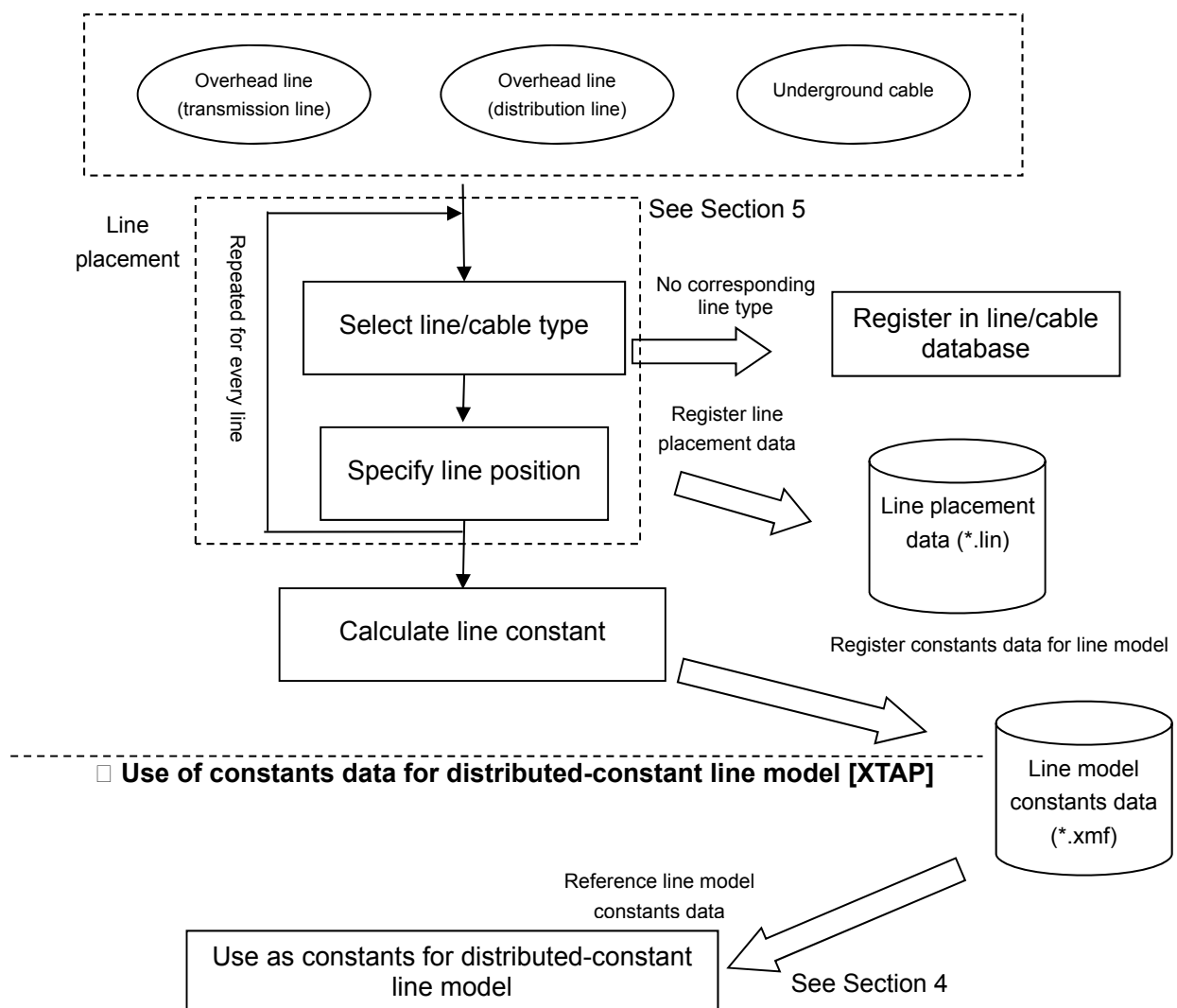
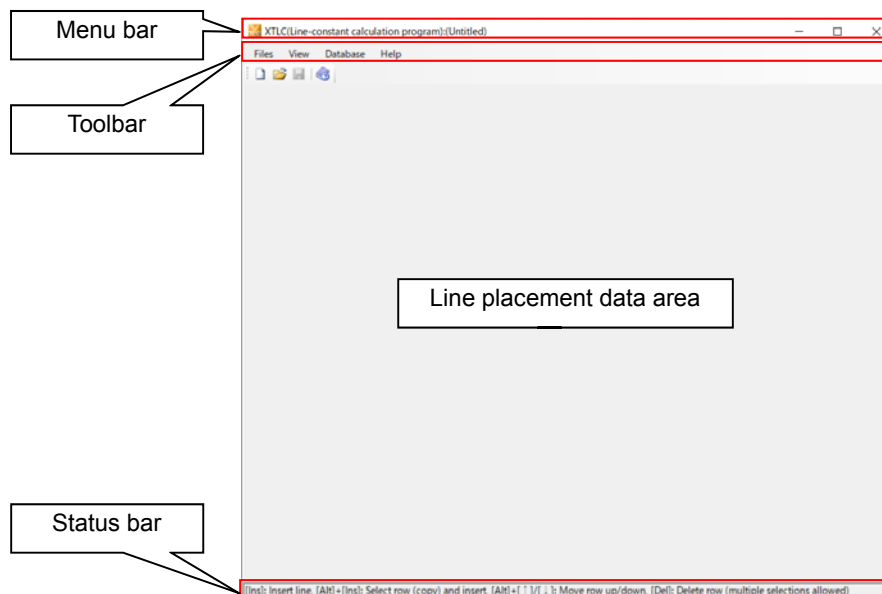


Figure 1. Basic XTLC operation workflow

- **Line-placement data** are contained in files used for calculating constants. They include physical information for each line, such as overhead-line pole information and buried cable locations.
- **Line model constant data** contain saved line-constant information calculated by XTLC. They can be referenced by XTAP distributed-constant line parts.

1.2 Main window

The main window, that appears when XTLC is started, is the following.

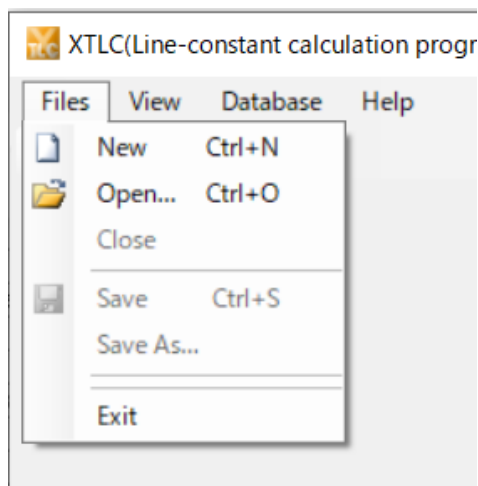


It consists of a menu bar, toolbar, and status bar, and an area for displaying line-placement data.

1.3 Menu bar and toolbar

The XTLC menu bar is described below.

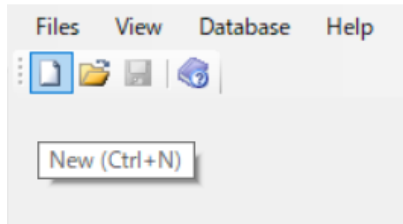
File (F) menu



New

- creates a new line-placement data file. The same operation can also be performed by clicking the toolbar button shown below.

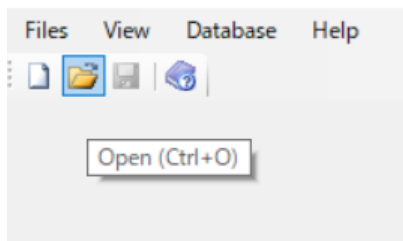
XTLC(Line-constant calculation pro



Open

- loads an existing line-placement data file and opens it in the line-placement data input screen. The same operation can also be performed by clicking the toolbar button shown below.

XTLC(Line-constant calculation prog



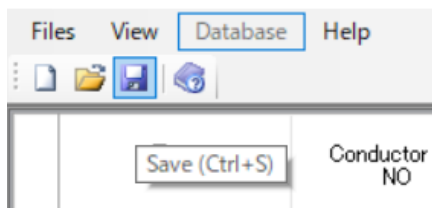
Close

- closes the line-placement data input screen.

Save

- saves the line-placement data (replacing the old data with a newer version). The same operation can also be performed by clicking the toolbar button shown below.

XTLC(Line-constant calculation progra



Save as

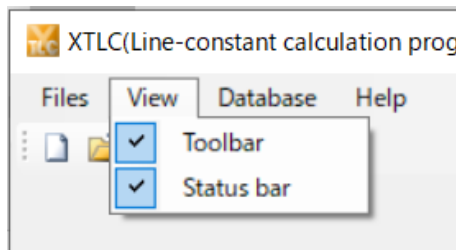
- saves the line-placement data under a new user-defined name if required.

Exit

- terminates XTLC.

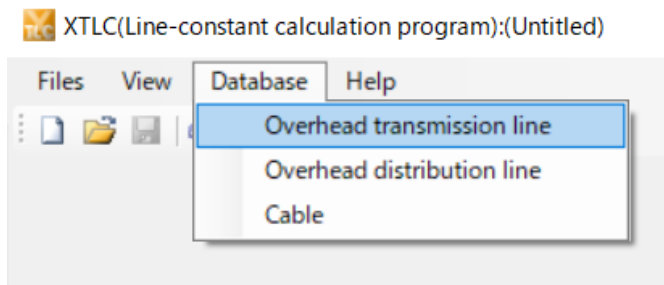
View (V) menu

- enables or disables the display of the toolbar or status bar.



Databases menu

- opens the line/cable databases, letting the user search and edit line types already registered in the databases, and register new lines. See Section 5 for details of how to register a new line type.



Overhead transmission line

- opens the overhead-transmission-line database.

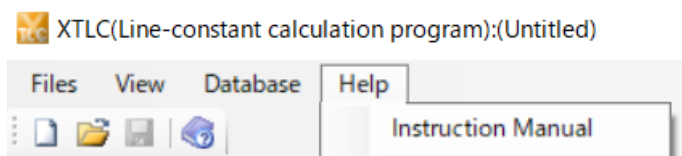
Overhead distribution line

- opens the overhead-distribution-line database.

Cable

- opens the cable database.

Help (H) menu




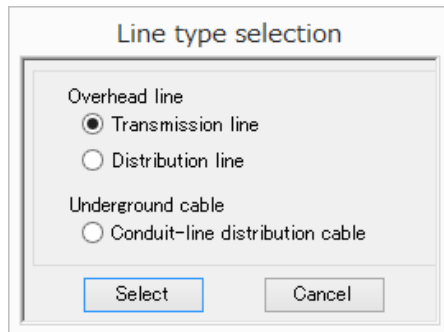
Instruction manual

- gives access to this manual.

2 Creating New Line-Placement Data

2.1 Selecting the line type

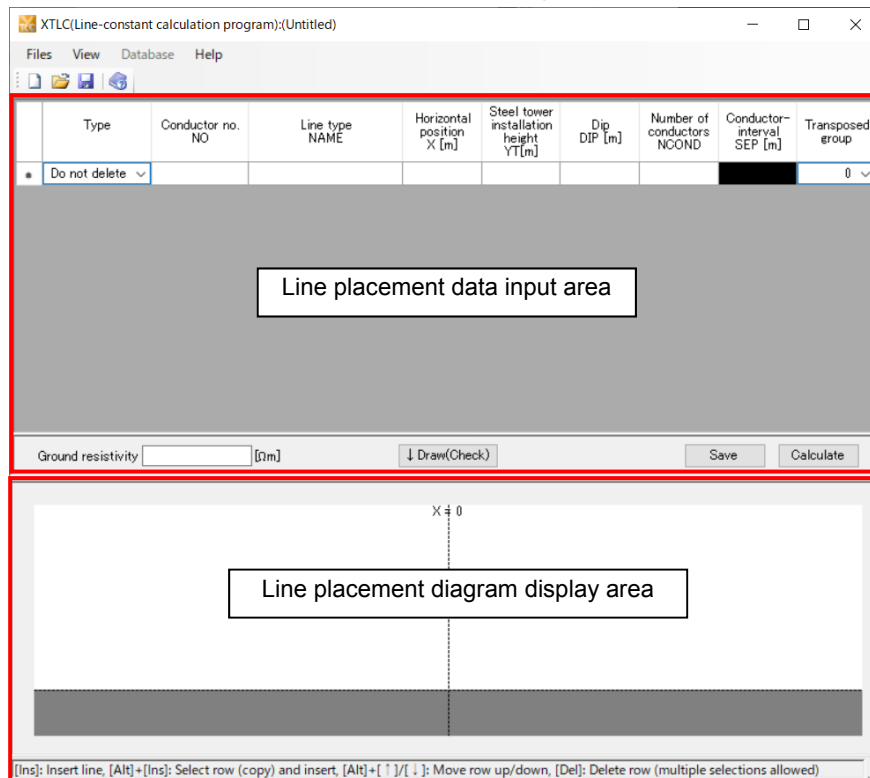
Select 'New' from the 'File' menu in the menu bar, or click the  toolbar button. The 'line-type selection' dialog appears.



Select the type of line-placement data to create, then click the 'Select' button.

2.2 Entering line-placement data

The line-placement data input screen is divided into areas for inputting line-placement data and displaying them diagrammatically. The items specified in the input area differ for overhead lines and underground cables.



The input area is used to enter line-placement data and the ground resistivity. The data are entered in the following steps:

- ① Select the type.

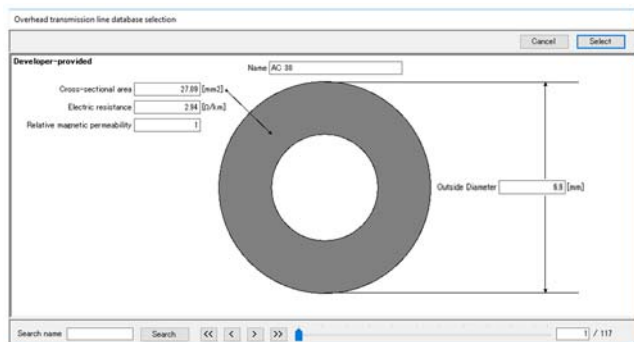
Select either 'Erase' or 'Do not erase'. (Select 'Erase' if it is assumed that the full line is at zero potential. This item is not available in the case of an underground cable.)

② Conductor No. (input not required)

The conductor number is automatically assigned when the line type is specified (Item 3 below). (0 when selecting 'Erase' in the type selection.)

③ Specify the line type.

Double-click the 'Line type' item to specify a line type. The Line/cable database screen appears. Select the desired line or cable.



If the desired line or cable does not exist, save the data and close the line-placement screen. See Section 5 for details of how to register the desired line or cable in a line/cable database. After registering this line or cable, load the saved data and continue the procedure. (See Section 3 for details of how to load existing data.)

④ Enter the line placement.

• Overhead line

Specify the conductor placement by entering the pole information in the line-placement data. A distribution line does not involve poles or conductor-interval settings.

Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
Do not delete								0

Do not delete	1	KTACSR 410						0
Do not delete								0

To check conductor placement, click the '↓ Draw (check)' button to draw the conductor placement in the line-placement diagram, as shown below.

XTLC(Line-constant calculation program):C:\XTAP_CRIEPI\work\Examples\STL-02\vertically.lin

Files View Database Help

Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor-interval SEP [m]	Transposed group
Delete ▾	0	IACSR 150	-14.5	79.5	0	1		0 ▾
Delete ▾	0	IACSR 150	14.5	79.5	0	1		0 ▾
Do not delete ▾	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▾
Do not delete ▾	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▾
Do not delete ▾	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▾
Do not delete ▾	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▾
Do not delete ▾	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▾
Do not delete ▾	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▾

Ground resistivity 300 [Ωm] ↓ Draw(Check) Save Calculate

[Ins]: Insert line, [Alt]+[Ins]: Select row (copy) and insert, [Alt]+[↑]/[↓]: Move row up/down, [Del]: Delete row (multiple selections allowed)

• Underground cable

Specify the conductor placement by entering the buried cable positions and conduit-line information in the line-placement data.

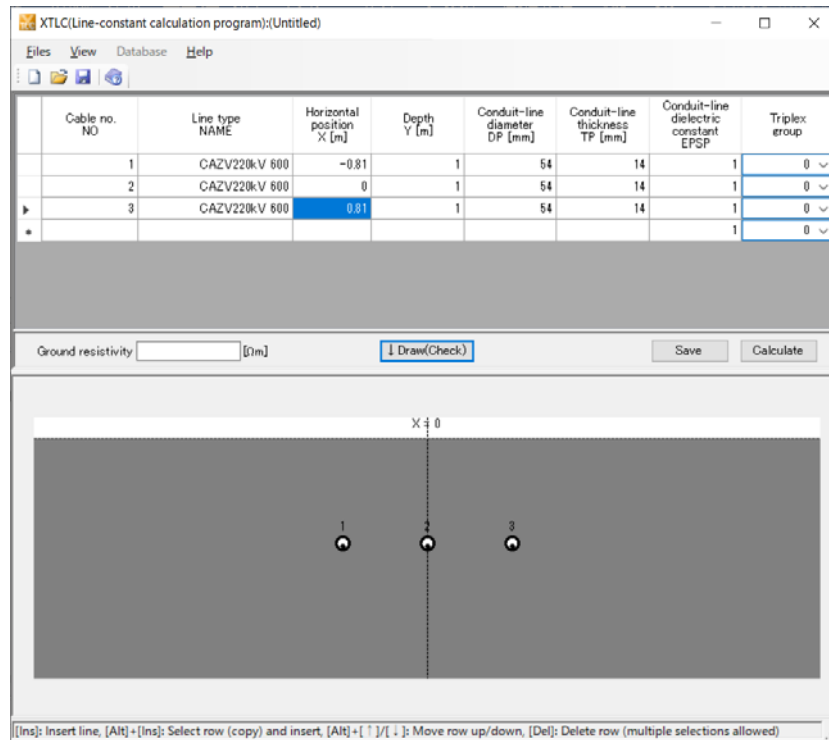
Cable no. NO	Line type NAME	Horizontal position X [m]	Depth Y [m]	Conduit-line diameter DP [mm]	Conduit-line thickness TP [mm]	Conduit-line dielectric constant EPSP	Triplex group
1						1	0 ▾

Enter

Add automatically

Cable no. NO	Line type NAME	Horizontal position X [m]	Depth Y [m]	Conduit-line diameter DP [mm]	Conduit-line thickness TP [mm]	Conduit-line dielectric constant EPSP	Triplex group
1	CAZV220kV 600					1	0 ▾
2						1	0 ▾

To check conductor placement, click the '↓ Draw (check)' button to draw the conductor placement in the line-placement diagram, as shown below.



⑤ Specifying a Twist group

For a 'Twist group' (overhead line) or 'Triplex group' (underground cable), specify three lines or cables as a group by selecting the same group number for them.

Example: Setting conductors 1 to 3 as Group 1, and 4 to 6 as Group 2.

Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor-interval SEP [m]	Transposed group
Delete	0	IACSR 150	-14.5	79.5	0	1		0
Delete	0	IACSR 150	14.5	79.5	0	1		0
Do not delete	1	TACSR 410	-11.2	72.0	0	6	0.5	1
Do not delete	2	TACSR 410	-11.6	57.5	0	6	0.5	1
Do not delete	3	TACSR 410	-12.0	43.0	0	6	0.5	1
Do not delete	4	TACSR 410	11.2	72.0	0	6	0.5	2
Do not delete	5	TACSR 410	11.6	57.5	0	6	0.5	2
Do not delete	6	TACSR 410	12.0	43.0	0	6	0.5	2


Group 1

Group 2

For a non-twist line, input a 0 under 'Twist group' or 'Triplex group'.

Since in the case of underground cables, the assumption is that each cable will be installed along the conduit lines, it is not possible to rigorously simulate situations where triplex cables are installed in a group along a single conduit line. When triplex cables are placed as underground cables, place three cables based on the assumption that all three phases of the triplex cable can be fit into the conduit line. Place in such a way that the three cables are attached in an equilateral triangle.

⑥ Save the line-placement data.

Select 'Save' from the 'File' menu on the menu bar, click the  toolbar button, or click the 'Save' button to save the data. To apply the updated information to the calculation, data must be saved before performing the line-constant calculation.

The following operations can be performed in the line-placement data input screen:

Selection

Click the first cell in a row to select that row.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
▶	Do not delete ▼	1	TACSR 410						0 ▼
*	Do not delete ▼								0 ▼

Click

The row is then selected.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
▶	Do not delete ▼	1	TACSR 410						0 ▼
*	Do not delete ▼								0 ▼

Selected

Multiple selection (continuous range)

Hold down the Shift key on the keyboard and click the first cells of the first and last rows in the desired range.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
▶	Do not delete ▼	0	IACSR 150	-14.5	79.5	0	1		0 ▼
	Do not delete ▼	0	IACSR 150	14.5	79.5	0	1		0 ▼
▶	Do not delete ▼	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▼
	Do not delete ▼	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▼
	Do not delete ▼	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▼
▶	Do not delete ▼	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▼

Last row

All the rows in that range will then be selected.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Delete ▼	0	IACSR 150	-14.5	79.5	0	1		0 ▼
	Delete ▼	0	IACSR 150	14.5	79.5	0	1		0 ▼
▶	Do not delete ▼	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▼
	Do not delete ▼	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▼
	Do not delete ▼	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▼
▶	Do not delete ▼	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▼

Selected

Multiple selection (separate rows)

Hold down the 'Ctrl' key on the keyboard and click the first cells of each required row.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Delete ▼	0	IACSR 150	-14.5	79.5	0	1		0 ▼
	Delete ▼	0	IACSR 150	14.5	79.5	0	1		0 ▼
▶	Do not delete ▼	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▼
▶	Do not delete ▼	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▼
	Do not delete ▼	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▼
	Do not delete ▼	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▼
▶	Do not delete ▼	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▼

Only these rows will then be selected.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Delete ▾	0	IACSR 150	-14.5	79.5	0	1		0 ▾
	Delete ▾	0	IACSR 150	14.5	79.5	0	1		0 ▾
	Do not delete ▾	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▾
	Do not delete ▾	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▾
	Do not delete ▾	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▾
	Do not delete ▾	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▾

Inserting

Select the location at which to insert a new row. Press the 'Insert' key on the keyboard to insert a new row immediately above this location.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Do not delete ▾	1							0 ▾
	Do not delete ▾	2	TACSR 410						0 ▾
	Do not delete ▾								0 ▾

New row is inserted

Inserting a selected (copied) row

Select the location at which to insert a new row. Press the 'Alt' and 'Insert' keys on the keyboard together to insert a new row immediately above this location. The new row has the same content as the row previously at this location. Conductor numbers are automatically changed.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Do not delete ▾	1	TACSR 410	-11.2	72.0	0	6		0 ▾
	Do not delete ▾	2	TACSR 410	-11.2	72.0	0	6		0 ▾
	Do not delete ▾								0 ▾

Copied row is inserted

Delete

Select one or more rows and press the 'Delete' key on the keyboard to delete them.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Delete ▾	0	IACSR 150	-14.5	79.5	0	1		0 ▾
	Delete ▾	0	IACSR 150	14.5	79.5	0	1		0 ▾
	Do not delete ▾	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▾
	Do not delete ▾	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▾
	Do not delete ▾	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▾
	Do not delete ▾	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▾

Conductor numbers are automatically changed.

	Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NCOND	Conductor- interval SEP [m]	Transposed group
	Delete ▾	0	IACSR 150	-14.5	79.5	0	1		0 ▾
	Delete ▾	0	IACSR 150	14.5	79.5	0	1		0 ▾
	Do not delete ▾	1	TACSR 410	-11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾	2	TACSR 410	11.2	72.0	0	6	0.5	0 ▾
	Do not delete ▾	3	TACSR 410	11.6	57.5	0	6	0.5	0 ▾
	Do not delete ▾								0 ▾

Moving

A single selected row can be moved up or down. Select the row and press 'Alt' and the 'up arrow' cursor key on the keyboard together to move the row upward. Similarly, use the 'down arrow' cursor key to move the row downward. Conductor numbers are automatically changed to handle the move.

2.3 Line-constant calculation

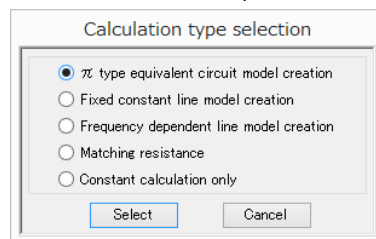
Click the 'Calculate' button to perform the line-constant calculation. After creating or updating line placement data, the data must be saved before proceeding to the line-constant calculation.



A screenshot of a software toolbar. It contains a text input field for 'Ground resistivity' with the value '300' and unit '[Ωm]'. To its right is a button labeled '↓ Draw(Check)'. Further right are two buttons: 'Save' and 'Calculate'. The 'Calculate' button is highlighted with a red rectangular border.

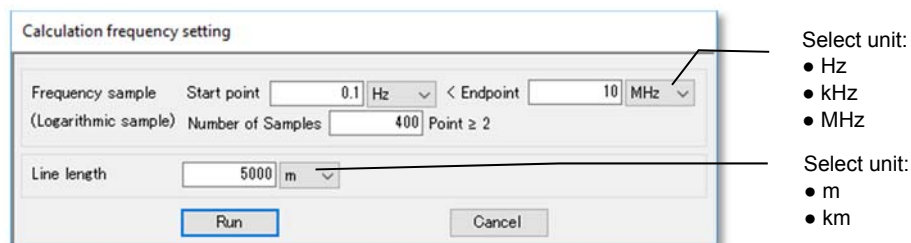
The 'calculation-type selection' dialog appears. Select the type of calculation to perform. If twist or triplex are specified, the 'Create fixed-constant line model' and 'Create frequency-dependent line model' items cannot be selected*.

*For transmission lines with a three-phase twist, it is possible to simulate a fixed-parameter line constant line model by using 'Three-phase twist line model (fixed-parameter)' in the XTAP distributed-constant line parts.



A dialog box titled 'Calculation type selection'. It contains five radio button options:
• ☒ π type equivalent circuit model creation
• ☐ Fixed constant line model creation
• ☐ Frequency dependent line model creation
• ☐ Matching resistance
• ☐ Constant calculation only
At the bottom are 'Select' and 'Cancel' buttons.

Different parameters are required for each of the selected calculation types. For example, when the frequency-dependent line model has been selected, the following screen appears. Enter the required parameters, then click 'Run' to start the line-constant calculation.

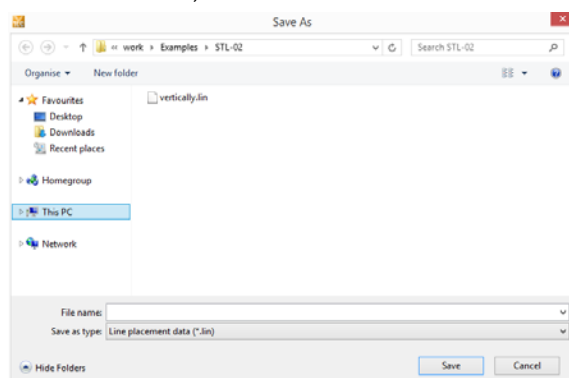


A dialog box titled 'Calculation frequency setting'. It contains the following fields and controls:
• 'Frequency sample' dropdown menu.
• 'Start point' input field with '0.1' and a unit dropdown showing 'Hz'.
• '< Endpoint' input field with '10' and a unit dropdown showing 'MHz'.
• '(Logarithmic sample) Number of Samples' input field with '400'.
• 'Point ≥ 2' label.
• 'Line length' input field with '5000' and a unit dropdown showing 'm'.
• 'Run' and 'Cancel' buttons at the bottom.
Two arrows point from text labels to the unit dropdowns:
1. An arrow points from 'Select unit: • Hz • kHz • MHz' to the 'Endpoint' unit dropdown.
2. An arrow points from 'Select unit: • m • km' to the 'Line length' unit dropdown.

If either of the following items is selected, a 'file selection' dialog appears when the calculation finishes. The file selected in the dialog is then used to save the line-model data for use by XTAP.

- **Create π -type equivalent circuit model**
- **Create fixed-constant line model**
- **Create frequency-dependent line model**
- **Matching resistance**

Enter a file name, then click 'Save'.



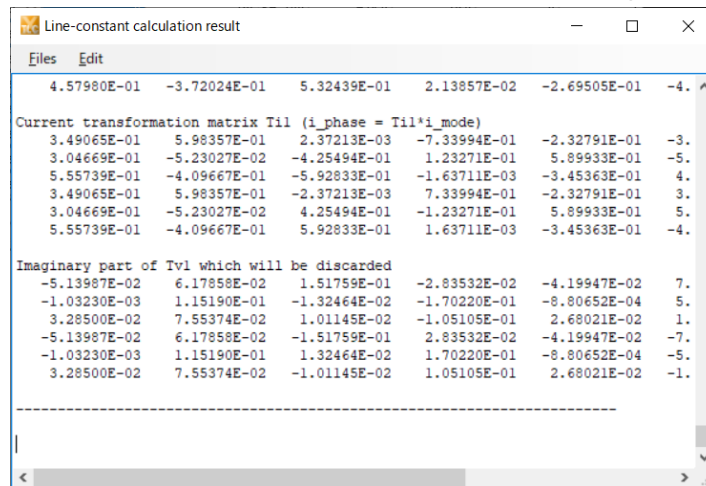
A 'Save As' file dialog box. The title bar says 'Save As'. The address bar shows the path 'work > Examples > STL-02'. The file list on the left shows 'vertically.in'. The 'File name' field is empty. The 'Save as type' dropdown is set to 'Line placement data (*.lin)'. 'Save' and 'Cancel' buttons are at the bottom.

2.4 Displaying line-constant calculation results

Line-constant calculation results are displayed in the 'Line-constant calculation results' or 'Plot dialog' screen. One of these screens appears when the line-constant calculation ends.

① Line-constant calculation results

This displays the line-constant calculation results. If 'Constant calculation only' or 'Create frequency-dependent line model' was selected in Section 2.3, the 'Plot dialog' described below in Item 2 is displayed first, and then the 'Line-constant calculation results' screen appears when this dialog is closed.



If 'Constant calculation only' or multiple 'Specified frequency' values were previously specified, only the result calculated first is displayed. To display all the frequency results, check the *.mat file located in the same folder as the result file. They can then be checked using MATLAB. To save the calculation results, select 'Save' from the 'File' menu. Alternatively, select 'Copy all' from the 'Edit' menu to save the results by pasting them into external software.

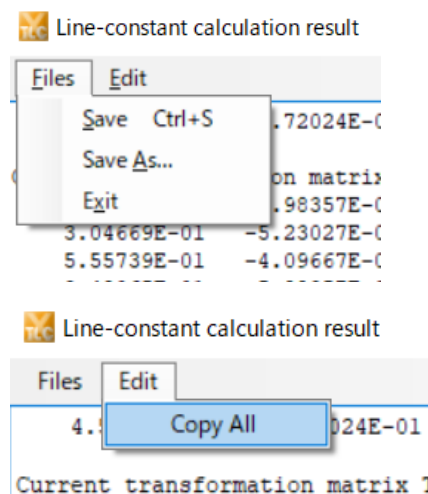


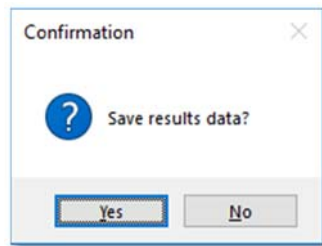
Table 1 shows the line constant list displayed in the 'Line constant results'. Refer to the following documentation for details regarding each constant.

- [1] Central Research Institute of Electric Power Industry (CRIEPI) Research Report H04014, "Development of a Line Constants Calculation Program for Overhead Transmission Lines - An eigenvector normalization algorithm and an approximate formula for ground-return impedance -" (can be downloaded from CRIEPI's Web site)

Table 1 Line constant list

Frequency = 5.00000E+001 [Hz]	
Phase domain constant (before GW elimination)	
PHASE DOMAIN QUANTITIES (before ground-wire elimination)	
Space inductance Ls in phase coordinates [H/m]	Space inductance
Ground-return impedance Ze (including Ls) in phase	Ground-return impedance
Conductor internal impedance Zc in phase coordinates [ohm/m]	Conductor internal impedance
Total series impedance Z in phase coordinates [ohm/m]	Line series impedance
Space capacitance Cs in phase coordinates [F/m]	Space capacitance
Shunt admittance in phase coordinates [S/m]	Line parallel admittance
Phase domain constant (after GW elimination)	
PHASE DOMAIN QUANTITIES (after ground-wire elimination)	
Total series impedance Z in phase coordinates [ohm/m]	Line series impedance
Shunt capacitance C in phase coordinates [F/m]	Line parallel capacitance
Shunt admittance Y in phase coordinates [S/m]	Line parallel admittance
Propagation constants Gamma in phase coordinates [1/m]	Propagation constant
Characteristic impedance Z0 in phase coordinates [ohm]	
Modal domain constant (before transformation matrix normalization)	
Series impedance z in modal coordinates [ohm/m]	
Shunt admittance y in modal coordinates [S/m]	
Propagation constants gamma in modal coordinates [1/m]	
Propagation velocities in modal coordinates [m/s]	
Characteristic impedance z0 in modal coordinates [ohm]	
Voltage transformation matrix Tv (v_phase = Tv*v_mode)	
Current transformation matrix Ti (i_phase = Ti*i_mode)	
Modal domain constant (after transformation matrix normalization)	
Series impedance z1 in modal coordinates [ohm/m]	
Shunt admittance y1 in modal coordinates [S/m]	
Characteristic impedance z01 in modal coordinates [ohm]	
Voltage transformation matrix Tv1 (v_phase = Tv1*v_mode)	
Current transformation matrix Ti1 (i_phase = Ti1*i_mode)	
Imaginary part of Tv1 which will be discarded	
Symmetrical components (only displayed in case of three-phase line)	
Positive-Sequence Impedance [ohm/km]	
Zero-Sequence Impedance [ohm/km]	

If the results screen is closed without saving, a 'save confirmation' dialog appears. Click 'Yes' to save the results.



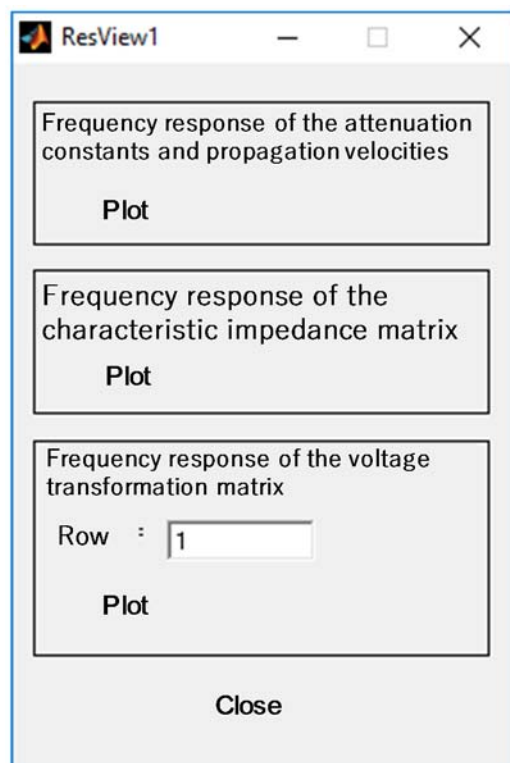
② Plot dialog

The 'Plot dialog' appears if 'Constant calculation only' or 'Create frequency-dependent line model' was selected in Section 2.3.

a. ResView1

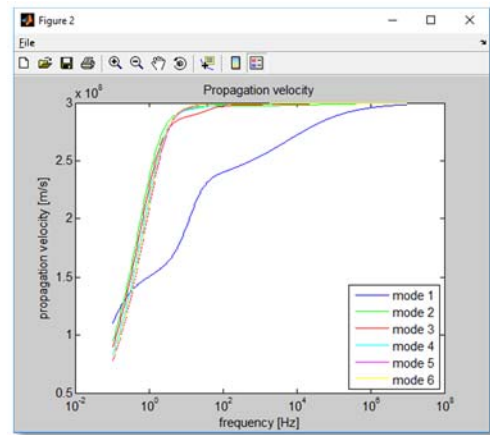
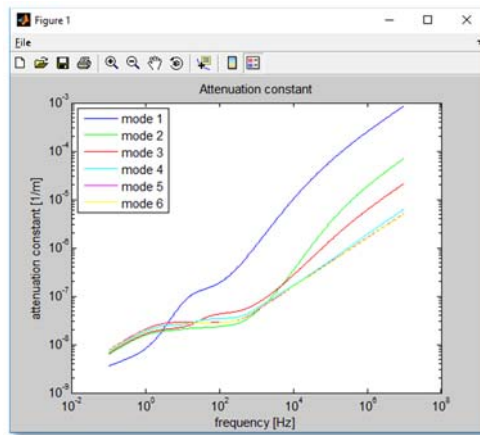
This is displayed when the item below is selected as the calculation type, but not when only one frequency has been specified.

• Constant calculation only

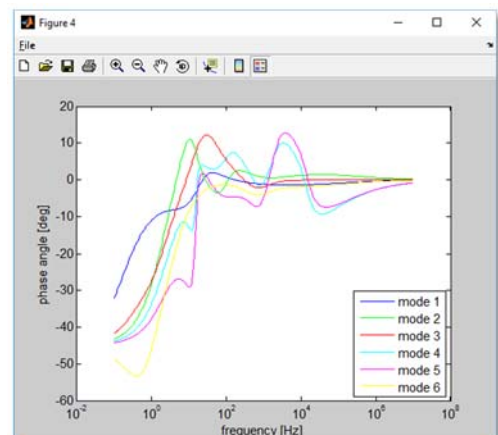
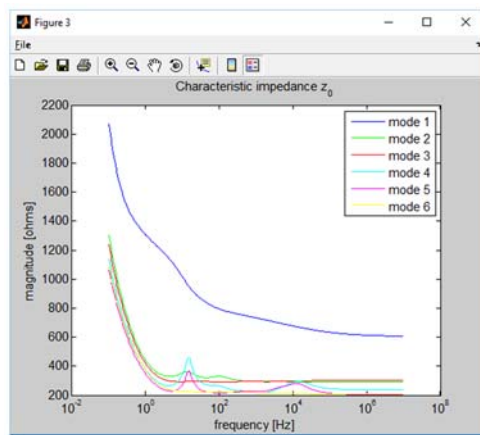


Each 'Plot' buttons displays the corresponding plot.

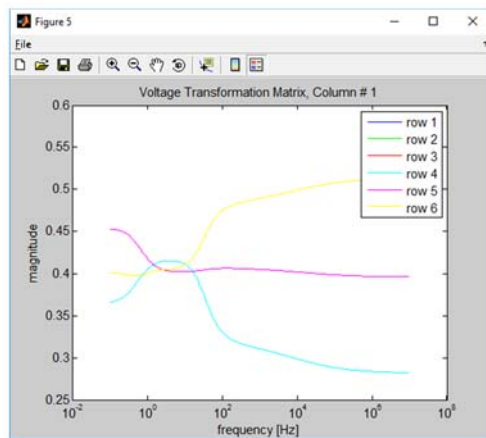
☐ Damping constant and propagation-velocity frequency response



☐ Characteristic-impedance frequency response



☐ Voltage transformation matrix frequency response

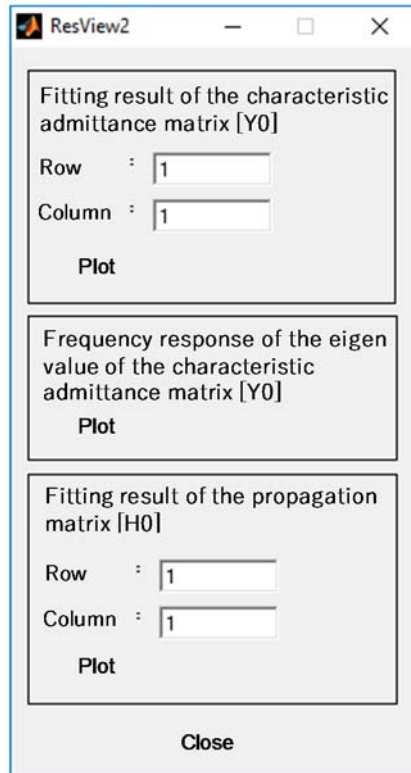


b. ResView2

This is displayed when the following item is selected as the calculation type.

• Create frequency-dependent line model

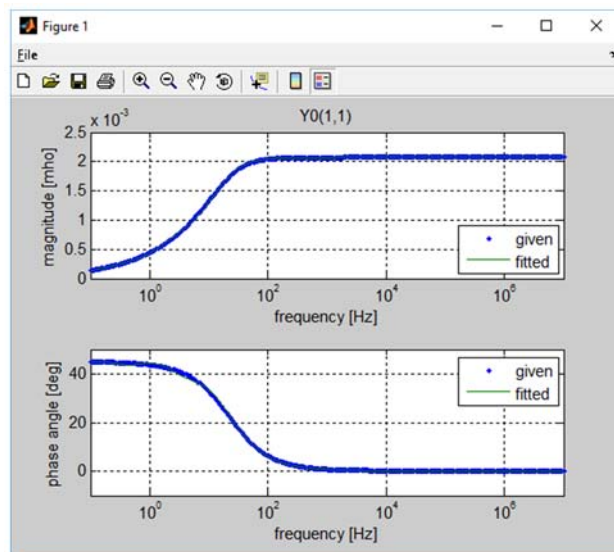
This dialog allows checking of the model precision by comparing the frequency characteristics of the calculated line constant and of a frequency-dependent model created by fitting it.



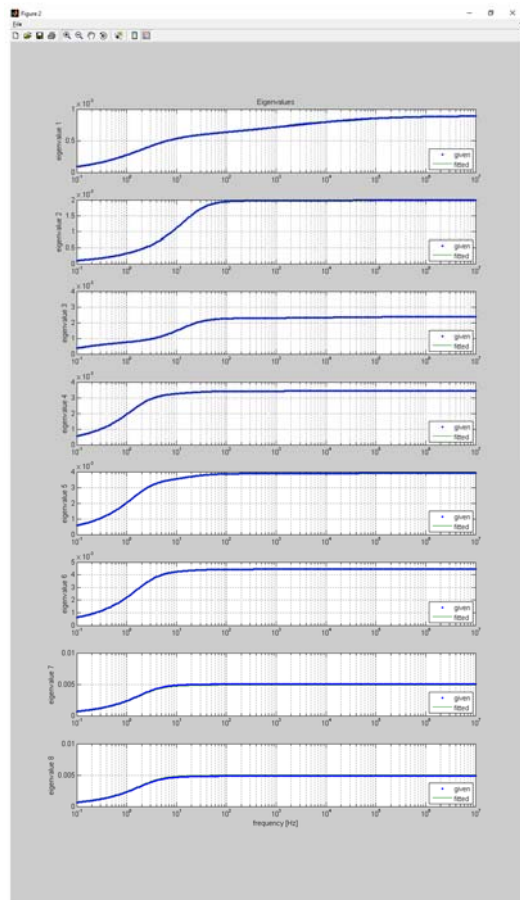
Each 'Plot' buttons displays the corresponding plot.

□ Characteristic admittance matrix [Y0] fitting results

The graphs for the specified row or column are displayed.

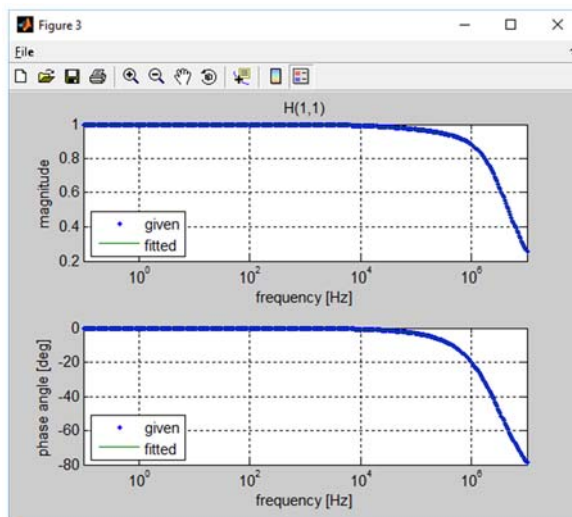


- Characteristic admittance matrix $[Y_0]$ eigenvalue frequency characteristics (stability evaluation)




- Propagation function matrix $[H]$ fitting results

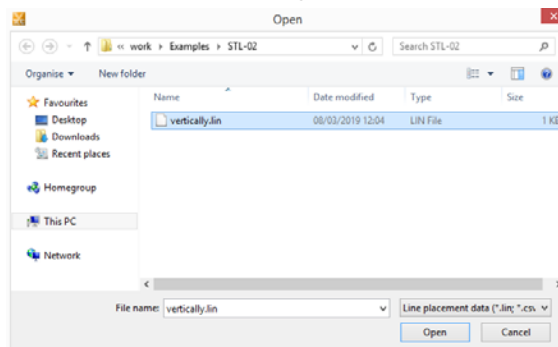
The graphs for the specified row or column are displayed.



3 Referencing Existing Line-Placement Data

3.1 Referencing line placement data

Select 'Open' from the 'File' menu on the menu bar, or click the  toolbar button. The 'file selection' dialog appears. Select a line-placement data file (*.lin file).

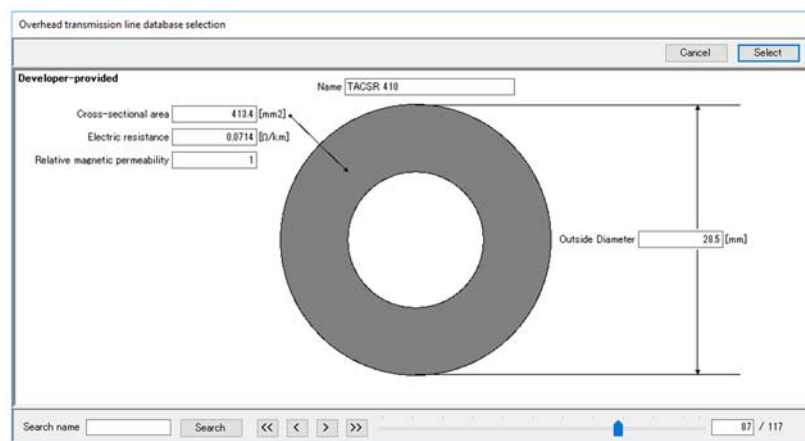


3.2 Revising line-type information

To revise a line type, double-click on the line type.

Type	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steering height Y [m]	SLIP [m]	Conductor NCONCD	Conductor-interval SEP [m]	Transposed group
Delete ▾	0	IACSR 150	-14.5	79.5	0	1		0 ▾
Delete ▾	0	IACSR 150		79.5	0	1		0 ▾
Do not delete ▾	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ▾
Do not delete ▾	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ▾
Do not delete ▾	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ▾
Do not delete ▾	4	TACSR 410	11.2	72.0	0	6	0.5	0 ▾
Do not delete ▾	5	TACSR 410	11.6	57.5	0	6	0.5	0 ▾
Do not delete ▾	6	TACSR 410	12.0	43.0	0	6	0.5	0 ▾

The 'line/cable database' screen appears. Select the desired line type.



If the desired line type does not exist, save the data and close the line-placement screen. Register the line type in the line/cable database (see Section 5), then load the saved data and continue the procedure.

3.3 Calculating line constants and displaying results

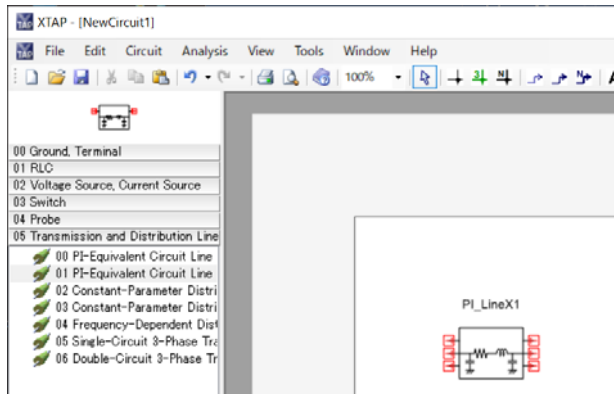
See Sections 2.3 and 2.4.

4 Using Line Model Data Created in XTLC

4.1 How to specify the data for the distributed-parameter line model

The following distributed-line constant parts can be used for created line-model data:

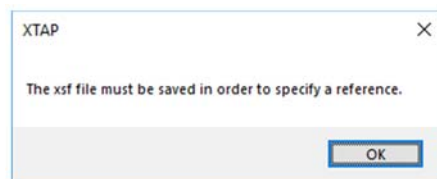
- 01 π -type equivalent-circuit line model (XTLC)
- 03 Fixed-parameter distributed-constant line model (XTLC)
- 04 Frequency-dependent distributed-constant line model



Click 'Browse' in the 'parameter setting' screen to select line-model data. It is also necessary to specify the number of phases (number of conductors) for the selected line model data along with the line length. (It is not necessary to specify the line length in the case of a distributed constant line model that is dependent on frequency).



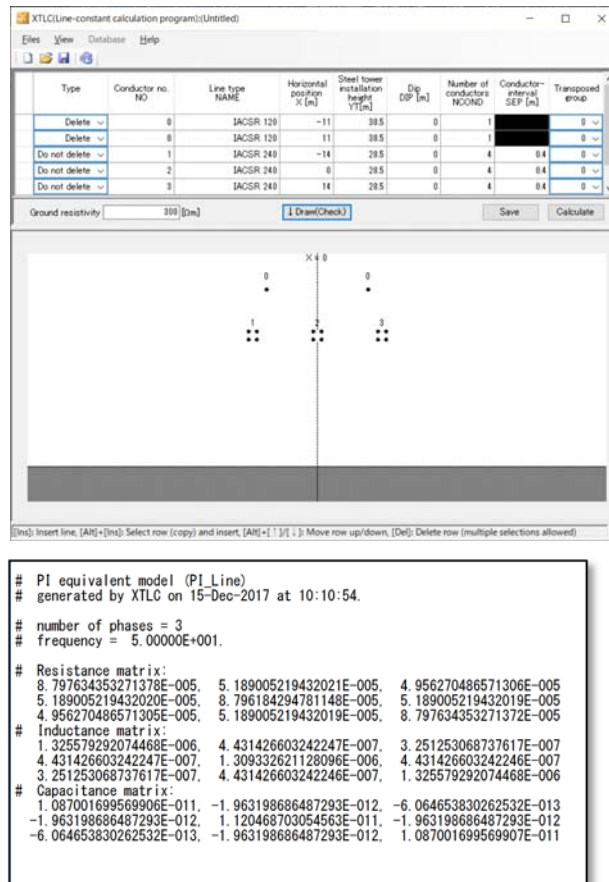
If the following message appears when clicking 'Browse', select 'Save as' from the 'File' menu on the menu bar to save the circuit diagram.



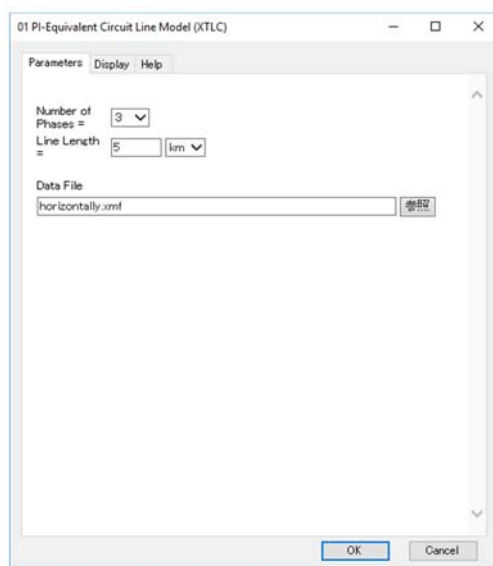
An example is shown below.

- Overhead line example

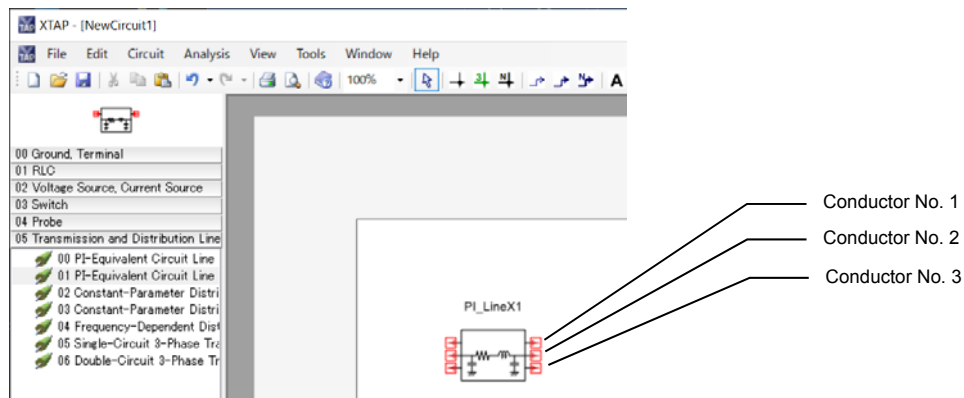
When a conductor placement in XTLC as shown in the figure on the left below is specified and then calculations are executed by selecting the π type equivalent circuit model, line model data is created as shown in the figure on the right below.



This example has three conductors and three phases. Therefore, specify '3' as the number of phases in the parameter setting screen. Select the line-model data above as the data file.

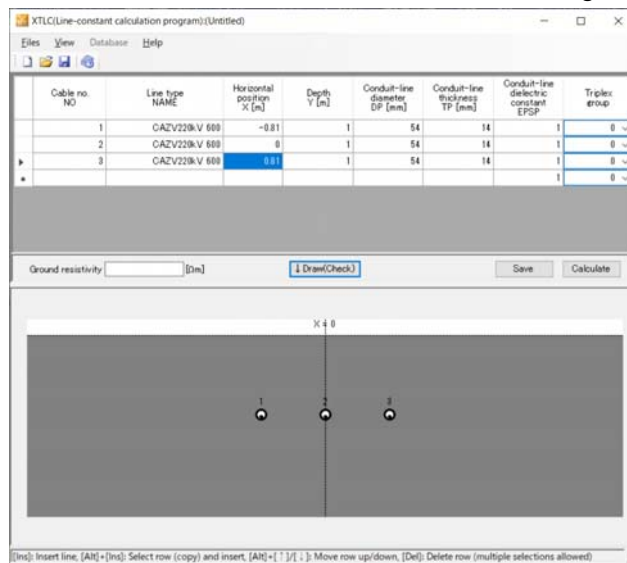


The number of part terminals in the circuit diagram is then updated to 3. The terminal sequence is the conductor number sequence specified in XTLC.



- Cable example

When a conductor placement in XTLC as shown in the figure on the left below is specified and calculations are executed by selecting the fixed-constant line model, line model data is created as shown in the figure on the right below.

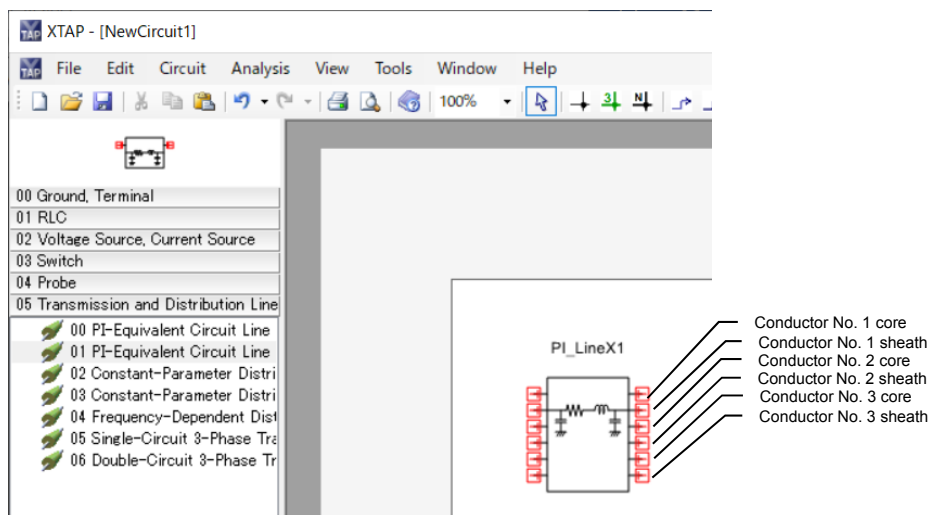


```
## Constant-parameter line model (CP_Line)
## generated by XTLC on 15-Dec-2017 at 10:20:37.
##
## number of phases = 6
## frequency = 5.00000E+001.
##
## Characteristic impedance, propagation velocity, and resistance pul
## for each mode:
2. 972326908382432E+002, 3. 159105162634020E+007, 3. 668319896976460E-004
1. 227186786011015E+002, 8. 317946528250203E+007, 7. 050765312056788E-005
1. 176880763622533E+002, 9. 901337678855768E+007, 6. 645058131619280E-005
4. 521474950815233E+001, 1. 275683485622399E+008, 7. 891812727780672E-005
4. 423883503844301E+001, 1. 312387314873098E+008, 7. 135928556933606E-005
4. 335513851276821E+001, 1. 359837004898909E+008, 6. 762140702187589E-005
##
## Current transformation matrix:
-1. 584153312457315E-003, -3. 248010232970500E-002, -7. 258210685965727E-002, -4. 190390423946063E-001, -7. 122868059792730E-001, 5. 72093840081267E-001
8. 084180185341912E-001, -9. 995790846765363E-001, -5. 680607949657368E-001, 4. 784474386698851E-001, 7. 403377494424469E-001, -5. 706882451779088E-001
-6. 471899427155092E-004, -1. 887379141862766E-015, 1. 404242915834744E-001, 8. 112427134639459E-001, 1. 931788062847772E-014, 5. 910257838259694E-001
8. 341811038951929E-001, 5. 328070518200751E-015, 1. 099834510856578E+000, -9. 26178835463520E-001, -1. 754152378807747E-014, -5. 904515421587737E-001
-1. 584153312457315E-003, 3. 248010232970511E-002, -7. 258210685964817E-002, -4. 190390423945189E-001, 7. 122868059792379E-001, 5. 720938400808080E-001
8. 084180185341897E-001, 9. 995790846765322E-001, -5. 680607949657289E-001, 4. 784474386697932E-001, -7. 403377494424119E-001, -5. 706882451778400E-001
```

This example has three conductors and three phases, with each phase consisting of a core-and-sheath combination. Therefore, specify '6' as the number of phases in the parameter setting screen. Select the above line-model data as the data file.



The number of part terminals in the circuit diagram is then updated to 6. The terminal sequence is the conductor number sequence specified in XTLC. For each conductor, the core is set first, then the sheath.



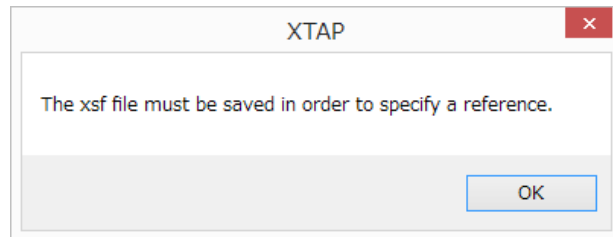
4.2 How to specify the data for matching resistance

The created model data of the matching resistance can be used by using the following component.

- 07 Matching Resistance

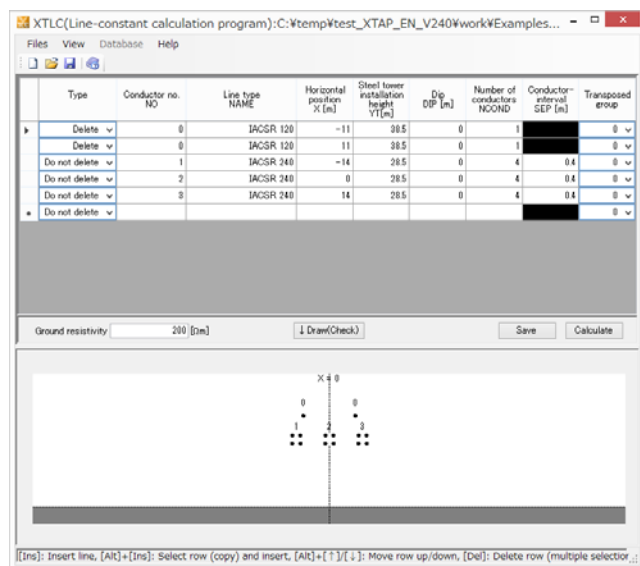
You can set the model data file to this component by selecting “Choose file”, in the parameter screen. Note that it is necessary to choose the number of phases used in the selected model data.

When the following screen appears when you click on “Choose file”, it is necessary to save the circuit schematic file by clicking on File > Save as, in the menu bar.



- Example using an overhead line

Specify the line position parameters in XTLC, like the figure below, then click on the button “Calculate”. Finally, choose “Matching Resistance” in the window that appears on the screen. After the calculation ends, the data about the matching resistance matrix will be displayed on the screen, like the figure below.

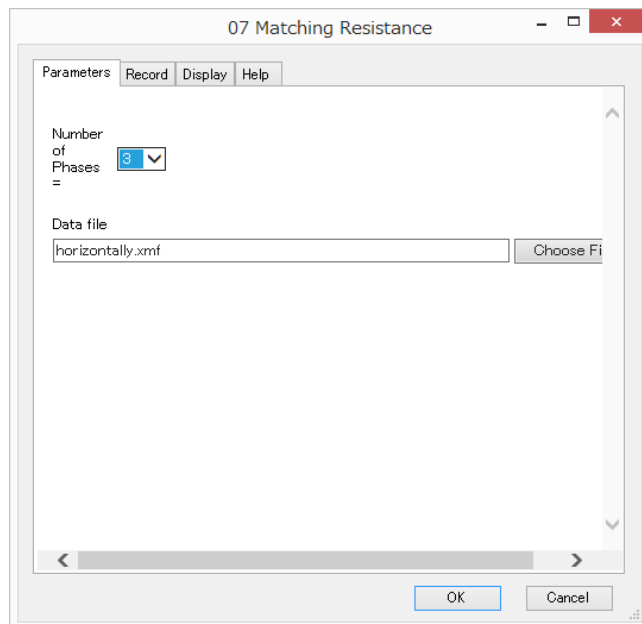


```
# Matching resistance model (MR_model)
# generated by XTLC on 15-Dec-2017 at 10:30:00.

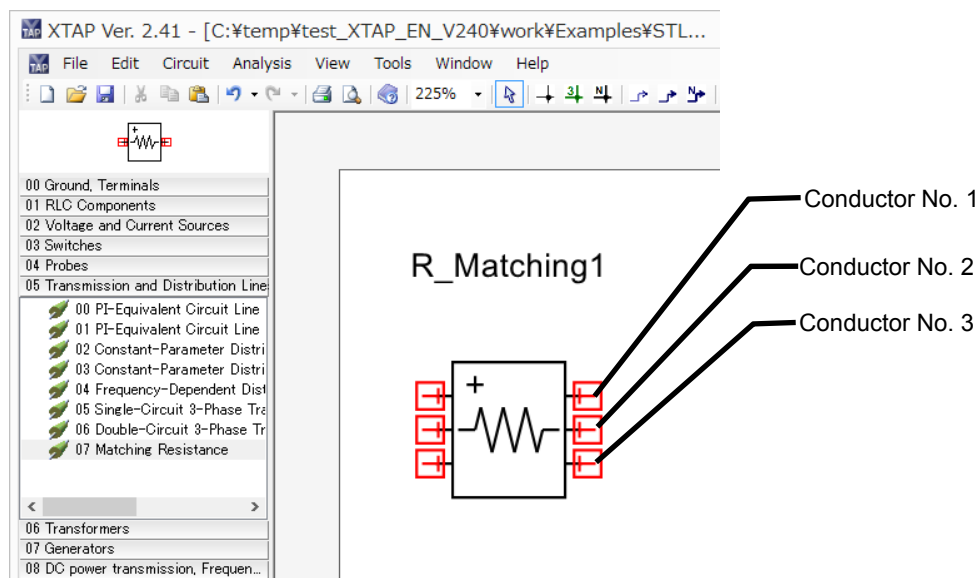
# number of phases = 3
# frequency = 1.00000E+003.

# Matching resistance matrix:
3.433253471860778E+002, 8.254967115982926E+001, 4.906233190223031E+001
8.254967115982930E+001, 3.414932169438846E+002, 8.254967115982927E+001
4.906233190223029E+001, 8.254967115982926E+001, 3.433253471860777E+002
```

Go back to XTAP and place the component “07 Matching Resistance” on the circuit schematic file and save it. In this example a 3-phase 3-conductors line is considered, therefore the parameter “Number of phases” of the XTAP component “07 Matching Resistance” is set to “3”. Moreover, the data file calculated by XTLC is chosen in the “Data file” parameter.

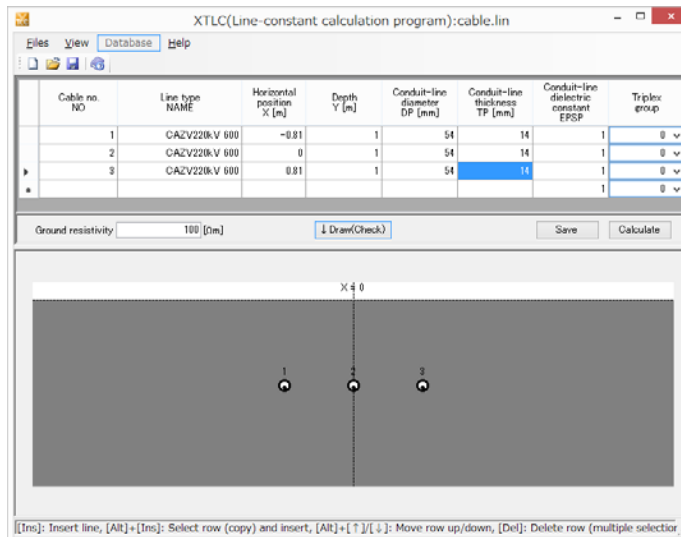


After you set these values and close the properties screen of the component, the number of terminals on each side of the component will be updated to be three. Moreover, the order of the terminals is the same order as specified in XTLC.



- Example using an underground cable

Specify the cable position parameters in XTLC, like the figure below, then click on the button “Calculate”. Finally, choose “Matching Resistance” in the window that appears on the screen. After the calculation ends, the data about the matching resistance matrix will be displayed on the screen, like the figure below.

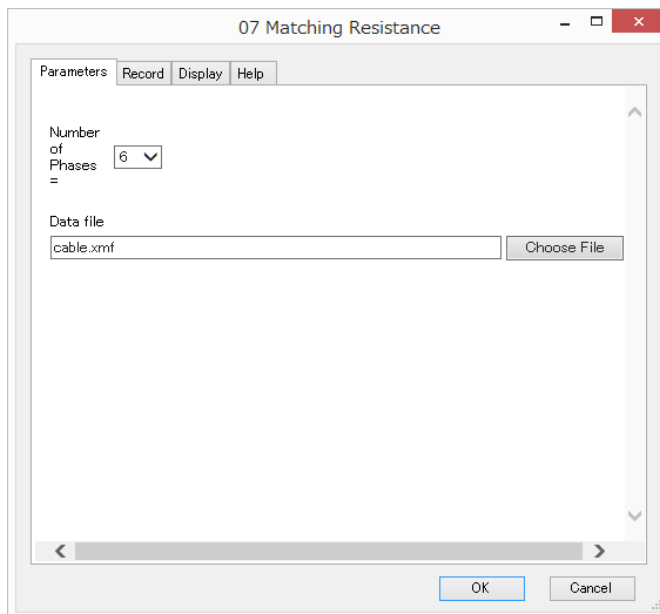


```

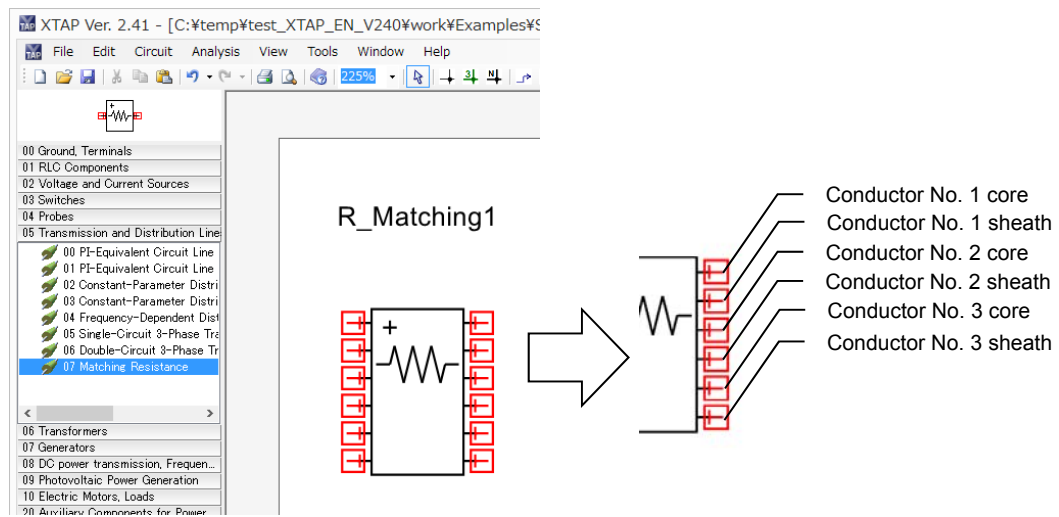
# Matching resistance model (MR model)
# generated by XTLC on 15-Dec-2017 at 10:38:26.
#
# number of phases = 6
# frequency = 1.00000E+003.
#
# Matching resistance matrix:
1. 194200748458744E+002. 8. 064406622622302E+001. 2. 860075999623837E+001. 2. 861797065603115E+001. 2. 332172964247621E+001. 2. 333476506847332E+001
8. 064406622622296E+001. 8. 057851796668292E+001. 2. 861797065603043E+001. 2. 863479841799112E+001. 2. 333476506847240E+001. 2. 334756921469983E+001
2. 860075999623719E+001. 2. 861797065602998E+001. 1. 176959494544502E+002. 7. 891693276165128E+001. 2. 860075999623617E+001. 2. 861797065602900E+001
2. 861797065603032E+001. 2. 863479841799101E+001. 7. 891693276165240E+001. 7. 894847732328951E+001. 2. 861797065603028E+001. 2. 863479841799098E+001
2. 332172964247524E+001. 2. 333476506847233E+001. 2. 860075999623760E+001. 2. 861797065603039E+001. 1. 194200748458744E+002. 8. 064406622622290E+001
2. 333476506847234E+001. 2. 334756921469977E+001. 2. 861797065603033E+001. 2. 863479841799103E+001. 8. 064406622622290E+001. 8. 057851796668288E+001

```

Go back to XTAP and place the component “07 Matching Resistance” on the circuit schematic file and save it. In this example a 3-phase 3-conductors line is considered, however for each cable its core and its sheath are considered separately. Therefore, it is necessary to set the parameter “Number of phases” of the XTAP component “07 Matching Resistance” to “6”. Moreover, the data file calculated by XTLC is chosen in the “Data file” parameter.



After you set these values and close the properties screen of the component, the number of terminals on each side of the component will be updated to be six. Moreover, the order of the terminals is the same order as specified in XTLC. However, for each cable its core and its sheath are considered as different phases



5 Registering Information in Line/Cable Databases

This section describes how to register information in line/cable databases. Before registering information, close the line-placement screen if it is open.

5.1 Databases

There are six line/cable databases:

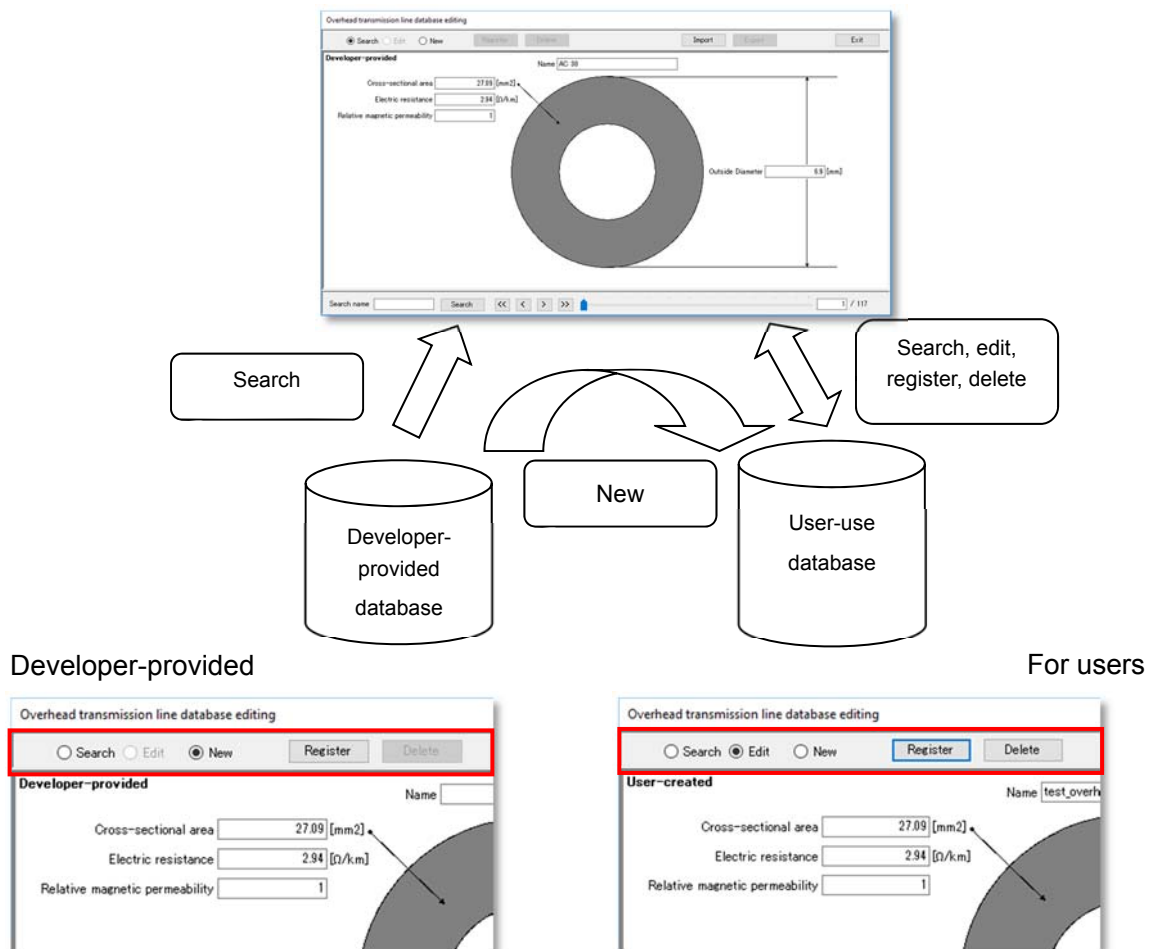
☐ Developer-provided databases

- Overhead transmission lines: Overhead transmission lines.cbl
- Overhead distribution lines: Overhead distribution lines.cbl
- Underground cables: Underground cables.cbl

☐ Databases for users

- Overhead transmission lines: Overhead transmission lines (for users).cbl
- Overhead distribution lines: Overhead distribution lines (for users).cbl
- Underground cables: Underground cables (for users).cbl

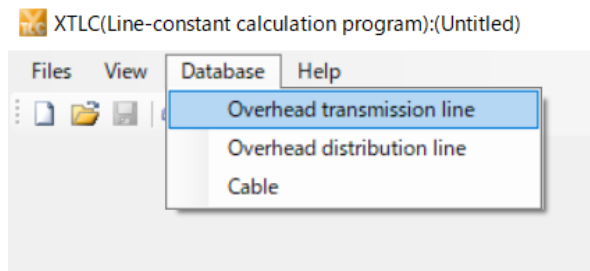
Searches are the only operation that can be done on developer-provided databases. These databases cannot be updated by adding or deleting information. Databases for users can have new or edited information registered in them. You can edit developer-provided information by searching for the required information and then performing the 'New' operation to rename it and register it in a database for users.



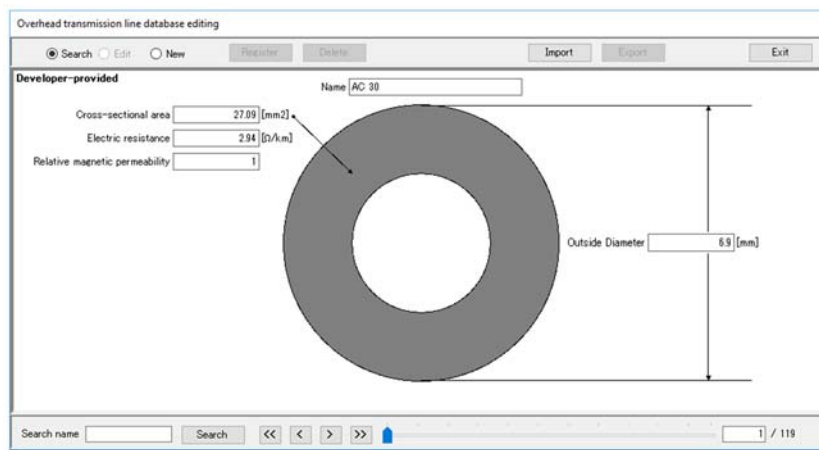
The search function and track-bar slider function enable simultaneous access to developer-provided and databases for users, enabling seamless use of the databases by the user.

5.2 Overhead transmission lines

Select 'Overhead transmission line' from the 'Databases' menu on the menu bar.



The 'overhead-transmission-line database editing' dialog then appears.

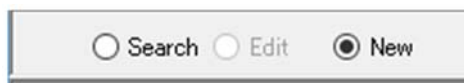


Different operations can be performed on the database, depending on whether the referenced information is registered in the developer-provided overhead-transmission-line database or the one for users. See Section 5.1 for the available operations.

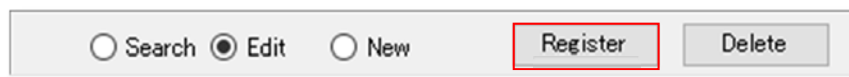
To edit previously registered data, select the 'Edit' radio button.



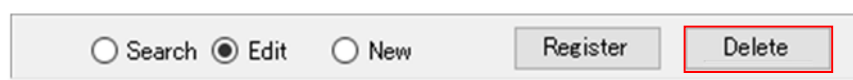
To register new data, select the 'New' radio button.



After revising or entering the data, click 'Register' to register the information in the database.



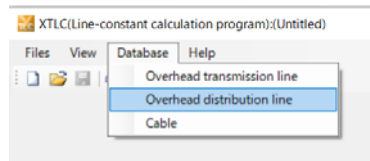
To delete registered data, select the 'Edit' radio button and click 'Delete'.



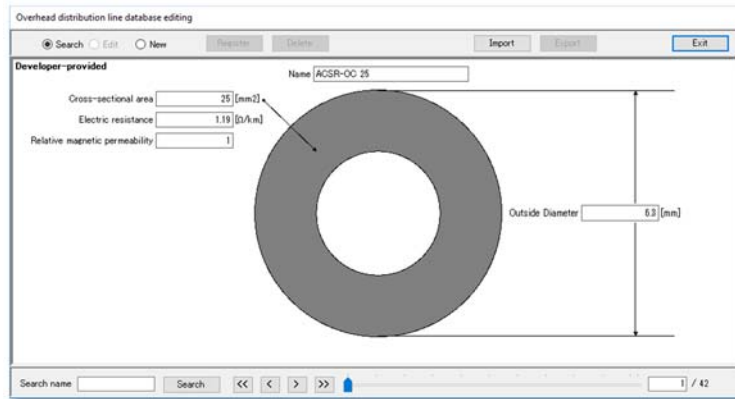
To finish editing, click 'Exit'.

5.3 Overhead distribution lines

Select 'Overhead distribution line' from the 'Databases' menu on the menu bar.



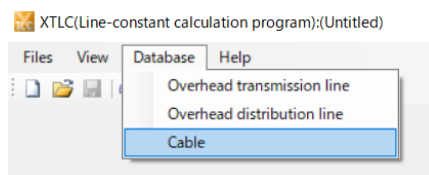
The 'overhead-distribution-line database editing' dialog appears.



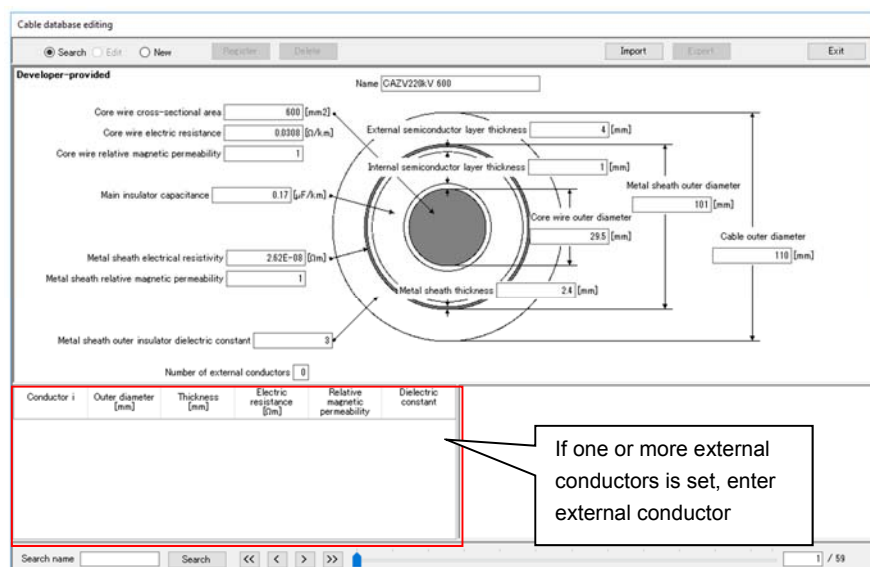
See Section 5.2 for details of how to register information.

5.4 Cables

Select 'Cable' from the 'Databases' menu on the menu bar.



The 'cable database editing' dialog appears.



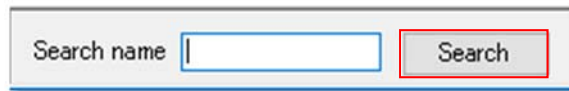
See Section 5.2 for details of how to register information.

5.5 Line/cable database functions

This section describes the line/cable database functions.

Search

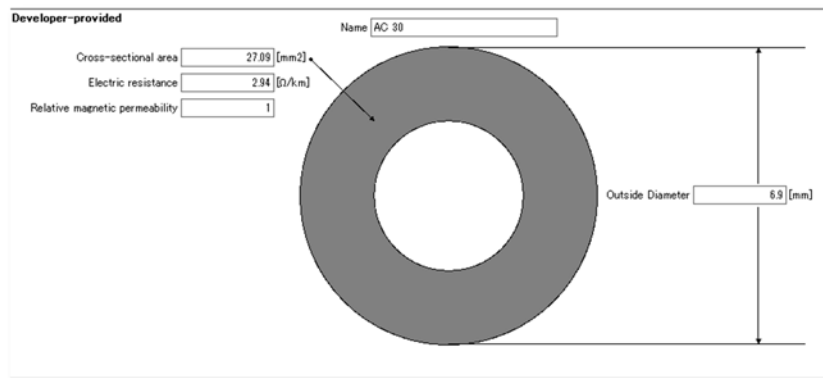
This facility is used to search the databases for lines or cables. Enter the text string to search for in the 'Search name' field, and click 'Search'. Searches are not case-sensitive and blanks are ignored.



Search name

The program searches for results containing the entered string. When multiple search results are returned, clicking 'Search' again displays the next search result.

Example: When 'AC' was entered as the search string, 'AC30' is displayed.



Developer-provided

Name

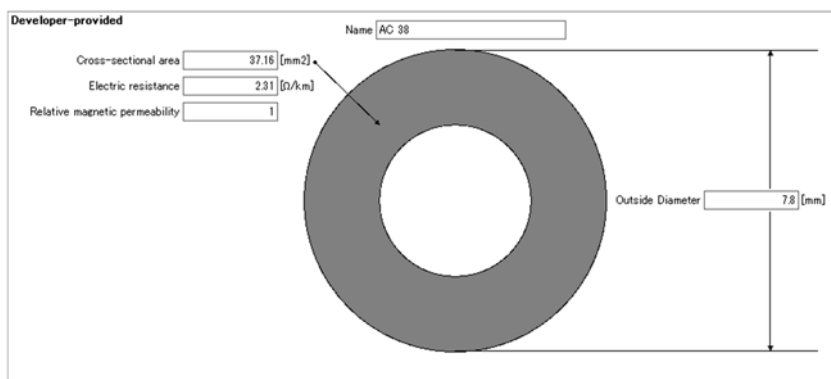
Cross-sectional area [mm²]

Electric resistance [Ω/km]

Relative magnetic permeability

Outside Diameter [mm]

Clicking 'Search' again displays the next search result, 'AC38'.



Developer-provided

Name

Cross-sectional area [mm²]

Electric resistance [Ω/km]

Relative magnetic permeability

Outside Diameter [mm]

Display switching buttons

The display switching buttons are used to change the data displayed.

- [\ll] button

Displays the first data.

- [\gg] button

Displays the last data.

- [\lt] button

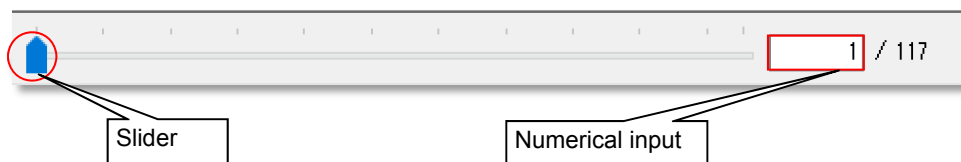
Displays the data immediately before the current data.

- **[>] button**

Displays the data immediately after the current data.

Track bar

Moving the track-bar slider right or left scrolls the displayed data forward or backward. A number can also be entered in the input field, before pressing the 'Enter' key to display the data at the indicated location.



Import

This registers user-created line/cable information in CSV format to a database for users. Overhead lines and cables each use a different file format.

- Import file layout

Overhead lines	Cables
Name	Name
Cross-sectional area	Number of external conductors
Outer diameter	Core wire cross-sectional area
Electrical resistance	Core wire outer diameter
Relative magnetic permeability	Core wire electrical resistance
	Core wire relative magnetic permeability
	Flag
	Main insulator capacitance, or dielectric constant
	Inner semiconductor layer thickness
	Outer semiconductor layer thickness
	Metal sheath outer diameter
	Metal sheath thickness
	Metal sheath electrical resistivity
	Metal sheath relative magnetic permeability
	Cable outer diameter, or metal sheath outer insulator outer diameter
	Metal sheath outer insulator dielectric constant

- Items with a pound sign (#) in the first column are treated as comments.

Example

#Name, cross-sectional surface area, outer diameter, electrical resistance, dielectric constant

AC 1,27.09,6.9,2.94,1

AC 2,125.5,14.5,0.688,1

Export

This outputs information registered in a database for users to a CSV-format text file.

Appendix 2

XFIT (System Identification Program)
Version 1.30

Instruction Manual

Central Research Institute of Electric Power Industry (CRIEPI)
October, 2021

XFIT (System Identification Program for XTAP)

Version 1.30

Instruction manual

Central Research Institute of Electric Power Industry (CRIEPI)

October, 2021

- Note that starting with Version 1.20, the file format of the frequency-characteristic data described in Section 3 of this manual has changed.

1. Overview

XFIT is one of the auxiliary programs accompanying the power system transient analysis program XTAP (Expandable Transient Analysis Program) developed by CRIEPI. XFIT is a fitting program that identifies a linear model given numerical frequency-characteristic data. It can identify models for multiple-input/multiple-output systems (multiple phases in the case of electrical circuit elements). It can also create an electrical-circuit model that uses the given frequency characteristic as its admittance frequency response, and use it in XTAP. This program was previously provided as a stand-alone program called SysIP (System Identification Program).

2. Linear Model fitting

Multiphase (multiple-input/multiple-output) pole-residue models are linear models fitted to reproduce a frequency characteristic provided in the form of numerical data. In other words, the numerical data provided follow a model expressed as

$$Y(j\omega) \cong \sum_{n=1}^N \frac{R_n}{j\omega - p_n} \quad (1)$$

where $Y(j\omega)$ is the frequency-response matrix provided by the user in the form of numerical data (in the case of M phases, this is a matrix of size $M \times M$), R_n is the residue matrix (of size $M \times M$), p_n is a pole, and ω is the angular frequency. Models were fitted using a least-squares approach. See the following references for the detailed procedure.

- [1] [CRIEPI Research Report T03013: "A sub-network reduction method for the calculation of electromagnetic transients in a power system \(Part 1\)"](#) [in Japanese] (← Can be downloaded from the CRIEPI website.)
- [2] [T. Noda, "Identification of a multiphase network equivalent for electromagnetic transient calculations using partitioned frequency response", IEEE Trans., Power Delivery, Vol. 20, No. 2, Part 1, pp. 1134-1142, Apr. 2005](#) (← Can be downloaded from the IEEE website.)
- [3] [T. Noda, "A binary frequency-region partitioning algorithm for the identification of a multiphase network equivalent for EMT studies", IEEE Trans., Power Delivery, Vol. 22, No. 2, pp. 1257-1258, Apr. 2007](#) (← Can be downloaded from the IEEE website.)
- [4] [J. Martinez \(ed.\) "Power System Transients: Parameter Determination", CRC Press](#) (← Link

to Amazon listing; the procedure is described in Appendix A of this book.)

3. Frequency-Characteristic Data Preparation

(1) The pole-residue models described by Formula (1) are fitted to reproduce a provided numerical frequency characteristic. This section describes the method for preparing the frequency characteristic data. XFIT assumes the data are stored in files with the following format:

- The numerical data is separated by spaces or tabs.
- If the model to be identified has M phases (M inputs and M outputs) (i.e., the $Y(j\omega)$ term in Formula (1) has size $M \times M$), the frequency-characteristic data are stored in M separate files.
- Each file stores the data for one row of $Y(j\omega)$.
- The number of frequencies must be the same in each of the M data files.
- The data format of the file storing the j th row of $Y(j\omega)$ is as follows:

```
f(1) mag(1,j,1) phi(1,j,1) mag(2,j,1) phi(2,j,1) ... mag(M,j,1) phi(M,j,1)
f(2) mag(1,j,2) phi(1,j,2) mag(2,j,2) phi(2,j,2) ... mag(M,j,2) phi(M,j,2)
:      :      :      :      :      :
f(k) mag(1,j,k) phi(1,j,k) mag(2,j,k) phi(2,j,k) ... mag(M,j,k) phi(M,j,k)
:      :      :      :      :      :
f(K) mag(1,j,K) phi(1,j,K) mag(2,j,K) phi(2,j,K) ... mag(M,j,K) phi(M,j,K)
```

where $f(k)$ is the k th frequency;
 $\text{mag}(i,j,k)$ is the amplitude of element (i,j) at the k th frequency;
 $\text{phi}(i,j,k)$ is the phase [deg] of element (i,j) at the k th frequency;
 K is the number of frequencies.

- Note that this is an updated format created for the current version of XFIT (Version 1.20).
- The first few lines of each file can be used freely as a header. Header lines are automatically skipped.

4. Five Steps for Model Fitting

Using the XFIT window (shown below), follow the five steps below to fit the model.

The screenshot shows the XFIT 1.3 window with the following settings:

- Number of phases:** Nph = 1
- Frequency response data files:** Icol = 1, Not yet selected. A 'Select' button is present, and a message states 'No files selected.'
- Error tolerance in relative quantity:** EpsR = 1.0 %
- Low level fitting parameters:**
 - Nmin = 6, Nmax = 16
 - Nitr = 30, Nsvd = 2
 - ☒ symmetric, ☐ limit poles
 - ☐ pole clustering, EpsP = 5.0 %
 - ☒ outputs
 - A 'Default' button is located below these options.
- Start fitting:** A 'Start' button is present, and a message states 'Not yet fitted.'
- At the bottom are 'Close', 'Help', and 'About' buttons.

Step 1

(Section in window: Number of phases)

Specify the number of phases M ('Nph' on the screen). M corresponds to the size of $Y(j\omega)$.

Step 2

(Section in window: Frequency response data files)

Specify the M files prepared as described in the last section.

Step 3

(Section in window: Error tolerance in relative quantity)

Specify the error tolerance as a relative value (%). This value is used for pole fitting. Specify a value in the range $0 \leq \text{EpsR}$. The default value is 1%.

Step 4

(Section in window: Low level fitting parameters)

Specify detailed parameter settings if necessary. Note that most of these parameters are set to default values considered to be appropriate, and should not normally need alteration.

- Nmin: Start order for an optimal model-order search. (Range: $1 \leq N_{min} \leq N_{max}$; Default: 6)
- Nmax: Stop order for an optimal model-order search. (Range: $N_{min} \leq N_{max}$; Default: 16)
- Nitr: Iteration stop count for a rational function fitting process. (Range: $1 \leq N_{itr}$; Default: 30)
- Nsvd: Constant used in SVD (singular value decomposition) calculations in the least-squares method. If the number of significant digits in a given value is less than this constant, the value is removed from the simultaneous equations. (Range: $0 \leq N_{svd} \leq 16$; Default: 2)
- symmetric: Flag indicating whether the system to be fitted (i.e., the matrix provided) is symmetric (checked) or asymmetric (unchecked). (Default: checked)
- limit poles: If this flag is checked, the poles with an absolute value greater than the maximum angular frequency are removed. (Default: unchecked)
- pole clustering: Flag determining whether or not to consolidate nearby poles into a single pole. Poles are consolidated when this setting is checked, and kept separate when unchecked. (Default: unchecked)
- EpsP: When the 'pole clustering' setting above is checked, any poles within the radius specified by this parameter are consolidated into a single pole. The value is specified as a relative value (%). (Default: 5%)
- outputs: When this flag is checked, intermediate results are output. When not checked, intermediate results are not output. (Default: checked)

Step 5

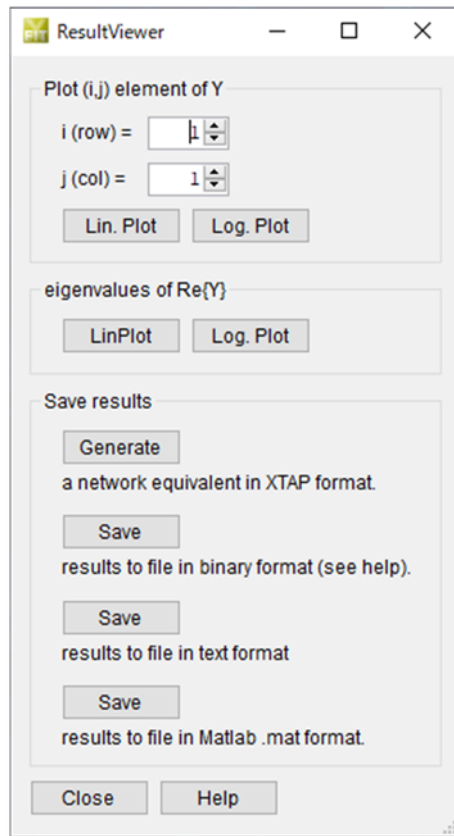
(Section in window: Start fitting)

Press 'Start' to begin fitting the model. When it is finished, the 'ResultViewer' window opens to display the results.

5. Displaying and Saving Identification Results

The 'ResultViewer' window shown below opens automatically when model fitting is completed. This window can be used to display the following graphs:

- Plot (i,j) element of Y: shows the fitting of model element (i, j). It compares the frequency characteristic of element (i, j) in the data provided to the frequency characteristic of element (i, j) in the model. The horizontal axis (frequency) may be either linear or logarithmic.
- Eigenvalues of $\text{Re}\{Y\}$: plots the eigenvalues of $\text{Re}\{Y(j\omega)\}$ versus frequency for the fitted model. The horizontal axis (frequency) may be either linear or logarithmic. If the eigenvalues of $\text{Re}\{Y(j\omega)\}$ are positive for all the frequencies, the fitted model is guaranteed to be positive-definite (i.e., passive).



The fit results can be saved in the following formats from the 'ResultViewer' window:

- network equivalent in XTAP format: generates and saves data in a format that can be used from a 'Condensed multiphase circuit model' (NetEquiv) XTAP part. An admittance element with the frequency response fitted by XFIT can be created in an XTAP circuit diagram by placing a condensed multiphase circuit model in the circuit diagram, double-clicking on it, and using the window that appears to specify the file generated by this item.
- binary format: outputs the identification results as a binary file.
- text format: outputs the identification results as a text file.
- MATLAB format: outputs the identification results as a MATLAB *.mat file, enabling them to be loaded into MATLAB using the 'load' command.

Binary files have the following format:

- 1) Number of phases M (integer)
- 2) Number of real poles N_r (integer)
- 3) Number of complex poles N_c (integer)
- 4) Real poles (double; number of values = N_r)
- 5) Real parts of complex poles (double; number of values = N_c)
- 6) Imaginary parts of complex poles (double; number of values = N_c)
- 7) Residue matrix of real poles (double; number of values = $M \times M \times N_r$; each matrix row is provided first, then N_r matrices)
- 8) Real part of residue matrix of complex poles (double; number of values = $M \times M \times N_c$; first each matrix row, then N_c matrices)
- 9) Imaginary part of residue matrix of complex poles (double; number of values = $M \times M \times N_c$; first each matrix row, then N_c matrices)

where

'integer' is an integer type (32-bit)

'double' is a floating-point real number type (64-bit)