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

 Introduction	1 1 2 3 5 7 8 9
3 Workflow From Circuit-Diagram Creation Through Analysis	.22
4 Creating Circuit Diagrams 4.1 Part types 4.2 Part placement (electrical-circuit part/control-block part) 4.3 Part operations 4.4 Part properties screen	.24 .24 .24 .26 .31
 5 Part Connections	. 35 . 35 . 38 . 39 . 40 . 40 . 43 . 47
 6 Analysis Processes. 6.1Analysis settings. 6.2Analysis procedure (analysis type: transient analysis) 6.3Analysis procedure (analysis type: steady-state initialization → transient analysis) 6.4Analysis procedure (analysis type: power flow calculation → steady-st initialization → transient analysis) 6.5Analysis procedure (analysis type: frequency characteristic calculation) 6.6Files related to analysis processes. 6.7Search function 	.53 .54 sis) tate .62 .68 .71 .72
 7 Consecutive-Analysis Function 7.1Overview of consecutive analysis function 7.2Opening the consecutive-analysis settings window 7.3Consecutive-analysis settings window 7.4Setting the number of cases and of parameters 7.5Parameter input method 7.6Ending/canceling parameter settings 7.7Executing a consecutive analysis results 7.8Statistical processes for analysis results 7.9Executing statistical processes 7.10Displaying analysis results in XPLT 	.73 .73 .74 .75 .76 .80 .81 .83 .84 .88
8 Statistical Switching-Surge Calculation Function 8.1Overview of statistical switching-surge calculation function 8.2Opening the settings window for a switching-surge calculation	. 89 . 89 . 89

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

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 syste	em
Operating	Microsoft Windows 8, 8.1 (32-bit and 64-bit versions)
system	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 requir	ed
Ooffician	MATLAB Compiler Runtime R2015b
Sonware	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 circuitdrawing 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.

TAP	ХТАР	×
	This file (""PE-01.xsf"") was created using a version of XTAP prior to XTAP Ver. 2.41. This file will be converted into an xsf file for Ver. 2.41 (although some files may be converted automatically, others will need to be converted by the user). Once this is finished, it will be impossible to properly open this file in older versions of XTAP. If you need to retain the original file, select "Stop" and back the file up. (You should save the original file until you can confirm that "'PE-01.xsf" operates without issue in Ver. 2.41?	
	Abort Continue	

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.

ТАР			Output circuit	diagram im	age			×
🔄 🏵 🗉 🕇 🌆	<< W	ork ⊧ Exa	mples → CB-01	~	Ç	Search CB-01		Q
Organise 🔻 Ne	w fold	er						0
▷ 퉲 Intel ▷ 퉲 kehi	^	Name	*			Date modified	Туре	
🍌 PerfLogs > 🍌 Program File	25			No items ma	tch you	ur search.		
Program File	es (
Users								
▷ 🌆 Windows ▷ 퉲 XTAP	Ŷ	۲						>
File name:	lmag	e.emf						¥
Save as type:	Emf li	mage (.em	f) (*.emf)					~
Hide Folders						Save	Cancel	

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

Edit	:	
uŋ.	Undo	Ctrl+Z
Çi.	Redo	Ctrl+Y
Ж	Cut	Ctrl+X
ila.	Сору	Ctrl+C
12	Paste	Ctrl+V
	Paste with same name	Ctrl+Shift+V
×	Delete	Del
	Select All	Ctrl+A

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.

A component with the same	name exists in the	canvas into which it	is being pasted.
component name: R1 component names must be pasted. Change the name of the co front of the original compor	unique, so compone mponent and paste ent name.)	ents with the same na it? (An underbar will t	ame cannot be be placed in

➢ 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.

A node with the same na	me exists in the pastir	ng canvas.	
Node name: Node_a The node with the same connected), even if it is Paste with the current r	name will be handled a not connected (it is de name?	is the same node (it stached) on the circi	will be uit diagram.

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.

				Record Properties			Show Properties	
	Type ∆	Name	Node Voltage Differences	Branch Current	Branch Power	Name	Parameter	Record
			All ON/OFF	MII ON/OFF	MII ON/OFF	🔽 All ON/OFF	MII ON/OFF	All ON/OFF
1	00 DC Voltage Source	Vdc				V	V	
2	00 Ground	GND				V		
3	00 Resistor	Ruv				V	7	
4	00 Resistor	Rvw					~	
5	00 Resistor	Rwu					~	
6	00 Voltage Probe (XPLT Output)	Vuv	✓			I		
7	00 Voltage Probe (XPLT Output)	Vwu	✓			I		
8	00 Voltage Probe (XPLT Output)	Vvw	✓			V		
9	01 Inductor	Lu		~		v	~	
0	01 Inductor	Lv		~		v	~	
1	01 Inductor	Lw		v		V	~	

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

<u>A</u> na	alysis					
P	Analysis <u>S</u> etup					
	<u>A</u> nalyze					
	Save & Analize F5					
	Consecutive-analysis settings					
$\flat \flat$	Run consecutive-analysis					

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 alysis

• 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

<u>V</u> ie	w
~	<u>T</u> oolbar
~	<u>S</u> tatus Bar
#	<u>G</u> rid
#	Dis <u>p</u> lay print-area.

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

Tools

Circuit diagram canvas size (X)

Component canvas size

- XPLT (graphing program)
- XTLC (line-constant calculation program)
- XFIT (system identification program)

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).

Horizontal	📓 × Vertical	-	2
	Television and a real		
	The second parameters of the second s		
	A surface of the second		
	Contracting of American Strategy Transmission Contracting Strategy Transmi		
	The second secon		
Paper	Construction American Construction American Construction Constru		
Paper Size:	A4		~
Paper Sige: Canvas orienta	A4 tion Margins (millimeters)		~
Paper Size: Canvas orienta @ Portrait	A4 Martins (nillmeters) Left: 25.4	Right	~ 25.4

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

Win	wob
	Open New Window
	Display overlapped (C)
	Display horizontally (V)
	Display vertically
	Close all
	Icon alignment
~	1 NewCircuit1
	2 NewCircuit2

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

Hel	p	
3	How to use (C)	
	About	

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)



• 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)' \rightarrow '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.

	TAP	XTAP -	[NewCi	rcuit1]		_
	TAP	File	Edit	Circuit	Analysis	View
	1	2	- X	ի 🖺	10 - (21	- 🖪
	00 Gr	round	Termina	1		
<	01 R	LC		\sim		
	02 V	oitage	Source,	Current S	ource	
	03 S(witch				
	04 Pr	obe				
	05 Tr	ansmi	ssion an	d Distribu	tion Line	
	06 Tr	ansfor	mer			
	07 G	enerati	or			
	08 D	C powe	er transr	nission, Fr	equen	
	09 P	V				
	10 EI	ectric	Motor			

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

TAP - [NewCircuit1]	_
File Edit Circuit Analysi	s View
i 🗋 💕 📓 X 🖻 🛍 19 - 0	× - 🗃
₫₩₽	
00 Ground, Terminal 01 RLC	
💋 00 Resistor 🛛 🔺	
🝠 01 Inductor	
🝠 02 Capacitor	
🛒 03 Nonlinear Single-Phase	
🝠 04 Nonlinear Single-Phase	
🛒 05 Nonlinear Single-Phase	
🞻 06 Nonlinear Single-Phase 🌱	
< >	
02 Voltage Source, Current Source	
03 Switch	
04 Probe	

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.



4.3 Part operations

(1) Selecting and moving parts

Clicking on a part selects it and displays green markers on it. The part can then be moved freely by standard drag-and-drop.



Clicking on different areas of a part selects different items.



Clicking on

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.



- 2 selects the part itself. A double click opens the 'properties' screen. See Section 4.4 for information on the 'part properties' screen. Right-clicking displays the 'part' menu, described in Item (2) on the next page.
- 3 selects the terminals (displayed in red), serving as the connection points to other parts.
- 4 selects the 'parameter' label. Its value can be changed on the 'properties' screen. Some parts do not have parameter labels.
- 5 selects the 'initial value' label, which can be changed on the 'properties' screen. Some parts do not have an initial value label.
- 6 This line is displayed when the XPLT output has been specified. See Section 4.4 for more information.

(2) Part menu

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



Cut

• cuts the part.

Сору

• 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.

M XTAP	×	₩ XTAP ×
A component with the same name exists in the canvas into which it is being pasted. component name: O1 component names must be unique, so components with the same name cannot be pasted. Change the name of the component and paste it? (An underbar will be placed in front of the original component name.)		A node with the same name exists in the pasting carvas. Node name: nodeA The node with the same name will be handled as the same node (it will be connected), even if it is not connected (it is detached) on the circuit diagram. Paste with the current name?
Ves Ves to All Abort]	Yes Ves to All Abort

(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 Capacitor	-		×
Parameters Initial Value Record Display Help			
C = 1 4F V			~
			~
	OK	Cance	el 👘

(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.

2 Capacitor						-		×
Parameters	Initial Value	Record [Display	Help				
Specie	fy the initial v	alue -						~
					OK		Cancel	~

(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.

00 Absolute Value	-	n x
Record Display Help		
Output Value		^
		~

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 circuitdiagram 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.

capacitor							-	
Parameters	Initial Value	Record D	Display	Help				
Capacitor	5							^
								-
This brand	ch is a linear :	single-phas	se capa	citor (capa	citance).			
This brand Specify th	ch is a linear : ne initial volta	single-phas pe value fro	se capa om the	citor (capa ″initial val	citance). ue" tab.			-
This brank Specify th	ch is a linear : ne initial volta	single-phas ge value fro	se capa om the	citor (capa ‴initial val	citance). ue" tab. Updated or	n March 1	14, 2012	

(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.



5.1 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.



 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 circuitdiagram canvas. Checking the checkbox displays the node name on the circuitdiagram canvas. The text box can be used to change the node name.

NDT_A		-		×
View Recording				
Name: NDT_A				^
	OK		Cancel	I

(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.

IDT_A	-	×
View Recording		
Node Voltage		~
		~

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.



 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 controlblock 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 function block]).



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

17 Vector Creation		-	×
Parameters Display	-inip		
Number of elements	2 3 4 5 6 7 7 9 9 9		^
			2

2. 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 $\Box \Box$ of the connected terminal will disappear, and an arrow indicating the direction of wiring will appear on the input-side terminal.



3. The wiring is completed once all of the single signals are connected. The output of the part 'vector creation (MTIPN1)' 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 \Box \Box).

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



4. 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.



5. 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.

Arameters Record Display Help	Vector Elem	ent Value	Output				-		×
Output element number	Parameters	Record	Display	Help					
	Output element number								^
						OK		Cance	

6. 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, doubleclick 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 blue 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.



 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.



		Wire name	Connection wire nam
re name	Connection wire name		w14
	WB	2	w16
	w10		

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.



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 circuits diagrams below are equivalent.





6 Analysis Processes

XTAP performs transient analysis (instantaneous-value analysis) and has a steadystate 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 \rightarrow Transient analysis
- (3) Power-flow calculation \rightarrow Steady-state initialization \rightarrow 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 \rightarrow 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 \rightarrow Steady-state initialization \rightarrow Transient analysis

The program uses a power-flow calculation (the 'L method') to determine the steadystate 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

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.

halysis-file save location ysis-file (.xtf file) halysis-type selection ysis-type (Analysis) halysis conditions a name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	PE-01.xtf Transient analysis Case 1 1.0 # s 0.04
ysis-file (.xtf file) malysis-type selection ysis-type (Analysis) malysis conditions e name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	PE-01.xtf Transient analysis Case 1 1.0 # s 0.04
nalysis-type selection ysis-type (Analysis) nalysis conditions e name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	Transient analysis Case 1 1.0 # s 0.04
ysis-type (Analysis) nalysis conditions e name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	Transient analysis Case 1 1.0 µs 0.04
nalysis conditions e name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	Case 1 1.0 µs 0.04
e name (name) ulation time increment (Delta_t) nits ulation end time (T_max) nits	Case 1 1.0 µs 0.04
ulation time increment (Delta_t) nits ulation end time (T_max) nits	1.0 µs 0.04
nits ulation end time (T_max) nits	μs 0.04
ulation end time (T_max) nits	0.04
nits	
	S
switching iteration count (NumBNR)	30
enelson switching iteration count (Nun	nKa 60
ulation result output file (BinFile)	PE-01.xpf
out start time (TimeWindowBegin)	0.02
nits	s
out end time (TimeWindowEnd)	T_max
nits	s
out frequency (OutputFreq)	1
out calculation result in text format/CS	SV · Yes (text format)
format output file (CSV_File)	PE-01.csv
ug level (DebugLevel)	0
a me (name) the case name (name).	
	ulation result output file (BinFile) ut start time (TimeWindowBegin) nits ut end time (TimeWindowEnd) nits ut frequency (OutputFreq) ut calculation result in text format/CS format output file (CSV_File) ig level (DebugLevel) ame (name) the case name (name).

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.

alysis 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
se name (name) cify the case name (name).	
	1 Analysis-file save location Analysis-file (xtf file) 2 Analysis-type selection Analysis-type (Analysis) 3 Analysis conditions Case name (name) Calculation time increment (Delta_t) Units Calculation end time (T_max) Units BNR switching iteration count (NumBNR) Katzenelson switching iteration count (NumKa Calculation result output file (BinFile) Output start time (TimeWindowEegin) Units Output end time (TimeWindowEegin) Units Output requency (OutputFreq) Output calculation result in text format/CSV CSV format output file (CSV File) Debug level (DebugLevel) Ete name (name) cify the case name (name).

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¥xcore.exe	-	×
		^
Computational Core Module		
CRIEPI, Japan.		
>> Data Preprocessing		
0 % done.		
		\sim

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).

🥘 xcoremsg.out - 火モ帳 -	×
ファイル(F) 編集(E) 書式(O) 表示(V) ヘルプ(H)	
	^
XTAP - eXpandable Transient Analysis Program, Ver. 2.41 Computational Core Module	
CRIEPI, Japan.	
Last line read is as follows: L, "Lu",-0.01	
Error on line 72, in the file "C:¥temp¥test_XTAP_EN_V240¥work¥Examples¥PE-01¥PE-01_V241.xtf": inductance value must be greater than zero.	
I	
	~
(15年1月	 >
12 41, 1 34	

6.3 Analysis procedure (analysis type: steady-state initialization \rightarrow 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).

02 Sinusoidal Voltage Source	-		×
Parameters Record Display Help			
Amplitude 500.0 kV v v÷√3 v×√2 Frequency 50.0 Hz v Initial Phase 0.0 des Angle -1 vs v Enter negative value	ative		~
			~
	OK	Cance	

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

1 Applied a file and leasting	
	2011 01 - 10
Analysis-file (.xtf file)	22M-01.Xtt
2 Analysis-type selection	
Analysis-type (Analysis)	Steady-state initialization -> Transient analysis
3 Analysis conditions	
Case name (name)	Case 1
System frequency (SysFreq) [Hz]	50.0
Calculation time increment (Delta_t)	10.0
Units	μs
Calculation end time (T_max)	20.0
Units	ms
BNR switching iteration count (NumBNR)	30
Katzenelson switching iteration count (NumKat)	60
Calculation result output file (BinFile)	SSW-01.xpf
Output start time (TimeWindowBegin)	0.0
Units	s
Output end time (TimeWindowEnd)	T_max
Units	s
Output frequency (OutputFreq)	1
Output calculation result in text format/CSV for	mat No
Output power-flow/steady-state analysis result	outr No
Debug level (DebugLevel)	0
alysis-type (Analysis) ect ""Transient analysis,", ""Steady-state initia	alization -> Transient analysis,"" ""Flow calculation ->

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 \rightarrow 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 = 1.00000e-005 T_max = 2.00000e-002 NumBNR = 30 NumKat = 60
Network Info: NumNodes = 18 NumBranches = 10

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

XTAP ×
Simulation Finished
ОК

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.

SSW-01_pfss.txt - Notepad		-	×
<u>File Edit Fo</u> rmat <u>V</u> iew <u>H</u> elp			
>> Steady-State Solution	n		î
Node Voltages:			
3.12080e+002 [Vrms]], 172.21643 [deg	3]	
3.08566e+002 [Vrms]], 58.44541 [deg	3]	
3.32846e+002 [Vrms]], -51.06452 [deg	3]	
2.28776e+001 [Vrms]], 129.50987 [deg	3]	
1.68838e+001 [Vrms]], 47.58089 [deg	3]	
			~
<			>

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)
- O2 Synchronous generator (d1, q1)
- O3 Synchronous generator (d1, q2)
- O4 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-flowcalculation 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 powerflow 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.

```
VbusBus voltage [V]theta_bus Bus voltage phase [rad]PGbusBus generator active power [W]QGbusBus generator reactive power [var]PLbusBus load active power [W]QLbusBus load reactive power [var]theta_shiftedPhase difference from bus voltage reference phase [deg]
```

Caution

For synchronous generators and infinite bus generators, power-flowcalculation information acquisition is enabled by default (and cannot be disabled). Problems such as information not being considered during steadystate 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.

- 63 -

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.

- 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.

	06 Transmission-Line Branch			×
	Parameters Display Help			
BUS1 BUS2 j0.15 pu RL4 WM Sw LinePP1 LinePP1	Positier-Segance Series Resistance Positier-Segance Series Noticence Positier-Segance Shunt Capacitance Frequency Base Voltace	0 0 V 0.00031 H V 0 EFV 50 He 220 V V		-
				è
		0	Cancel	

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'.



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

Analysis Setup	×
1 Analysis-file save location	
Analysis-file (.xtf file)	MAC-01-Axtf
2 Analysis-type selection	
Analysis-type (Analysis)	Flow calculation -> Steady-state initialization -> Transient analysis
V 3 Analysis conditions	
Case name (name)	MAC-01-A
System frequency (SysFreq) [Hz]	50.0
Reference capacity (VoltAmpereBase) [MVA]	0.1
Calculation time increment (Delta t)	100.0
Units	# s
Calculation end time (T max)	10.0
Units	8
BNR switching iteration count (NumBNR)	30
Katzenelson switching iteration count (NumKat)	60
Calculation result output file (BinFile)	MAC-01-A xpf
Output start time (TimeWindowBegin)	0.0
Units	\$
Output end time (TimeWindowEnd)	T_max
Units	
Output frequency (OutputFreq)	10
Output calculation result in text format/CSV format.	No
Output power-flow/steady-state analysis result output file	No
Debug level (DebugLevel)	0
Case name (name) Specify the case name (name).	
Additional	OK Cancel

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 \rightarrow Steady-state initialization \rightarrow 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 \rightarrow transient analysis' process.

Reference capacity:

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

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

	nment for Windows7	_	×
1.000000	pu], 0.00000 [deg]		^
Active and Re	ctive Powers at Busses (PG, QG, PL, and QL):		10
0.900000.	0.135278, -0.000000, 0.000000 [pu]		
0.000000,	0.000000, 0.000000, 0.000000 [pu]		
BU53 -0.891880,	0.128566, 0.000000, 0.000000 [pu]		
Vode Voltages U(NDT_A)	Solution		
U(NDT_B) 1.28287e+0	2 [Vrms], -103.24911 [des]		
1.28287e+0	2 [Vrms], 136.75089 [des]		
LIGN/TELP/TI//	0 [Vrms]. 0.00000 [des]		
0.00000e+0	0 1/0 01		
0.00000e+0 U(SyncGen1/ 0.00000e+0	[] V0_B) [] [[Yrms]], 0.00000 [des]		

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.

MAC-01-A_pfss.txt - Notepad	72	×
Eile Edit Format View Help		
>> Power-Flow Solution		^
volt-ampere base = 1.00000e+005 [VA] system frequency = 50.0 [Hz]		
Bus Voltages (magnitude and phase angle):		
1.010000 [pu], 16.75089 [deg] BLS2		
0.998893 [pu], 9.06408 [deg] BUS3		
1.000000 [pu], 0.00000 [deg]		
Active and Reactive Powers at Busses (PG, QG, PL, and QL):		
0.900000, 0.135279, -0.000000, 0.000000 [pu]		
0.0000000, 0.000000, 0.000000, 0.000000 [pu]		
-0.891880, 0.128566, 0.000000, 0.000000 [pu]		
>> Steady-State Solution		
Node Voltage:		
1.28287e+002 [Vrms], 16.75089 [deg]		
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 "dynamsg.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.

2 Sinusoidal Voltage Source	- 0	×
Parameters Record Display Help		
Amplitude 10 V V C+73 ×72 Frequency 600 Hz V Initial Phase 00 deg Starting -1 mov		<
	Enter negative value]
		~
		-
3. Select 'Analysis settings' from the 'Analysis' menu in the main window. The analysis settings screen appears.

Analysis Setup	×
1 Analysis-file save location	
Analysis-file (.xtf file)	FS1.xtf
2 Analysis-type selection	
Analysis-type (Analysis)	Frequency characteristic calculation (Freque
S Analysis conditions	
Case name (name)	Case 1
Start frequency (FreqBegin) [Hz]	0.0
End frequency (FreqEnd) [Hz]	0.0
Frequency count (NumFreq)	0
Frequency sampling	Equal interval (linear)
Calculation result output file (BinFile)	PE-01_se.xpf
Output calculation result in text format/CSV	· No
Debug level (DebugLevel)	0
3 Analysis conditions	

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.

xcoremsg.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.

dynamsg.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, .hmtl 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.

Text string to search:				V 50
FWD		<u>E</u> ind	L	<u>C</u> los
Upper/	lowerca	ase letter disting 2		
Search results:				
4				
Path Name		Path name in use		
Qu1/FwD		02 Diode		
Qu2/FwD		02 Diode		
Qv1/FwD	5	02 Diode		
Qv2/FwD	5	02 Diode		
Qw1/FwD		02 Diode		
0w2/EwD		02 Diode		

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.

	<u>A</u> na	alysis	
	P	Analysis <u>S</u> etup	
		<u>A</u> nalyze	
		Save & Analize F5	
ſ		<u>Consecutive-analysis settings</u>]
		Run cons <u>e</u> cutive-analysis	

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 multipleanalysis 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.

K	Consec	utive-ana	lysis	setting	gs - [(ATX/:	P_e	n\work	c\sa	mple\P8	-01	\PE-01.3	sf]					-		>	K
General settings	Statis	tical swite	chine	-surge	calc	lation	set	tines													
Number of cases	;	20 🗘		0	All	Run		釦	Con	secutive	e pri	ocessine	of	analysis	res	sults		ок			
Number of paramet	ters	20 🗧		6	All	Stop		3	Pro	cess set	ting	3					-	Cancel	1		
Matri	x~size			Run all	/stop	all set	tine			Ana	lysi	s result	pro	cess				Close			
parame	ter	Case nu	mber	2										7				0		10	
Bun /ston	ontting	2 Bun		Rin		Bun		Ban	1.	Bin	10	Bun	1	Ban	-	Ban		BRun	13	BR	
1 Param A	setting.	- rouri	*	Part		Pull	-	- run	Y	Se run	1	a run	Y	Full	Y	Se run	1 Y	Se Puer	1×	Se l'SUI	ĺ
2			+																		
3			+																		
4			t				1														
5																					
6																					
7																					
8																					
9																					
10																					
11																					
•			_				_														1

3. Enter the parameter value for each case.

×		Consec	utive-a	nalysi	s settings -	[C:\XTAP_en	\work\sar	mple\PE-0	I\PE-01.xsf]				-		×	<
Gen	eral settings	Statis	tical sw	itchin	e-suree ca	dculation setti	nes									
Numi Numi	ber of cases	rs	20 C	:	0	All Run All Stop	Con	secutive pr cess settin	ocessing of gs	analysis re	sults		OK Cancel			
	Matrix	size			Run all/st	op all setting.		Analys	is résult pro	ocess		() 30 ()	Close			
_	paramete	ur.	Case r	umbe	r											ŀ
1	parament		1		2	3	4	5	8	7	1	1	9	-	10	ſ
	Run /stop se	etting.	i 🖓 Rug					🔁 Run 🗸	😂 Run 🗸	🕼 Run 🗸	Ru	n 🖂	Run 🖓	~	Run	1
1	Param_A			1.5	2.5	3.5	4.5	_								ł
2				_												ł
3																ł
4																ł
0			-													ł
7																1
8																1
9									-							1
10																1
11																
																Æ

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.



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.

ж	Cons	ecutive-analysis	settings - [C:	\XTAP_en\w	ork\samp	le\PE-01	\PE	-01.xsf]						×	¢
Ger	neral settings	Statistical switch	ine-suree ca	lculation set	tings										
Nun	nber of cases	20 0 20 0	3 F 0 F	All Run All Stop	Gor	isecutive cess set	e pro	ocessine s	of an	alysis	results		OK Cancel		
	Matrix-s	ize	Run all/st	op all setting		Ana	lysi	s result	proces	s'			Close		
=RC[-1]*2									_					
		Case num	ber				_					_		3	
	parameter	1	2	3	4	5		6		7		8	9		Н
	Run /stop sett	ing. 😰 Run 🗸	🗐 Run 🗸	🧐 Run 🗸 i	🛛 Run 🗸	😨 Run	×	😨 Run	~ 0	Run	~ 🖗 R	un 🗸	😔 Rur	1 ~	
1	Param_A	20.	=RC[-1]*2												
2	Param_B	33.	4												H
3	Param_C	22.3	3												
4															
5															
6															
7															
0					_		_					_			-

X.	Cor	secuti	ve-analys	is s	ettings - [(:\XTAP_	en\wa	ork\sam	nple	e\PE-01	PE	-01.xsf]								×
Ger	eral settings	Statis	tical swit	chi	ne-suree c	alculatio	n sett	nes												
Num	ber of cases		20 🗘		0	All Run		10	ions	secutive	pri	ocessing	t of	analysi	s re	sults		ок		
Num	ber of paramete	rs	20	\$	0	All Stop		₽ P	roc	ess set	tine	s					1	Canc	el	
	Matrice	size			Run all/s	top all s	etting.			Ana	lysi	is result	pro	cess				Close	5	
RC[-	1]*2			_									_				_	_	_	
_			Case nu	ımb	er								_			_			_	1
	paramete	ar.	1		2	3		4		5		6		7		8			9	
	Run /stop se	etting.	Run	~	🧐 Run 🗸 🗸	🖓 Run	~ 4	Run	~	😨 Run	>	😨 Run	×	😵 Run	~	🗐 Run	×	1 🖓 F	un	-
1	Param_A		2	0.4	40.	8														
2	Param_B		3	3.4																
3	Param_C		2	2.3																
4																				
5																				
6																				
7																				
0								_			_		_		_				- 11	

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.)

×.	Cor	secuti	ve-anal	ysis s	ettings	- [C:\	XTAP_e	n\wo	ork\san	nple	PE-0		-01.	xsf]					-		C			X
Gen	neral settings	Statis	tical su	vitchi	ne-sure	e cal	culation	setti	ngs															
Num	nber of cases		20 ‡			A	II Run		創	Cons	ecutiv	ve pr	oces	sine	of	inaly	sis r	esul	ts		OK			
Num	nber of paramete	rs	21	:	10	3 A	II Stop		3 F	roce	855 56	etting	ts								Ca	ncel		
	Matric	size			Run al	II/sto	p all se	tting.			An	alysi	is re	sult	proc	ess					Ck	se		
SECO	OND																							
SEC	COND(value 1)		Case	numb	er																			E
SE	COND(value 1) parameter	ettine	Case	numb	er 2		3 Bun	~ 0	4 Bun		5 Bu		0	6 Bun		B	7		8		8	9 Bur		
<u>SE</u>	COND(value 1) perament Run /stop se Param_A	ettine.	Case	numb n 🗸 20.4	er 2 Pun =SECO	ND(3 Pun	~ ©	4 Run	~	5 Prur		01	6 Run	~	© R	7 un i		8 Rur	n 🗸	- 23	9 Rur	n 🔍	
SEC 1 2	COND(value 1) perameter Run /stop se Param_A Param_B	etting.	Case	numb 20.4 33.4	er 2 SRun =SECO	ND(3 Se Run	~ 0	4 Run	~	5 Pur	n 🗸	01	6 Run	~	© R	7 un i	1	8 Rur	n 🗸	- 23	9 Rur		
SE(1 2 3	COND(value1) Paramete Param_A Param_B Param_C	etting.	Case 1 © Ru	n v 20.4 33.4 22.3	er 2 Pun =SECO	ND(3 Se Run	~ 0	4 Run	~	5 Prur	n 🖓	01	6 Run	~	© R	7 un i		8 Rur	n 🗠		9 Rur	n 🔍	
1 2 3 4	COND(value1) param_A Param_B Param_C	etting.	Case 1 2 Ru	numb 20.4 33.4 22.3	2 Pun SECO	ND(3 SRun	~ 0	4) Run	~	5 Pur	n 🖂	0	6 Run		© R	7 un [8 Rur	n 🗸		9 Rur	n 🔍	
1 2 3 4 5	COND(value 1) parameter Run /stop se Param_A Param_B Param_C	etting.	Case	n v 20.4 33.4 22.3	er 2 Run =SECO	ND(3 © Run	~ 0	4 Run	~	5 Pur	n 🖂	01	6 Run		© R	7 un [•		8 Rur	n 🗠		9 Rur		
1 2 3 4 5 6	COND(value 1) Paraman Run /stop se Param_A Param_B Param_C	etting.	Case	n v 20.4 33.4 22.3	2 Prime Run SECO	ND(3 Se Run	~ 0	4 Run	~	5 Pur		0	6 Run	>	2 R	7 un i		8 Rur	n 🗸		9 Rur	1	
1 2 3 4 5 6 7	COND(value)) Param_A Param_A Param_B Param_C	etting.	Case	numb 20.4 33.4 22.3	er 2 PRun =SECO	ND(3 S Run	~ 0	4 Run	× 4	5 Prur	n 🖂	0	6 Run		© R	7 un		8) Rur	n 🗸		9 Rur	n 🔍	

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.

XTAP		×
	Text format calculation result output is enabled. If consecutive-analysis is performed at this time, text files will be output for all cases. Stop processing?	

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

XTAP	×
Predicted total bytes of calculation results file	es (xpf) for all cases: 2.3 MB
	ОК

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.

1/	2	Calculating
	Ga	ncel
L	Ua	ncel

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:

[PostProcessingSection] HZ=50

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.

X.	Co	nsecuti	ve-analys	is s	ettings -	[C:\XTAP_	en\wo	rk\sam	ple\PE-0		-01.xsf]				-			×
Ger	eral settings	Statis	tical swit	chi	ne-suree	calculation	n setti	nes						••	,			
Nun	ber of cases	rs	20 0	:	6	All Run All Stop		∰Co Z Pr	onsecuti ocess s	ve pr etting	ocessin ts	€ of	analysis	s res	ults)K Jancel	
	Matrix	-size			Run all	/stop all se	atting		Ar	nalys	is result	t pro	cess			C	lose	
=RC[-	•1]*2	_			_		_	_	_	_	_	_	_	_	_	_	_	_
	paramet	er	Case ni	ımb	er 2	3		4	5		6	5	7		8		9	
	Run /stop s	etting.	😢 Run	~	🗐 Run	v 😒 Run	~ @	Run	Ru	n v	Run	~	Run	~	🙄 Run	~	🖗 Run	~
1	Param_A		2	0.4	4	0.8												
2	Param_B		3	3.4														
3	Param_C		2	2.3														
4																		
5																		
6																		
7																		

(1) Consecutive processing of analysis results

×.	Co	nsecuti	ive-analys	sis s	ettings - [C:\XTAP_	en\w	ork\sar	np	le\PE-01	\PE	-01.xsf]							3	×
Ger	neral settings	Stati	stical swi	tchi	ne-suree	calculation	n sett	ines												
Nun Nun	ber of cases	ers	20 C	0	0	All Run All Stop		110 121	Con	isecutive Jess set	e pr	ocessine Is	; of	analysi	s re	sults		OK Cance	51	
	Matrio	(~size			Run all/	stop all se	etting.			Ana	lys	is result	pro	cess			0	Close		
RC[·	1]*2						_						_				_		_	
		parameter Case nu				_	_				_		_		_	_	_	_	_	T
	paramet	parameter			2	3		4		5		6		7		8			9	ľ
	Run /stop s	etting.	🙄 Run	~	😨 Run 🕔	🖉 🖓 Run	~ 4	Run	~	😌 Run	~	😨 Run	×	😨 Run	~	😂 Run	~	2 R	un 🗸	
1	Param_A		1	20.4	40	.8														1
2	Param_B		1	33.4																1
3	Param_C		1	22.3																
4																				1
5						-														1
6						-														1
7																-				1
8												-							0.0	1
4			-																	

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.

🔠 Open all waveform data.		×
(는 -> > 수 🦲 « Windows (C:) > XTAP_en > work > sample > PE-01 🧹 👌	Search PE-01	Q
Organize - New folder	BEE -	• • •
Cucktaccess Desktop # Downloads # Downloads # Downloads # Downloads # Documents # GoogleDrive # This PC # Creative Cloud Fil Dropbox CneDrive This PC		Date r 2/28/2
CD Drive (Ht) HD- 🗸 🧃		>
File game 🗌 🗸 🗸	Data analysis Files (* <u>O</u> pen	all.csv) (*al v Cancel

Statistical processing is executed on each waveform file.

7 / 9 Processing		XTAP
	9 Processing	
		-
Cancel	Cancel	

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.

📕 🏹 🔜 🗢 PE-01				- 0	×	
File Home Share View					~ 📀	
Pin to Quick access Copy Paste Clipboard	Move Copy to * Organize	New folder New	Properties History Open	Select all Select none Invert selection Select	on	
← → → ↑ 📑 > This PC > Windows	(C:) > XTAP_en > work > samp	le > PE-01	~ (Search PE	Q	
✓ ★ Quick access	Ecsy Files of AX.csy	output b	Date modified 2/25/2019 10:18 y statistical p	Type AM Adobe Acro	ig calc	ulations
Pictures 🖈 🔂 PE-01_se_M	AXABS.csv		2/28/2019 9:07 P	M Microsoft E	xcel	
💪 GoogleDrive 🖈 📫 PE-01_se_M	IN.csv		2/28/2019 9:07 P	M Microsoft E	ixcel	
This PC 🖈 🖾 xcoremsg.ou	ut		2728/2019 2:41 P	M OUT File		
Creative Cloud Fil PE-01_se.xtf	.pp		2/28/2019 1:33 P 2/28/2019 2:41 P	PM PP File PM PP File		
> 😻 Dropbox 🖉 PE-01_se_1.3 🖉 PE-01_se_2.3	tf.pp tf.pp		2/28/2019 2:50 P 2/28/2019 2:50 P	PM PP File PM PP File	v	
> ConeDrive v c 47 items				B		

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.

6	ਜ • • ∂	- 8 -		PE-01_se_MA	K.csv - Excel	F	likid			-		×
	ile Home	Insert Pa	ge Layout Fo	ormulas Data	a Review V	/iew Develope	r Help A	CROBAT TE	am Q	Tell me	, Я s	hare
Pa	pboard	游ゴシック B I U 田 • 〇 〇 •	- 11 - A^ A - A - A - A - A - A - A - A - A - A - A - A - A - A - A - A - A	• = = = = • € • • Align	≡ ^{ab} ≡ ⊡ • ≫ • ment G	General • \$ • % * \$ \$ Number	Form	ditional Form nat as Table * Styles * Styles	atting -	Cells	Editing	~
114	1 ~	· : . ×	$\checkmark f_X$									~
	A	В	С	D	E	F	G	н	1		J	- F 🖻
1	File Name	PE-01_se										
2	Number o	f cases9										
3	Analysis-	MAX										
4												
5		NodeVolta	age		BranchCu	rrent						
6	Case Nurr	Node_u	Node_v	Node_w	Lu	Lv						
7	1	50	50	50	6.84E-31	5.92E-31						
8	2	100	100	100	1.37E-30	1.18E-30						
9	3	150	150	150	-4.07E-16	-4.07E-16						
10												
11												
12												
13												
14												
	4 4	PE-01_se_	MAX	+		1						•
Rea	dy 🛅						E	9 (00) E	9	1	+	100%

(2) Process settings

6	Con	secutio	ve-analys	is s	ettings - [C:	XTAP_	en\w	ork\san	mpl	le\PE-01	I\PE	E-01	.xsf]					-				×
Gen	eral settings	Statis	tical swit	chi	ne-suree ca	lculation	sett	tines														
Num Num	ber of cases	\$	20 0	•	Ø P 3 P	All Run All Stop		(1) (1)	Con Proc	isecutivi cess set	e pr	oce	ssine	of	analys	is re	sults	A COLORADO		OK Cani	cel	
	Matrix-	size			Run all/sto	op all se	tting			Ana	lys	is re	esult	pro	cess					Clos	e	
-106	1]*2						_									_	_			_	_	_
_		parameter Case nu					_		_	_	_	_	_	_	_	_	_	_	_	_	_	
	parameter		1		2	3		4		5			6		7	5		8			9	
	Run /stop set	tting.	🙄 Run	~	😨 Run 🖂	🧼 Run	~	🖉 Run	~	😨 Run	~	0	Run	Y	😨 Ru	n ~	0	Run	~	01	Run	~
1	Param_A		2	0.4	40.8																	
2	Param_B		3	3.4																		
3	Param_C		2	2.3																		
4																						
5																						
6																						
7																						

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.

	Analysis-1	ype	32-bit DLL file name	64-bit DLL file name		Function N	lame
1	MAX	~	PostProcessingLibrary32.dll	 PostProcessingLibrary64.dll	-	_max	
2	MIN	~	PostProcessingLibrary32.dll	 PostProcessingLibrary64.dll	-	_min	-
3	MAXABS	~	PostProcessingLibrary32.dll	 PostProcessingLibrary64.dll	-	_maxabs	
4	AVE	~	PostProcessingLibrary32.dll	 PostProcessingLibrary64.dll	-	_ave	
5		~			-		-
6		~			-		-
7		~					-
8		\sim			-		-
9		~			-		
0		~					-

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.

	Analysis-ty	pe	32-bit DLL file name		64-bit DLL file name	Function N	lame
1	MAX	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 _max	1
2	MIN	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 _min	,
3	MAXABS	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 _maxabs	`
4	AVE	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 _ave	
5	INTEGRAL	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 _integral	,
6	RMS	~	PostProcessingLibrary32.dll		PostProcessingLibrary64.dll	 rms	`
7		~					`
8		~					`
9		~		+++			`
0		~					1

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.



8.1 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 tines	Consecutives an Statistical switching-surge calculat	alysis settings (CA 3 is	ample\SSW-02\SS 4	5	6	7
Search breaker	Calculation case-count	100 - Manual setting Seeds 0 1 Random number settings	Fixed time T1 0 Power-source frequency 1 ON command settings	Hz Analyze Analyzis	😧 Help Help	Cancel
Breaker n 1 2 3 4 5 6 7 8 9 10 11 12 13	ame A Fixed time T2	derd deviation σ (sec)	Maximum deviation σ [sec]			

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.

¥.	Consecutive-analy	sis settings - [C:\XTAP_en\work\s	ample\SSW-02\SSW-02.xsf]		-	o ×
General settings	Statistical switching-surge calculation	settings				
Search break	Calculation case-count	Auto Setuo Inval setting Sends 0 C Random number settings	Fixed time TI 0 sec Power-source frequency 1 Hz ON command settings	Analyze Analyzis	😢 Help Help	Close
Breaker Breaker 1 SwCB1 SwCB1 2 SwCB2 SwCB3 4 SwCB3 SwCB3 5 S SwCB3 6 7 SwCB3 9 10 11 11 12 13	name	Standard deviation σ [sec]	Maximum deviation σ [sec]			

(3) Random number setting Specify the method for calculating random numbers. When selecting the manual method, enter a seed value.

3	Consecutive-ana	alysis settings - [C:\XTAP_en\work\s	ample\SSW-02\SSW-02.xsf]		-	
General settings	Statistical switching-surge calculation	on settines				
Search breaker	Calculation case-count	Auto Setus 100 - Manual setting Seeds 0 0	Fixed time T1 0 se Power-source frequency 1 Hz	C E Analyze	😧 Help	Cancel
Breaker n 1 SwCB1 2 SwCB2 3 SwCB3 4 5 6 7 7 8 9 10 11 12 13	Fixed time T2 [sec]	Standard deviation of [sec]	Maximum deviation σ [sec]			

(4) ON command settings

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

16	Consecutive-anal	ysis settings - [C:\XTAP_en\work\s	ample\SSW-02\SSW-02.xsf]			-		×
General settings	Statistical switching-surge calculation	n settings						
Search breaker	Calculation case-count	100 - Auto Setus Manual setting Seeds 0 1	Fixed time T1 0 Power-source frequency	sec 1 Hz	📔 Analyze	🥑 Help	Canc	æl
Search breaker	Calculation case-count	Random number settings	ON command settings		Analysis	Help	Clos	e
Breaker Breaker <t< td=""><td>Ame A Food time 12 [sec]</td><td>Standard deviation of [sec]</td><td>Maximum deviation of [sec]</td><td></td><td></td><td></td><td></td><td></td></t<>	Ame A Food time 12 [sec]	Standard deviation of [sec]	Maximum deviation of [sec]					

(5) Breaker parameter input

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

3	Consecutive-analysis sett	ings - [C:\XTAP_en\work\sa	ample\SSW-02\SSW-02.xsf]		-	
General settings St	Statistical switching-surge calculation setting	ta,				
Search breaker C	Calculation case-count 100 -	Auto Setus Manual setting Seeds 0 0	Fixed time T1 0 sec Power-source frequency 1 Hz	PE Analyze	😧 Help	Cancel
Search breaker	Calculation case-count	Random number settings	ON command settings	Analysis	Help	Close
2 SwCB2 3 SwCB3 4 5 7 8 9 9 10 11 12						

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 σmax: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.

XTAP		×
?	Analyze all wav	eform data?
	<u>Y</u> es	No

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.

Too	ls
	Circuit diagram canvas size (X)
	Component canvas size
N-	XPLT (graphing program)
πc	XTLC (line-constant calculation program)
M	XFIT (system identification program)

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.

📓 XPLT	-	×
Files Tools Charts Frequency characteristics calculation result setting Help		
🞯 📘 Q、 ヴ 蒜 朝 課 迷 🗙 藤 🛛 💽 Low-resolution mode 个 🦆		
Add File Delete File No. Director 1 PE-01 Image: Control Stands Calculation result executed before starting XPLT		
Drawine Modified eraph Grid On Draw eraph Graph properties		

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 lowand 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.

📷 Draw graph	×
Select the modification method.	
 Integral 	
 Fourier Transform 	
Phase Angle On	
O Fourier-series expansion (specified interval)	
Base Frequency 1 Hz	
Phase Angle On	
◯ Constant (a*f(t)+b)	
0 0	
⊖ XYPlot	
	_
Drawing Cancel	

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*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	
O Fourier Transform	
Phase Angle On	
O Fourier-series expansion (specified interval)	
Base Frequency 1 Hz	
Phase Angle On	
Onstant (a*f(t)+b)	
5 3	
⊖ XYPlot	
Drawing Cancel	
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- Φ 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.

Specify the XY plot.		×
No. File Name #1 PQ-02-A		
<	>	
Node Voltages Branch Currents Branch Voltages Control Signals		
u(pri <u>a)</u> u(pri <u>b)</u> u(pri_c)	X-axis >	
	Y-axis >	
		OK Cancel

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.

No. File Name			
#1 PQ-02-A			
<	>		
lode Voltages Branch Currents Branch Voltages Control Signals			
(INT1)		X-axia >	(SwCB_a) #1
		< X-axis	
		Y-axis >	s(INT1) #1
		1 Taxis	

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



(4) Panel button operations



(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 tabseparated 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.)

Imp636869881613107670 - Notepad	- 0	×
Ele Edit Format View Help		
Imp68866881131070℃-Hexepad Exe [sf spmst Yee bite Exe [sf spmst Yee bite Exe [s] Branch Currents [A] i(Lu) #1 Time [s] Branch Currents [A] i(Lu) #1 2.000000000e=002 -2.259551858e+000 2.000200000e=002 -2.245041878+000 2.000200000e=002 -2.245041878+000 2.000600000e=002 -2.28557828e+000 2.000600000e=002 -2.28557828e+000 2.000600000e=002 -2.281782610e+000 2.000600000e=002 -2.281782610e+000 2.000700000e=002 -2.281782610e+000 2.000700000e=002 -2.240727228+000 2.000800000e=002 -2.2440727228+000 2.0011000000=002 -2.248578088e+000 2.0011000000=002 -2.248578088e+000 2.001100000=002 -2.2485787878700 2.001100000=002 -2.2485787878700 2.001100000=002 -2.2485787878700 2.0011000000=002 -2.2485787878700 2.0011000000=002 -2.24857878787000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.24857878787000 2.0011000000=002 -2.24857878787000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.248578787877000 2.0011000000=002 -2.2485787877777877777777777777777777777777	 Branch Currents [A] i(Lv) f1 Branch Currents -2.399419247e+000 4.8558370905e+000 -2.414597010e+000 4.85583847e+000 -2.414597010e+000 4.85583847e+000 -2.42174511e+000 4.8569372470e+000 -2.435843006e+000 4.8684376861e+000 -2.435844370e+000 4.868437023e+000 -2.435844370e+000 4.867409582e+000 -2.435844370e+000 4.867409582e+000 -2.4485182658+000 4.858387228+000 -2.4481152658+000 4.858387228+000 -2.4481152658+000 4.858387228+000 -2.458706061a+000 4.858387228+000 -2.458706061a+000 4.858387228+000 -2.458706061a+000 4.9592851488+000 -2.4587086638+000 4.7064758398+000 -2.4587286638=000 4.7064758398+000 -2.4587286631299+000 4.7054758398+000	×
2.40174000000-002 -2.24227258+000 2.001700000-002 -2.24257258+000 2.001700000-002 -2.245579088+000 2.001700000-002 -2.245579088+000 2.001700000-002 -2.25555258+000 2.001700000-002 -2.25555258+000 2.001700000-002 -2.2555227488+000 2.001700000-002 -2.2585224+000 2.001700000-002 -2.25823724+000 2.001700000-002 -2.2587324+000 2.001700000-002 -2.258735449+000	-2, 4400 152/054 (000) -2, 45118174+000 -2, 453705061 e+000 -2, 4553705061 e+000 -2, 4552747422e+000 -2, 455274742e+000 -2, 4553705059e+000 -2, 4553705059e+000 -2, 4553705059e+000 -2, 4553705059e+000 -2, 455387059e+000 -2, 455385019e+000 -2, 455385019e+000 -2, 455387258e+000 -2, 47537258e+000 -2, 475375886 -2, 4753725886 -2, 475375886 -2, 4753725886 -2, 475375886 -2, 4753758686 -2, 47557586	
.002100000e-002 -2.270223386#+000 2.002100000e-002 -2.27023386#+000 2.002200000e-002 -2.275870055e+000 2.002300000e-002 -2.278389355e+000 2.002800000e-002 -2.28110513e+000 2.002800000e-002 -2.28119788e+000 2.002800000e-002 -2.288319788e+000 2.00270000e-002 -2.288319788e+000 2.002800000e-002 -2.288319788e+000	-2.474248435e+000 4.744271803e+000 -2.479007102e+000 4.751955168e+000 -2.48152240e+000 4.751955168e+000 -2.48152240e+000 4.75219323e+000 -2.4805851e+000 4.782426186e+000 -2.48956556e+000 4.772876244e+000 -2.49056556e+000 4.77803517e+000 -2.490560589e+000 4.783035252e+000	
¢		>

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.

~	X-axis
	X-axis Min 0.02
	Minimum value 🔽 True
	X-axis Max 0.04
	Maximum value 🔽 True
~	Y-axis
	Y-axis Min -6
	Minimum value
	Y-axis Max 6
~	V9-avia
*	V2-avia Min 0
	Vanuia May 0
	Maximum value
х-	axis Min
X-	axis Min acity the minimum value of the X-avis
X- Spe	axis Min scify the minimum value of the X-axis. ad the file and display it again.
X- Spe Re	axis Min ecify the minimum value of the X-axis. ad the file and display it again.
X- Spi Re	axis Min scify the minimum value of the X-axis. ad the file and display it again.
X- Spr Re	axis Min ecify the minimum value of the X-axis. ad the file and display it again. w graph Graph properties
K- Spe Re	axis Min scify the minimum value of the X-axis. ad the file and display it again. w graph Graph properties

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 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.

Numb	ber of graph	s displayed	on 1 pa	age: 5	~
Margin	s(millimeter	s)			
100	ft- 20] D:-W	+ D0		
Le	20	rugn	. 20		
To	p: 20] Rign] Bottor	n: 20		

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

Тоо	s									
4	Select									
Q	Enlarge waveform									
17	Reset									
44	Time Lag On									
20	Display waveform values 1									
	Display waveform values 2									
梁	Marker Off									
X	Change entire graph size									
⊠ DFF	Reset all checkboxes									
ø	Redraw graph									
	Save as CSV Files									
*	Full-resolution mode									
1	Move up	Ctrl+Up								
Ψ.	Move down	Ctrl+Down								

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.

		Signal type	Analysis result file No.
		Part name	
Imp636869887630113412 - Notepad			×
Elie fair Farmat Yiew Help Case 1 (Some omitted) Time [s] Branch (0 2.000000000e-002 2.00000000e-002 2.00060000e-002 2.00100000e-002 2.00100000e-002 2.00100000e-002 2.00100000e-002 2.001200000e-002 2.00180000e-002 2.00180000e-002 2.00180000e-002 2.00220000e-002 2.00220000e-002 2.00220000e-002 2.00220000e-002 2.00280000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00320000e-002 2.00340000e-002 2.004400000e-002 2.00480000e-002 2.00540000e-002 2.00540000e-002 2.005600000e-002 2.00560000e-002	Currents [A] i(Lu) #1 1 -2.259551658e+000 -2.230561225e+000 -2.230561225e+000 -2.230561225e+000 -2.223027410e+000 -2.234555495e+000 -2.251074591e+000 -2.251074591e+000 -2.256559235e+000 -2.267495964e+000 -2.267495964e+000 -2.278383935e+000 -2.283839385e+000 -2.30048375e+000 -2.30048376e+000 -2.30543696e+000 -2.30543696e+000 -2.298570474e+000 -2.285970474e+000 -2.285970474e+000 -2.288951e+000 -2.2838951e+000 -2.288951e+000 -2.2838951e+000 -2.2838951e+000 -2.283199414e+000 -2.263199414e+000 -2.263199414e+000 -2.263199414e+000 -2.258672440e+000	Branch Currents [A] -2.393419247e+000 -2.414597010e+000 -2.429744437e+000 -2.438402613e+000 -2.443514066e+000 -2.443514066e+000 -2.44587042e+000 -2.458768663e+000 -2.468917223e+000 -2.468917236e+000 -2.468917236e+000 -2.484056851e+000 -2.49406608861e+000 -2.49406608651e+000 -2.49405608651e+000 -2.5097545563e+000 -2.50934651e+000 -2.50934651e+000 -2.497543835e+000 -2.497543835e+000 -2.497543835e+000 -2.497543835e+000 -2.49762671e+000 -2.49266880e+000 -2.462764912e+000 -2.462764912e+000 -2.457838703e+000	i(Lv) #1 Branch Currents [▲ 4.658970905e+000 4.659638947e+000 4.6690305661e+000 4.667430023e+000 4.678069562e+000 4.68981255e+000 4.720416277e+000 4.720416277e+000 4.720416277e+000 4.751955166e+000 4.772876294e+000 4.78305526e+000 4.78305526e+000 4.804101515e+000 4.814468347e+000 4.814468347e+000 4.8124272851e+000 4.77314412e+000 4.773457610e+000 4.773438412e+000 4.7543841

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:

🞯 🔖 🔍 🤊 | 👬 🗰 🗮 💥 🎢 🕼 💽 💽 Low-resolution mode | 🕇 🤳

Toolbar in full-resolution mode:

💕 🔖 🔍 🤊 | 👬 🏥 🧱 🐹 🎉 🌠 🗐 💽 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

Charts Frequency characteristics cale	
Sav	1
Save to Clipboard 🕨	
	EMF

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>

PKr	Setup
Numerical data output setting Graph settings 1	Numerical data output settings Program used for displaying numerical data File Setup notepadexe
	Case name display setting when outputting numerical data
	✓ Display case name
< >>	OK Carrel
	OK Cancel

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>

PLT	Setup	×
Numerical data output setting Graph settings 1	Graph settings 1	
	Low-resolution display setting	
	Number of data points set as omission threshold: 10000 When low-resolution mode is specified and the number of points exceeds the number of data points specified here, the	
	eraph is displayed with data omitted with the number of points set here as the upper limit.	
	1000 total data points for analysis result -> Display as is (1000 points in total)	
	6000 total data points for analysis result -> Display 1 point for every 3 points of original data (2000 points in total)	
	Phase display setting of frequency characteristic calculation result	
	✔ Display phase	
< >>		
	OK Cance	:I .:i

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



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 macropart 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.

00 Constant				-	×
Parameters	Record	Display	Help		
					^
C:	FREQ		×		
		(Sp	ecifiable by numbers only)		

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].

Inductor					-	×
Parameters	Initial Value	Record	Display	Help		
						^
L Lval	mH V	9				

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 partcategory 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.

Category:	99 User's Components	~	Ok	
Subcategory:	<none></none>	~	Cancel	
<u>N</u> ame:	[~		
Co <u>m</u> ponent type:		~		
Seets:		~		

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.

Enter a caption name.	
SIN_Wave	
	Ok

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

TAP - [NewCircuit1]																			_			×
File Edit Circuit Analysis	View	Tools	Window	w Help																	- 1	5 X
	- 1.08	DA LAR	Month	- 5	1 3	N				-	-		 	1				1.0			100	
			200%		+ -	+ -+	1.1	- 34	A		10.0		1			<i>r</i> .	N N	~	~	P.C	192	
																						^
l 🖁																						
00 Ground, Terminal																						C.,
01 RLC																						
02 Voltage Source, Current Source																						
03 Switch																						
04 Probe																						
05 Transmission and Distribution Line																						
06 Transformer																						
07 Generator																						
08 DC power transmission, Frequen							L .															
09 PV							SI	N	Wa	ve1												
10 Electric Motor																						
20 Auxiliary Component for Power																						
40 Arithmetic Function Block									T -													
41 Four Arithmetic Operations Block									_													
42 Control Function Block		_						1	-													
43 Dynamic Control Block									_													
44 Time Delay Block									_													
45 Logic Function Block									N													
46 Special Function Block									-													
47 Signal Source									Т					11						-		
48 alpha-beta, d-q, pu Conversion																						
49 DLL																						
80 Lightning Overvoltage Studies of _																						
81 Lightning Overvoltage Studies of _																						
82 Switching Transient Overvoltage									_				 1				_	1	-			
99 User's Components																						
SIN_Wave													 1						-			
						-			_						_							
											-		 -		-				-	_		
						++		-			-		 	-	-	-	++-		-	_		-
						1			-		-		 -	++	-					-		
						++		_					 	++	-			\rightarrow				
						++			-		-		 -	++	-			1				- 4
	<	and the second										100										>
Status																			9.166	66	216.66	67 .

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.

<u>C</u> ategory:	99 User's Components	\sim	Ok	
Subcategory:	user 1	~	Cancel	
<u>N</u> ame:	SIN_Wave2	~		
Co <u>m</u> ponent type:		~		
Seeto:		0		



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'.

Open component				×
<u>Category:</u>	99 User's Components	~	Ok	
Subcategory:	<none></none>	~	Cancel	
<u>N</u> ame:	SIN_Wave	~	Delete	
Component type:	Circuit	~	Properties	
See <u>t</u> s:	0	~	Display component	

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'.

<u>F</u> ile	1	
	Create circuit diagram	(<u>N</u>) Ctrl+N
2	<u>O</u> pen circuit diagram	Ctrl+O
	Create component	Ctrl+Shift+N
2	Open component (<u>O</u>)	Ctrl+Shift+O
	<u>R</u> ename component	
	Create category	Ctrl+G
	<u>C</u> lose	
	<u>S</u> ave	Ctrl+S
	S <u>a</u> ve as	
	<u>E</u> xport	•
	<u>I</u> mport	•
	Page Set <u>u</u> p	
a	<u>P</u> rint	Ctrl+P
4	Print Preview	
	E <u>x</u> it XTAP	

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

Open component				×
	-	1		
Category:	99 User's Components	~	Ok	
Subcategory:	<none></none>	~	Cancel	
<u>N</u> ame:	SIN_Wave	~	Delete	
Component type:	Circuit	~	<u>P</u> roperties	
Seets:	0	~	Display component	

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	Ok
Pen Color:	Cancel
Coordinate information	, Arrow
Begin: -42 -13	20 Add arrow
End: 32 -12	20

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.

'en information Pen Width:	Fill information.	Ok
Pen Color:	Fill Colors:	Cancel
Coordinate informat	ion	
Location 1: 🗙 [-7 Y: -99	
Location2: X: [-26 Y: -61	
Location & X	30 Y: -66	

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.

RectangleDialog		×
Pen information Pen Width: 1 Pen Color:	Fill information. Fill: Fill Colors:	Ok Cancel
Coordinate information Location: X -24 Width: 48	Y: -36 Height: 50	
Shapes Shape Rectangle v		

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: 1	Fill information.		Ok
Pen Color:	Fill Colors:		Cancel
Coordinate information			
Location: X: -30	Y: 46		
Width: 64	Height: 49		
Customize ellipsoid	Start angle: Swe	ер	
	Start angle.		

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			
Font:	Arial	Setup	OK
Font Size:	9		Cancel
Font Color:			Oditicei
Coordinate i Center Loo	nformation X cation: 1	Y 120	
Text Direction:	on Horizontal	~	

Current font

Click 'Set' to set the font.

ont		Font style:		Size:	_	
Arial		Regular		9	1	OK
Arial	^	Regular	^	3 10	^	Cancel
Arial Rounded MT		Narrow Bold		11		
Arial Unicode MS		Narrow Bold Itali		14		
Atlanta	~	Bold	~	16	~	
Effects Strikeout Underline Color:		Sample AaBbY	'yZz	-		
Black v	1	Script				
-	1	Western			\sim	

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)

Fig	ure (<u>Z</u>)
9	Check terminal consistency.
۲	<u>S</u> elect
0	Terminal (<u>P</u>)
$^{\circ}$	Input terminal
$^{\circ}$	Output terminal
0	Multiple-signal input terminal
$^{\circ}$	Multiple-signal output terminal
$^{\circ}$	Line
\odot	<u>T</u> riangle
\bigcirc	<u>R</u> ectangle
\bigcirc	<u>E</u> llipse
\bigcirc	Te <u>x</u> t

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.

Terminatoratog	-	ц _
Terminal Type: Terminal	Terminal Direction	Ok
Color:	Right(Defa ↓	Cancel
Coordinate informatio	X Y	
Coordinate informatio Center Location: [· Name	^{on} X Y -108 -48	
Coordinate informatio Center Location: [· Name [3	²ⁿ X Y -108 -48	
Coordinate informatic Center Location: [Name 3 Terminal number	nX Y -108 -48	

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'.

≥ategory:	99 User's Components	~	Ok	
jubcategory:	<none></none>	~	Cancel	
lame:	SIN_Wave	~	Delete	
o <u>m</u> ponent type:	Circuit		Properties	
Seets:	0	14	Display component	

3. The part's properties editing screen appears.

age	Setup					
	Tab Title	File Name		10		^
•	Parameter	XmfParamsh	tml	***		
	Recording	RecordOptionsSub html		***		
	View	DisplayOptio	nsCaption.html			
	Acquire power-flow calculatio	PowerFlowIn	fohtml	-		
	Encryption	Sub_EncPass	word	-		
	Caption		Display referen	ce labe	GDP_NONE	_
•	SubParams		Label			
	Caption		Display referen	ce labe	Falsa	
	captiondisp		Editing possible	3	False	
	XFIXID_POWERFLOWINFO		Script		T diac	
	SubOutout		Input script			
			Output script	_		- 1
			Output		True	_
			Output sequence	e	2	
		1	/ Units			
			Unit conversion	n label		~
		111	ahel			

4. 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'.

age	Setup			
	Tab Title	File Name		^
•	Parameter	XmfParamshtml		
	Recording	RecordOptionsSub.html		
	Display	DisplayOptionsCaption.html		
	Acquire power-flow calculatio	PowerFlowInfohtml		
	Encryption	Sub_EncPassword		~
	Caption	Display reference Editing possible	ce labe False	
	SubExpression	Direction	CDP_NONE	
	Caption	Editing possible	False	
	captiondisp	New	False	
	XFIXID_POWERFLOWINFO	⊿ Script Input script		-
	SubOutput	Output script		
		Text output		
		Output	True	- 1
		⊿ Unit	e Z	- 1
		Unit conversion	label	~
		Value		

5. 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'.

'age	Setup	53 N			
	Tab Litle				- î
•	Parameter	XmtParamshtml			-1
	Recording	RecordOptionsSub.ht	ml		-1
	Display	DisplayOptionsCaptio	on.html		_
	Acquire power-flow calculatio	PowerFlowInfohtml			_
	Encryption	Sub_EncPassword			~
•	SubExpression	Direc	tion av reference labe	CDP_NONE	-1
	SubParams		1	a = 5 * b	~
•	SubExpression	Direc	tion	CDP_NONE	-1
		1 1 1 2 2 1 1	ay reference labe		_
	Caption	Editir	e possible	False	
	Caption captiondisp	Editir New	g possible	False False	
	Caption captiondisp XFIXID_POWERFLOWINFO	Editir New	e possible	False False	ł
	Caption captiondisp XFIXID_POWERFLOWINFO SubOutput	Editin New Scrig Input	g possible it script t script	False False	
•	Caption captiondisp XFDID_POWERFLOWINFO SubOutput	Dispir Editin New Scrig Input Outpu d Text	g possible script it script output	False False	
	Caption captiondisp XFDID_POWERFLOWINFO SubOutput	A Scrig Input Outpu Outpu	g possible script tt script output t	False False	
*	Caption captiondisp XFDID_POWERFLOWINFO SubOutput		g possible script tt script output tt sequence	False False True 3	
•	Caption captiondisp XFDID_POWERFLOWINFO SubOutput	Corpus	g possible script script output it sequence conversion label	False False True 3	

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

encomponent				
Category:	99 User's Components	~	Ok	
Subcategory:	<none></none>	~	Cancel	
<u>N</u> ame:	SIN_Wave	~	Delete	
Component type:	Circuit	\sim	Properties	
Seets:	0	\sim	Display component	

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

			SIN_Wave	-		×
Parameter	Recording	Display	Acquire power-flow calculation information.	Help		
Paramete	ers					^
b=100					^	
					~	
<				>		
Math Equ	uations					
a = 5 *	۶ b				~	
						~
					~	
			ОК	Ca	ancel	

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 usercreated 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.

nter category	×
Category Name:99 User's Components	Ok
user2	Cancel

60 Lightning Overvoltage Studies of	
62 Switching Transient Overvoltage	
99 User's Components	
🚞 user2	
	<

(2) Deleting categories/macro parts

To delete a created category, right-click on its name and select 'Delete' from the rightclick 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.

11.1 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.

Tools

	Circuit diagram canvas size (X)
	Component canvas size
PLT	XPLT (graphing program)
πc	XTLC (line-constant calculation program)
¥	XFIT (system identification program)

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

É D 😂 🖬 🐰 ካ 🗞 🔊 • ♥ - 🛃 Q, 🎯 150% - • 🔖 🕂 斗 斗 斗 ♪ ♪ ♪ A 1⊅ 💷 | # Ø 😭 🕨 🖬 🕪 🗵 🔍 🛠 🔩 🚮 🛃

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.

Tools

	Circuit diagram canvas size (X)
	Component canvas size
PUT	XPLT (graphing program)
τc	XTLC (line-constant calculation program)
ΡT	XFIT (system identification program)

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

I D B B K = 8 9 • C • B C 0 150% • R + 4 4 ... > > > A D III # 0 B > D II > E E V M K C

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
1.1Overview
1.2Main window
1.3Menu bar and toolbar146
2 Creating New Line-Placement Data
2.1Selecting the line type
2.2Entering line-placement data
2.3Line-constant calculation155
2.4Displaying line-constant calculation results
3 Referencing Existing Line-Placement Data
3.1Referencing line placement data162
3.2Revising line-type information162
3.3Calculating line constants and displaying results
4 Using Line Model Data Created in XTLC
4.1How to specify the data for the distributed-parameter line model
4.2How to specify the data for matching resistance
5 Registering Information in Line/Cable Databases
5.1Databases
5.2Overhead transmission lines
5.3Overhead distribution lines
5.4Cables
5.5Line/cable database functions

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.

Menu bar	XTLC(Line-constant calculation program):(Untitled) -	J X
Toolbar	Files View Database Help Image: Image	
	Line placement data area	
Status bar]	
	[ins]: insert line. [Alt] - [ins]: Select row (copy) and insert. [Alt] + []/[]: Move row up/down. [Del]: Delete row (multiple selections allow	ed) .

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

1.3 Menu bar and toolbar

The XTLC menu bar is described below.

🔣 х	TLC(Line	-constant calcu	ulation prog
File	s View	Database	Help
	New	Ctrl+N	
2	Open	Ctrl+O	
	Close		
H	Save	Ctrl+S	
	Save As		
	Exit		
_			-

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.

TL 📷	.C(Line-c	onstant calcu	lation pro
Files	View	Database	Help
	3	3	
New	(Ctrl+N)		

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.

🔣 XTL	C(Line-c	onstant calcu	lation prog
Files	View	Database	Help
i 🗋 🔽	3	3	
	Open (Ctrl+O)	

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								
File	25	View	Database	Help				
1	2	; 🔒	3					
		Sav	re (Ctrl+S)	Conductor NO				

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.

🔣 XTL	C(Lin	ie-c	onstant calc	ulation prog			
Files View Database Help							
1 🗋 🖻	~	Т	oolbar				
	_			_			

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

XTLC(Line-constant calculation program):(Untitled)

Files	View	Database	Help
i 🗋 🖆		3	Instruction Manual

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.

Line type selection
Overhead line Transmission line Distribution line
Underground cable O Conduit-line distribution cable
Select Cancel

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 lineplacement 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 lineplacement 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 lineplacement data. A distribution line does not involve poles or conductor-interval settings.



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

1	ZTLC(Line-constant calculation program):C:¥XTAP_CRIEPI¥work¥Examples¥STL-02¥vertically.lin – — X									
		uase neip								
	Туре	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 ~	1
	Delete 🗸	0	IACSR 150	14.5	79.5	0	1		0 ~	1
	Do not delete 🗸	1	TACSR 410	-11.2	72.0	0	6	0.5	0 ~	1
	Do not delete \smallsetminus	2	TACSR 410	-11.6	57.5	0	6	0.5	0 ~	
	Do not delete \smallsetminus	3	TACSR 410	-12.0	43.0	0	6	0.5	0 ~	
	Do not delete \lor	4	TACSR 410	11.2	72.0	0	6	0.5	0 ~	
	Do not delete \smallsetminus	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 ~	~
				× = 0 0	0 4					^
				2	5					
				å	6 •••					
<									د	ļ ~

• Underground cable

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



	C(Line-constant cal	culation program):(Unti	tied)				-		>
iles	🖸 🛃 🛛 😚	Help							
Γ	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	Tripl grou	lex IP
1	1	CAZV220kV 600	-0.81	1	54	14	1		0
T	2	CAZV220kV 600	0	1	54	14	1		0
	3	CAZV220kV 600	0.81	1	54	14	1		0
							1		0
				×ŧ0					
					å				
			G	Ψ	W				
			0	Ϊ	•				
			•		•				
			G)	Ŭ				

To check conductor placement, click the ' \downarrow 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.

Туре	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 ~		Group 1
Do not delete 🗸	2	TACSR 410	-11.6	57.5	0	6	0.5	1 ~	٢.	0.000
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 🗸	LE	Group 2
Do not delete 🗸	5	TACSR 410	11.6	57.5	0	6	0.5	2 ~	٢.	Croup 2
Do not delete 🗸	6	TACSR 410	12.0	43.0	0	6	0.5	2 ~		

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

	Туре	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 pa	_							0 ~
-	Click								

The row is then selected.

	Туре	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 \lor	1	TACSR 410						0 ~
	Do 🗤 delete 🗸								. 0 ~
	Selecte	ed							

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.

First ro	OW _{pr 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
elete ~	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 ~

All the rows in that range will then be selected.

Туре	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 no delete $ \sim $	4	TACSR 410	11.2	72.0	0		0.5	0 ~
Do not a 🛛 te 🗸	5	TACSR 410	11.6	57.5	0		0.5	0 ~
Do not dele	6	TACSR 410	12.0	43.0	0	6	0.5	0 ~

Multiple selection (separate rows)

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

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
0	IACSR 150	-14.5	79.5	0	1		0 ~
0	IACSR 150	14.5	79.5	0	1		0 ~
1	TACSR 410	-11.2	72.0	0	6	0.5	0 ~
2	TACSR 410	-11.6	57.5	0	6	0.5	0 ~
3	TACSR 410	-12.0	43.0	0	6	0.5	0 ~
4	TACSR 410	11.2	72.0	0	6	0.5	0 ~
5	TACSR 410	11.6	57.5	0	6	0.5	0 ~
6	TACSR 410	12.0	43.0	0	6	0.5	0 ~
	Conductor no. 0 1 2 3 4 5 6	Conductor no. NO Line type NAME 0 IACSR 150 0 IACSR 150 1 TACSR 410 2 TACSR 410 3 TACSR 410 4 TACSR 410 5 TACSR 410 6 TACSR 410	Conductor no. NO Line type NAME Horizontal position X (m) 0 IACSR 150 -14.5 0 IACSR 150 14.5 1 TACSR 410 -11.2 2 TACSR 410 -11.0 3 TACSR 410 -11.2 4 TACSR 410 -11.2 5 TACSR 410 11.2 6 TACSR 410 12.0	Conductor no. NO Line type NAME Horizontal position X [m] Horizontal installation position bright 0 IACSR 150 -14.5 79.5 0 IACSR 150 14.5 79.5 1 TACSR 410 -11.2 72.0 2 TACSR 410 -11.5 57.5 3 TACSR 410 -12.0 43.0 4 TACSR 410 11.2 72.0 5 TACSR 410 11.2 72.0 6 TACSR 410 11.6 57.5	Conductor no. NO Line type NAME Horizontal X [m] Stele tower installation height YT[m] Dip DIP [m] 0 IACSR 150 -14.5 79.5 0 1 IACSR 150 14.5 79.5 0 2 TACSR 410 -11.2 72.0 0 3 TACSR 410 -11.2 57.5 0 4 TACSR 410 -12.0 43.0 0 5 TACSR 410 11.8 57.5 0	Conductor no. NO Line type NAME Horizontal X [m] Stele tower installation height YT[m] Dip DiP Im Dip Nomber of conductors 0 IACSR 150 -14.5 79.5 0 1 0 IACSR 150 14.5 79.5 0 1 1 TACSR 410 -11.2 72.0 0 6 2 TACSR 410 -11.2 57.5 0 6 3 TACSR 410 -12.0 43.0 0 6 4 TACSR 410 11.2 72.0 0 6 5 TACSR 410 11.2 72.0 0 6 6 TACSR 410 11.2 72.0 0 6	Conductor no. NO Line type NAME Horizontal SEP (m) Steel tower installation beight 'IT(T_1) Dip DiP DIP (m) Number of conductors DIP (m) Conductors installation DIP (m) 0 IACSR 150 -14.5 79.5 0 1 1 IACSR 150 14.5 79.5 0 1 1 IACSR 410 -11.2 72.0 0 6 0.5 2 TACSR 410 -11.2 57.5 0 6 0.5 3 TACSR 410 -12.0 43.0 0 6 0.5 4 TACSR 410 11.2 72.0 0 6 0.5 5 TACSR 410 11.2 72.0 0 6 0.5 6 TACSR 410 11.2 72.0 0 6 0.5 6 TACSR 410 11.2 72.0 0 6 0.5 6 TACSR 410 11.6 57.5 0 6 0.5

Only these rows will then be selected.

Туре	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.

	Туре	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
1	Do not delete 🗸	1							0 ~
×.	Do tulete ~	2	TACSR 410						0 ~
-	New r	ow is inser	rted		1			d1	0 ~

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.

	Туре	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 no. telete v	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 🗸	\square							0 ~
-		Copied r	ow is inserted						

Delete

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

Туре	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.

Туре		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.

Ground resistivity	300 [Ωm]	1 Draw(Check)	Save	Calculate

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.

Calculation type selection
\odot π type equivalent circuit model creation
 Fixed constant line model creation
 Frequency dependent line model creation
 Matching resistance
 Constant calculation only
Select Cancel

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.

Frequency sample (Logarithmic sample)	Start point 0.1 Hz < Endpoint 10 MHz Number of Samples 400 Point ≥ 2	● Hz ● kHz ● MHz
Line length	5000 m V	— Select un
	Run Cancel	• km

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'.

**		Save As				x
🐑 🤿 - 🕇 🎴	« work > Examples > STL-02		v C	Search STL-02		ρ
Organise 👻 Ne	w folder				H •	
Argentiates Argentiates Desktop Bownloads Recent places Argentiate Argentiate	_ verically.lin					
File name:						v
Save as type:	Line placement data (*.lin)					۷
Hide Folders				Save	Cancel	

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.

🔣 Line	e-constant calc	ulation result			- 🗆	×
<u>F</u> iles	<u>E</u> dit					
4.	57980E-01	-3.72024E-01	5.32439E-01	2.13857E-02	-2.69505E-01	-4. ^
Current	t transform	ation matrix T	il (i_phase = T:	il*i_mode)		
3.	49065E-01	5.98357E-01	2.37213E-03	-7.33994E-01	-2.32791E-01	-3.
3.	04669E-01	-5.23027E-02	-4.25494E-01	1.23271E-01	5.89933E-01	-5.
5.	55739E-01	-4.09667E-01	-5.92833E-01	-1.63711E-03	-3.45363E-01	4.
3.	49065E-01	5.98357E-01	-2.37213E-03	7.33994E-01	-2.32791E-01	3.
3.	04669E-01	-5.23027E-02	4.25494E-01	-1.23271E-01	5.89933E-01	5.
5.	55739E-01	-4.09667E-01	5.92833E-01	1.63711E-03	-3.45363E-01	-4.
Imagin	ary part of	Tvl which wil	l be discarded			
-5.	13987E-02	6.17858E-02	1.51759E-01	-2.83532E-02	-4.19947E-02	7.
-1.	03230E-03	1.15190E-01	-1.32464E-02	-1.70220E-01	-8.80652E-04	5.
3.3	28500E-02	7.55374E-02	1.01145E-02	-1.05105E-01	2.68021E-02	1.
-5.	13987E-02	6.17858E-02	-1.51759E-01	2.83532E-02	-4.19947E-02	-7.
-1.	03230E-03	1.15190E-01	1.32464E-02	1.70220E-01	-8.80652E-04	-5.
3.	28500E-02	7.55374E-02	-1.01145E-02	1.05105E-01	2.68021E-02	-1.
						~
<						>:

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.

Kale Line-constant calculation result
<u>F</u> iles <u>E</u> dit
Save Ctrl+S .72024E-0
Save As
Exit .98357E-(
3.04669E-01 -5.23027E-0
5.55/39E-01 -4.0966/E-0
Kine-constant calculation result
Files Edit
4.: Copy All 024E-01
Current transformation matrix 1

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)

	Characteristic impedance Z0 in phase coordinates [ohm] Real part: 3 56809E-002 9 46781E-001 608974E-001
Deace domain constant (before C)W elimination)	9.46781E+001 3.54126E+002 9.46781E+001
Phase domain constant (before GW elimination)	6.08974E+001 9.46781E+001 Characteristic impedance
PHASE DOMAIN QUANTITIES (before ground-wire elimination)	-3.58050E+001 -1.88627E+001 -1.77957E+001 -1.88627E+001 -3.60287E+001 -1.88627E+001
Space inductance Ls in phase coordinates [H/m] 115332E-006 2866565-007 1633865-007 3720045-007 1953575-007	-1.77957E+001 -1.88627E+001 -3.58050E+001
2.86656E-007 1.15332E-006 2.86656E-007 3.03782E-00	Modal domain constant
1.63786E-007 2.86656E-007 1.15332E-006 1.95357E-00 Space inductance 3.72004E-007 3.03782E-007 1.95357E-007 1.81650E-00	MODA (before transformation matrix normalization)
1.95357E-007 3.03782E-007 3.72004E-007 2.58400E-007 1.81650E-006	Series impedance z in modal coordinates [ohm/m]
Ground-return impedance Ze (including Ls) in phase Real part:	Real part: 1.91824E-004 3.84136E-005 3
4.71730E-005 4.71656E-005 4.71435E-005 4.68207E-005 4.67989E-005 4.71656E-005 4.71730E-005 4.71656E-005 4.68167E-005 4.68167E-005	Imaginary part: Line series impedance
4.71435E-005 4.71656E-005 4.71730E-005 4.67989E-005 4.68207E-005	
4.67989E-005 4.68167E-005 4.68207E-005 4.6400E-005 4.6400E-005	Real part:
Imaginary part: 5.61990E-004 2.87882E-004 2.44337E-004 3.06716E-004 2.4/195E-004	Imaginary part: Line parallel admittance
2.87882E-004 5.61990E-004 2.87882E-004 2.84512E-004 2.84512E-004 2.44337E-004 2.87882E-004 5.61990E-004 2.47195E-004 3.06716E-004	2.50197E-009 3.60544E-009 4. 21310E 009
3.06716E-004 2.84512E-004 2.47195E-004 7.52239E-004 2.60287E-004 2.47195E-004 2.84512E-004 3.06716E-004 2.60287E-004 7.52239E-004	Propagation constants gamma in modal coordinates [1/m] Real part:
Conductor internal impedance Zc in phase coordinates [ohm/m]	1.82258E-007 6.49314E-008 7.08599E Propagation constant
Real part: 3.52761E-005_3.52761E-005_3.52761E-00	1.30484E-006 1.06650E-006 1.05462E-006
Imaginary part: 1927115-006 1927115-006 Conductor internal impedance	Propagation velocities in modal coordinates [Propagation speed
i otali series impedance 2 in phase coordinates [onm/m] Real part:	Characteristic impedance zu in modal coordinates [ohm] Real part:
8.24491E-005 4.71656E-005 4.71435E-005 4.68207E-005 4.67989E-005 4.71656E-005 8.24491E-005 4.71656E-005 4.68167E-005 4.68167E-005	5.21332E+002 2.95802E+002 2. Imaginary part: Characteristic impedance
4.71435E-005 4.71656E-005 8.24491E-005 4.67989E-005 4.68207E-005 4.68207E-005 4.68167E-005 4.67989E-005 3.39503E-004 4.64600E-005	-7.41909E+001 -1.80093E+001 -1.79006E+001
4.67989E-005 4.68167E-005 4.68207E-005 4.64 Imaginary part: Line series impedance	Voltage transformation matrix Tv (v_phase = Tv≉v_mode) Real part:
5.63927E-004 2.87882E-004 2.44337E-004 3.00710E-004 2.47130E-004	5.79195E-001 -7.07107E-001 -3.68007E-001 5.7829E-001 -7.46545E-015 8.52899E-001
2.44337E-004 2.87882E-004 5.63927E-004 2.47195E-004 3.06712E-004 2.44337E-004 2.87882E-004 5.63927E-004 2.47195E-004 3.06712E-004	5.79195E-001 7.07107E-00 Voltage transformation matrix
3.06716E=004 2.84512E=004 2.47195E=004 7.58026E=004 2.60287E=004 2.47195E=004 2.84512E=004 3.06716E=004 2.60287E=004 7.58026E=004	Imaginary part: -1.94289E-016 -1.11022E-016 -2.62710E-003
Space capacitance Cs in phase coordinates [F/m]	2.47400E-003 -3.62296E-017 0.00000E+000 0.00000E+000 0.00000E+000 -2.82716E-003
1.08700E-011 -1.96320E-012 -6.06465E-013 -1.76637E-012 -4.65239E-013 -1.96320E-012 1.12047E-011 -1.96320E-012 -1.10364E-010 -1.10064E-010	Current transformation matrix Ti (i_phase = Ti*i_mode)
-6.06465E-013 -1.96320E-012 1.08700E-011 -4.65239E- -1.76637E-012 -1.10364E-012 -4.65239E-013 6.79219E- Space capacitance	Real part: 6.05018E-001 -7.07107E-001 -4.06454E-001
-4.65239E-013 -1.10364E-012 -1.76637E-012 -4.96389E-013 6.79219E-012	5.21512E-001 -5.02228E-01 6.05018E-001 7.07107E-00 Current transformation matrix
Shunt admittance in phase coordinates [S/m] Real part:	Imaginary part: -2.17105E-003 -4.80084E-016 -2.94431E-004
0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000	2.13498E-003 8.63732E-016 -2.94526E-003 -2.17105E-003 -5.91107E-016 -2.94431E-004
0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000	Modal domain constant
0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000 0.00000E+000	(after transformation matrix normalization)
1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.	MODA (
-6.16757E-010 3.52006E-009 -6.16757E-010 -3.46718E-010 -3.46718E-010 -1.90527E-010 -6.16757E-010 3.41492E-009 -1.46159E-010 -5.54921E-010	Series impedance z1 in modal coordinates [ohm/m] Real part:
-5.54921E-010 -3.46718E-010 -1.46159E-010 2.13383E-009 -1.55945E-010 -1.46159E-010 -3.46718E-010 -5.54921E-010 -1.55945E-010 2.13383E-009	1.90103E-004 3.84136E-005 3 Line series impedance
Phase domain constant (after GW elimination)	6.67223E-004 3.14302E-004 2 .02672E-004
PHASE DOMAIN QUANTITIES (after ground-wire elimination)	Shunt admittance y1 in modal coordinates [S/m] Real part:
Total series impedance Z in phase coordinates [ohm/m]	0.00000E+000 0.00000E+000 0. Line parallel admittance
Real part: 8 79763E-005 5 18901E-005 4 95627E-005	2.50200E-009 3.60544E-009 4.21517E-009
5.18901E-005 8.79618E-005 5.18901E-005 4 05672E-005 5.1001E-005 8.79618E-005	Characteristic impedance z01 in modal coordinates [ohm]
Imaginary part: Line series impedance	5.21519E+002 2.95802E+002 2. Characteristic impedance
4.16443E-004 1.33217E-004 1.02141E-004	-7.28452E+001 -1.80093E+001 -1.08103E+001
1.02141E-004 1.3921/E-004 4.16443E-004	Voltage transformation matrix Tv1 (v_phase = Tv1*v_mode)
Shunt capacitance C in phase coordinates [F/m] 1.08700E-011 -1.96320E-012 -6.06465E-013	5.79194E-001 -7.07107E- 5.73641E-001 -7.46545E- Voltage transformation matrix
-1.96320E-012 1.12047E-011 -1.96320E-012 -6.06465E-013 -1.96320E-012 1.08700E-011	5.79194E-001 7.07107E-0
Shunt admittance Y in phase coordinates [S/m]	Current transformation matrix Til (iphase = Til*imode) 6.05016E-001 -7.07107E-
Real part: 0.00000E+000 0.00000E+000 0.00000E+000	5.21505E-001 -5.05808E-0 Current transformation matrix
0.00000E+000 0.00000E+000 0.00000E+000	Imaginary part of Tv1 which will be discarded
Imaginary part: 34/402E-000 -616757E-010 -190527E-010	Imaginary part of voltage transformation matrix
-6.16757E-010 3.52006E-009 -6.16757E-010	
-1.90327E-010 -0.10797E-010 3.41492E-009	Symmetrical components
Propagation constants Gamma in phase coordinates [1/m] Real part:	symme (only displayed in case of three-phase line)
1.07247E-007 3.33391E-008 4.23153E-008 3.85996E-008 1.03556E-007 3.85996E-008	Positive-Sequence Impedance [ohm/km]
4.23153E-008 3.33391E-008 1.07247E-007 Imaginary part:	Real part: 3.69E-02 Positive-sequence-phase impedance
1.14810E-006 7.57169E-008 8.16053E-008 8.68693E-008 1.12976E-006 8.68693E-008	Imaginary p
8.16053E-008 7.57169E-008 1.14810E-006	Zero-Sequence Impedance [ohm/km]
	Real part:
	0.08E-01

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

Confirmation	×
Save resul	ts data?
Yes	No

2 Plot dialog

The 'Plot dialog' appears if 'Constant calculation only' or 'Create frequencydependent 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

ResView1	-		×
Frequency respo constants and p	nse of th ropagatio	e attenua n velociti	ation es
Frequency res characteristic Plot	ponse o impeda	f the nce mat	rix
Frequency respo transformation r Row [:] 1 Plot	onse of th natrix	e voltage)
(Close		

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.

			X
Fitting result of admittance m	of the ch atrix [Y(naracter)]	ristic
Row : 1		1	
Column : 1			
Plot			
Frequency resvalue of the c admittance m Plot	sponse (haracte atrix [Y(of the e ristic 0]	igen
Fitting result matrix [H0]	of the p	ropagal	tion
Row : 1		-	
Column : 1		-	
Plot			

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 [Y0] 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 *interceptility* 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.

Туре	Conductor no. NO	Line type NAME	Horizontal position X [m]	Ste Dou heigh	Ible-click	NCOND	Conductor- interval SEP [m]	Transposed group
Delete v	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	-112	72.0	0	6	0.5	0~
Do not delete 🗸	2	TACSR 41	-11.6	57.5	0	6	0.5	0 ~
Donot 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

XTAP - [NewCircuit1]	
🜃 File Edit Circuit Analysis	View Tools Window Help
i 🗋 💕 🛃 X 🗈 🏝 🔊 • (*	- 🛃 🚨 🚳 100% - 💽 🕂 빅 씨 🗠 👉 💁 A
<u>11</u>	
00 Ground, Terminal	
01 RLC	
02 Voltage Source, Current Source	
03 Switch	
04 Probe	
05 Transmission and Distribution Line	
In PI-Equivalent Circuit Line	
01 PI-Equivalent Circuit Line	
02 Constant-Parameter Distri	
Ø 03 Constant-Parameter Distri	PI_LineX1
A Di Frequency-Dependent Dist	
05 Single=Circuit 3-Phase Trail	
0.0 Ongle Ordult 3-Phase Tr	<u>i i i i i i i i i i i i i i i i i i i </u>
w vo pouble offcult offidse if	

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).

01 PI-Equivalent Circuit Line Model (XTLC)	-	-		×
Parameters Display Help				_
Number of 1 V Phases = Line Length m V				^
Data File		密照		
				~
[OK	(Cancel	Ľ

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

ХТАР	×
The xsf file must be	l in order to specify a reference.
	ОК

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.

	Conductor no. NO	Line type NAME	Horizontal position ×[m]	Steel tower installation height	Dip DIP [m]	Number of conductors NCOND	Conduct intervi SEP (r	tor- al n]	Transpose group
Delete ~	0	IACSR 120	-11	38.5	0	1			0
Delete ~	0	IACSR 120	11	38.5	0	1			0
Do not delete ~	1	1ACSR 240	-14	28.5	0	4		0.4	0
Do not delete 🗸	2	IACSR 240	0	28.5	0	4		0.4	0
Do not delete \vee	- 3	IACSR 240	14	28.5	0	4		0.4	0
Ground resistivity [300]B	[m]	1 Draw(Che	ick)		E	Save	1	Calculate
			× # 0						
		0		•					
				3					
			Ť						
					_				
ji Insert line, [Alt]+[Int): Select row (cop	y) and insert, [Alt]+[_]]/[↓]: Move	row up/down,	[Dei]: Delete	row (multipl	e selectio	ns all	iowed)
); Insert line, [Alt]+[Ins): Select row (cop	y) and insert, [Alt]+[]]]/[1]: Move	row up/down,	[Del]: Delete	row (multipl	e selectio	ns all	lowed)
); insert line; [Alt]+[PI equival generated	Int): Select row (cop ent model by XTLC on	y) and insert, [Alt]-[] (PI_Line) 15-Dec-2017 a	1/[+]: Move at 10:10	row up/down,):54.	[Del]: Deleta	now (multipl	e selectio	ns all	owed)
PI equival generated number of	Int: Select row (cop lent model by XTLC on phases = 3	y) and insert [AN]=[] (PI_Line) 15-Dec-2017 a	10:10	row up/down.):54.	[Del]: Delete	roe (multipl	e selectio	ns all	lowed)
PI equival generated frequency	Int) Select row (cop lent model (by XTLC on phases = 3 = 5.00000E	y) and insert [Att[-[]] (P[_Line) 15-Dec-2017 (+001.	∭.]:Move at 10:10	row up/down.):54.	[Del]: Delete	row (multipl	e selectio	ns all	owed)
PI equival generated number of frequency Desistors	Ins) Select row (cop lent model by XTLC on phases = 3 = 5,00000	y) and insert. [AN]-[]] (P[_Line) 15-Dec-2017 a 5+001.	11:1:Move	row up/down.):54.	[Del]: Delete	row (multipl	e selectio	ns all	iowed)
PI equival generated number of frequency Resistance 8 70762428	Ins) Select row (cop lent model (by XTLC on phases = 3 = 5.00000E = matrix:	y) and insert [AN]-[1] (P[_Line) 15-Dec-2017 4 (+001.	at 10:10	row up/down.):54. 21E-005	[Del]: Deleta	row (multipl	e selectio	ns all	iowed)
PI equival generated frequency Resistance 8.79763435 5.18900521	ent model by XTLC on phases = 3 = 5.00000E matrix: 32211378E-00	y) and insert, [AN]-[]] (PI_Line) 15-Dec-2017 (+001. (5, 5, 189005) 5, 8, 796184	21943202 29478114	row up/down,):54. 21E-005, 18E-005,	(Del]: Delete 4. 956/ 5. 1890	row (multipl 27048657	e selectio	ns all	iowed)
I heret line; [Att]=[PI equival generated number of frequency Resistance 8. 19763435 5. 18900521 4. 95627045	Ind; Select row (cop lent model (by XTLC on phases = 3 = 5,0000E = matrix: 32271378E-00 9432020E-00 6571305E-00	y) and insert [Alt]-[] [P] Line) 15-Dec-2017 a :+001. 15. 5. 189005; 15. 5. 189005; 15. 5. 189005;	<pre>//:): Move at 10:10 21943202 29478114 21943202</pre>	row up/down, 0:54. 21E-005, 18E-005, 19E-005,	(Del): Delete 4. 9556 5. 1890 8. 7977	row (multip) 27048657 3243522	e selectio 71306E 2019E 11372E	-00 -00	owed) 05 05
PI equival generated number of frequency Resistance 8. 19763435 5. 18900521 4. 95627046 Inductance	Int: Select new (cop by XTLC on phases = 3 = 5.00000E matrix: 32371378E-00 432020E-00 66571305E-00 matrix: matrix:	y) and insert [AN]-[] (P[_Line) 15-Dec-2017 a (+001.)5, 5, 189005/ 15, 8, 796184/ 15, 5, 189005/	at 10:10 21943202 29478114 21943201	row up/down, 0:54. 21E-005, 18E-005, 9E-005,	[Del]: Delete 4. 9562 5. 1890 8. 7970	row (multip) 27048657 00521943 33435327	e selectio 71306E 82019E 71372E	-00 -00	iowed) 05 05
PI equival generated number of frequency Resistance 1, 9562704 Inductance 1, 32557920	ind; Select row (cop ent, model) by XTLC on phases = 3 = 5,000000 94320200-00 56571305E-00 matrix: 2074468E-00 matrix:	y) and insert [AN]-[1] (PI_Line) 15-Dec-2017 a (+001. (5, 5, 189005) (5, 5, 189005) (6, 4, 431426)	at 10:10 21943202 29478114 21943201 60324224	row up/down, 0:54. 21E-005, 18E-005, 19E-005, 47E-007,	(Del): Deleta 4. 9566 5. 1890 8. 7970 3. 2512	row (multipl 27048655 00521943 33435327 25306873	e selectio 11306E 22019E 11372E 87617E	-00 -00 -00	iowed) 05 05 07
PI equival generated number of frequency Resistance 3, 1976343 5, 18900521 4, 95627045 1.040557925 4, 4314266	Int) Selectrow (cop lent model) by XTLC on phases = 3 = 5.00000E 9:matrix: 3271378E-00 9432020E-00 9422020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 9432020E-00 94200E-00 94000 9400E-00 94000E-00 9400E-00 9400E-00 9400E-00 94000E-00 94000	y) and insert [AH]=[1] (PI_Line) 15-Dec-2017 a 5+001. (5, 5, 189005) (5, 8, 796184) (5, 5, 189005) (6, 4, 4314261 (7, 1, 309332)	P[1]: Move at 10:10 21943202 29478114 21943201 60324224 62112805	row up/down, 0:54. 21E-005, 18E-005, 18E-005, 19E-007, 26E-006,	(Del): Delete 4. 9562 5. 189 8. 7970 3. 2512 4. 431	row (multipl 27048657 00521943 33435327 25306873 42660324	11306E 2019E 11372E 27617E 22246E	-00 -00 -00	owed) 05 05 05 07
PI equival generated number of frequency Resistance 3.976343 5.1890052 1.42557022 4.43142860 3.251230	Int: Select new (cop by XTLC on phases = 3 = 5.00000E e matrix: 2074468E-00 3242247E-00 3242247E-00 32424247E-00	y) and insert [AN]-[1] (P[_Line) 15-Dec-2017 a (+001.)5, 5, 189005 (5, 5, 189005) (5, 5, 189005) (6, 4, 431426()7, 1, 309332 (7, 4, 431426)	at 10:10 21943202 29478114 21943201 60324224 50324224 50324224	row up/down. 0:54. 21E-005, 18E-005, 9E-005, 47E-007, 16E-006, 16E-006,	(Del): Delete 4. 9562 5. 1890 8. 7976 3. 2511 4. 431 1. 3255	row (multipl 27048657 0521943 3343522 25306873 25306873 25306873 25306873	e selectio 11306E 12019E 11372E 37617E 12246E		lowed) 05 05 05 07 07
PI equival generated number of frequency Resistance 3. 19763433 5. 18900521 4. 9562704 1.32557922 4. 3142660 3.251250 3.251250 2.351250 2.3512500 2.3512500000000000000000000000000000000000	ing Select new (cop ent model) by XTLC on phases = 3 = 5.00000E ematrix: 2074468E-0 3242247E-00 33242247E-00 33242247E-00 33242247E-00 33242247E-00 33242247E-00 33242247E-00 332427E-00 332427E-00 34247E-00 3427E	y) and insert [AN]-ET (PI_Line) 15-Dec-2017 a 5-001. 15. 5. 1890052 15. 5. 1890052 15. 5. 1890052 15. 5. 1890052 15. 4.31426 11.062192 11.06	at 10:10 21943202 29478114 21943201 60324224 50324224 50648732	row up/down, 2:54. 21E-005, 18E-005, 19E-005, 47E-007, 16E-006, 16E-007, 20E-012	(Del): Delete 4. 9562 5. 1899 8. 7970 3. 2512 4. 4314 1. 3255	row (multipl 27048655 0521943 33435327 25306872 25306872 25306872 25306872 25306872 25306872 25306872 25306872 25306872 25306873 2530587 25306873 25300873 25300875 25300000000000000000000000000000000000	e selectio 11306E 12019E 11372E 11372E 1246E 12468E 12468E	-00 -00 -00 -00	owed)

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.

01 PI-Equivalent Circuit Line Model (XTLC)	-	- 1	0	×
Parameters Display Help				
Number of Phases = Line Length =				^
Data File				
horizontally.xmf		参照]	
				~
	OK	0	ancel	

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.



[[Ins]: Insert line; [Ah]+[Ins]: Select row (copy) and insert; [Ah]+[1]/[1]: Move row up/down; [Del]: Delete row (multiple selections allowed)

ſ	# 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.
	<pre>f Characteristic impedance, propagation velocity, and resistance pul f or each model impedance, propagation velocity, and resistance pul 1 for each model in 105 ePo2. 3 159105162540202-007. 3 668219809076400E-004 1.227165780511015E+002. 8 179456225032+007. 1 005076531205788E-005 1.276680763622533E+002. 9 901337678855768E+007. 6 645058131679280E-005 4.238383503844301E+001. 1 .312387314873098E+008. 7 .15592855693360EE-005 4.3355138517278621E+001. 1 .35983704845909E+008. 6 .7521407012757858E-005 5 Current transformation matrix: 1.58415312475315E-003. 3 .248010232970500E-0027.258210685965727E-0024.190390423946063E-0017.122868059792730E-001. 5.720938400081267E-001 8.084180185341912E-0019.995790846765368E-0015.680607949657368E-001. 4.7844734586851E-001. 7.403377494424469E-0015.706882451770582E-001 8.084180185341912E-0019.995790846765368E-0015.680607949657368E-001. 4.72447345868581E-001. 7.43377484424469E-0015.1068824517705825694E-001 8.13841811247155092E-0041.88773914182766E-015. 1.099834570856578E-0009.261798835433226E-0011.741523780774E-014. 5.94051542159737E-001 -1.5841531547315E-003232010518200751E-015. 1.099834570856578E-0009.261798835433226E-0011.741523780774E-014. 5.2405154215592E-0041.52415321569042-01252821066856578E-0012528179859484782E-015. 1.29821782E-015.</pre>

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.

PI-Equivalent Circuit Line Model (XTLC)	-		×
Parameters Display Help			
Number of 6 V Phases = 6 V Line Length 5 km V			<
Data File			
cableamf	3	明確	
			×

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.

	Туре	Conductor no. NO	Line type NAME	Horizontal position X [m]	Steel tower installation height YT[m]	Dip DIP [m]	Number of conductors NOOND	Conductor- interval SEP [m]	Transposed eroup
	Delete 🗸	0	IACSR 120	-11	38.5	0	1		0.5
	Delete 🗸	0	IACSR 120	11	38.5	0	1		0.
Г	Do not delete 🗸	1	IACSR 240	-14	28.5	0	4	0.4	
	Do not delete 🗸	2	IACSR 240	0	28.5	0	4	0.4	
E	Do not delete 🗸	3	IACSR 240	14	28.5	0	4	0.4	8.4
	Do not delete 🗸								0
Gr	Do not delete v	200 [0	n) [1 Draw(Check)		S	sve C	alculate

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.

07 Matching Resistance	- 🗆 🗙
Parameters Record Display Help	
Nuclea	~
of Phases	
Data file	
horizontally.xmf	Choose Fi
<	> ×
ОК	Cancel

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.

k		XTLC(Li	ne-constan	t calculation	n program):	cable.lin	-	- 🗆 🗙	F	
Ð	les ⊻iew Databa	se <u>H</u> elp								
Γ	Cable no. NO	Line type NAME	Horizontal position × [m]	Depth Y (m)	Conduit-line diameter DP [mm]	Conduit-line thickness TP [mm]	Conduit-line dielectric constant	Triplex group	r	
F	1	CAZV220kV 600	-0.81	1	54	14	1 EPOP	0 v		
	2	CAZV220kV 600	0	1	54	14	1	0 v		
Þ.	3	CAZV220kV 600	0.81	1	54	14	1	0 v		
•							1	0 4		
	Ground resistivity	100 [0m]		1 Draw(Check)			Save	Calculate		
_									-	
				X = 0						
			1	-	3					
			G		Ģ					
ftes	:): Insert line, [Alt]+[Insl: Select row (con	w) and insert.	[Alt]+[1]/[].	1: Move row un	/down. [Del]: [elete row (multi	inle selection -		
fund	if more me, parti-	maj. Selection (cop	y) und moercy	parter here.	J. Hore for up	footing [bei]. i	verete ron (main	the actection		
	Matching resistan generated by XTLC	ce model (MR_mode on 15-Dec-2017 a	1) t 10:38:26.							
	number of phases	= 6								
1	frequency = 1.00	000E+003.								
1	Matching resistan	ce matrix:	22622202E+0	01 2 96003	E000623927E.	001 2 9617	070656021155-	001 2.222	1720642476216+001	2 2224765069472225+00
1	8. 064406622622296	E+002, 8.0644066 E+001, 8.0578517	96668292E+0	01, 2.86007	7065603043E	001, 2.8634	79841799112E+	001, 2.332	476506847240E+001,	2. 333476506847332E+00 2. 334756921469983E+00
	z. 860075999623719 2. 861797065603032	E+001, 2.8617970 E+001, 2.8634798	65602998E+0 41799101E+0	01. 1.17695 01. 7.89169	9494544502E+ 3276165240E+	002, 7.8916	93276165128E+ 47732828951E+	001, 2.860	075999623617E+001, 797065603028E+001,	2.861797065602900E+00 2.863479841799098E+00
	2. 332172964247524 2. 333476506847234	E+001, 2.3334765 E+001, 2.3347569	06847233E+0 21469977E+0	01. 2.86007	5999623760E	001, 2.8617	97065603039E+ 79841799103E+	001, 1.194	200748458744E+002. 406622622290E+001	8. 064406622622290E+00 8. 057851796668288E+00
- 2		2.0011000				2.0004			CONTRACTOR OUT	0. 000000000 00E · 00

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.

07 Matching Resistance		×
Parameters Record Display Help		
Number of Phases =		^
Data file cable.xmf	Choose File	
		\sim
<	>	
ОК	Cancel	

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

Market Karley Ka	_XTAP_EN_V240¥work¥Examples¥	5	
🚟 File Edit Circuit Analysis Vie	ew Tools Window Help		
i 🗋 💕 📕 X 🗈 🎕 🤊 • 🔍 • 🛃	집 중 225% → 장 수 작 사		
∎ [↓] ₩~₽			
00 Ground, Terminals 01 RLC Components 02 Voltage and Current Sources 03 Switches 04 Probes 05 Transmission and Distribution Line 0 01 Pt-Equivalent Circuit Line 0 01 Pt-Equivalent Circuit Line 0 02 Constant-Parameter Distri 0 04 Frequency-Dependent Dist 0 05 Single-Circuit 3-Phase Tr 0 05 Single-Circuit 3-Phase Tr 0 07 Matching Resistance 03 Photovoltaic Power Generation 09 Photovoltaic Power Generation 10 Electric Motors, Loads	R_Matching1		Conductor No. 1 core Conductor No. 1 sheath Conductor No. 2 core Conductor No. 2 sheath Conductor No. 3 core Conductor No. 3 sheath

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.

Search C Colle O New	Flegischer Clebeta	Import Export	Exit
eveloper-provided	Name AC 30		
Cross-sectional area	27.09 [mm2]		
Electric resistance	2.94 [Ω/km]		
Relative magnetic permeability	1		
		Outside Diameter	6.9 [mm]

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.

🔿 Search 🖲 Edit	🔿 New
-----------------	-------

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

🔿 Search	🔿 Edit	New
----------	--------	-----

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

🔿 Search 🖲 Edit 🛛 New	Register	Delete	
-----------------------	----------	--------	--

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

🔾 Search 🖲 Edit	O New	Register	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.

🔣 XTLC(Line-constant calculation program):(Untitled)		
Files View	Database	Help
i 🗋 📂 🗔 I	Overhead transmission line	
	Overh	ead distribution line
	Cable	

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	I.	Search

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.

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



Display switching buttons

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

• [<<] button

Displays the first data.

• [>>] button

Displays the last data.

• [<] 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	c 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

$$Y(j\omega) \cong \sum_{n=1}^{N} \frac{R_n}{j\omega - p_n}$$
(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] <u>CRIEPI Research Report T03013</u>: "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] <u>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.)</u>
- [3] <u>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</u> (← 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:

where

f(k) is the *k*th frequency; mag(i,j,k) is the amplitude of element (*i*, *j*) at the *k*th frequency; 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.

XFIT 1.3	-		×
Number of phases			
Nph = 1			
Frequency response dat	ta files		
Icol = 1 🗘 Not	yet selected		
Select No fi	iles selected.		
Error tolerence in relative	e quantity		
EpsR = 1.0 %			
Low level fitting paramet	ers		
Nmin = 6 🜩	Nmax =	16 🜩	
Nitr = 30 🗭	Nsvd =	2 🜩	
synmmetric	limit poles		
pole clustering	EpsP =	5.0 9	%
outputs			
Default			
Start fitting			
Start Not y	vet fitted.		
Close Help	Abou	•	

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 $0 \le \text{EpsR}$. 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: $1 \le Nmin \le Nmax$; Default: 6)
- Nmax: Stop order for an optimal model-order search. (Range: Nmin ≤ Nmax; Default: 16)
- Nitr: Iteration stop count for a rational function fitting process. (Range: $1 \le Nitr$; 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 ≤ Nsvd ≤ 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 Re{Y}: plots the eigenvalues of Re{Y(*jω*)} versus frequency for the fitted model. The horizontal axis (frequency) may be either linear or logarithmic. If the eigenvalues of Re{Y(*jω*)} are positive for all the frequencies, the fitted model is guaranteed to be positive-definite (i.e., passive).

👬 ResultViewer	-		×			
Plot (i,j) element of Y						
i (row) = 1 😩						
j (col) = 1 🚖						
Lin. Plot Log). Plot					
eigenvalues of Re{Y}						
LinPlot Log). Plot					
Save results						
Generate						
a network equivalent in XTAP format.						
Save						
results to file in binary format (see help).						
Save						
results to file in text format						
Save						
results to file in Matlab .mat format.						
Close Help						

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)