Attracting Tomorrow

TDK Component Library for Keysight PathWave ADS

ver. 2022.07

TDK Corporation
Electronic Components Business Company
Marketing Strategy Group
Products & Application Collaboration
July 1, 2022

Caution

- **Applicable condition**

  The parameters in this library are obtained under the condition of 25ºC, no DC bias (excepting the DC bias model and the DC superimposition model), and small signal operation. Proper result might not be obtained if your condition is different from the above one.

- **Terms and conditions regarding TDK simulation models**

(1) This simulation model is being provided solely for informational purposes. Please refer to the specifications of the products in terms of detailed characteristics of such products.
(2) In no event shall TDK Corporation of any of its subsidiaries be liable for any loss or damage arising, directly or indirectly, from any information contained in this simulation model, including, but not limited to loss or damages arising from any inaccuracies, omissions or errors in connection with such information.
(3) Any and all copyrights on this simulation model are owned by TDK Corporation. Duplication or redistribution of this simulation model without prior written permission from TDK Corporation is prohibited.
(4) This simulation model is subject to any modification or change without any prior notice.
(5) Neither TDK Corporation nor any of its subsidiaries shall make any warranty, express or implied, including but not limited to the correctness, implied warranties of merchantability and fitness for a particular purpose with respect to this simulation models.
(6) The use of this simulation model shall be deemed to have consented to the terms and conditions hereof.

About this library

- **Feature of this library**
  - The actual property of components can be taken into your circuit simulation because equivalent circuit model that considers inner structure of a part and material property is used.
  - Artwork data (recommended pcb pattern) of parts are included
  - Easy operation like standard ADS components.
  - Both palette list and component library are used to put components.
  - Discrete optimize can be used.
  - Regarding capacitor and inductor for RF circuit, the models considering tolerance of the part are included
  - The DC superimposition characteristics of power-use inductors and the DC bias characteristics of high dielectric constant type ceramic chip capacitors can be simulated.

- **Supported ADS versions**

  This library can be used with ADS2004A or latter versions. However, this library might not be used depending on a simulation environment. Please acknowledge it beforehand.

- **Contents in this document**

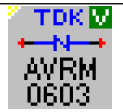
  This document is described assuming the following environment.
  - OS: Windows 10
  - ADS: ADS2020

  On different OS or ADS versions, screen display and/or operation procedure may not correspond to the contents of this document. Please acknowledge it beforehand.

About the model included in the library

- **Abstract of the model, and model for each product**

  5 types of model are included in this library. The followings describes the abstract of each model type, and the model used in each product.

model type	frequency model	tolerance model	DC superimposition model	DC bias model	voltage–current model
modeled property	・frequency characteristics	・frequency characteristics ・min. and max. values for inductance/capacitance	・frequency characteristics ・DC current dependence of inductance	・frequency characteristics ・DC voltage dependence of capacitance	・frequency characteristics ・voltage–current property of impedance
example of icons					

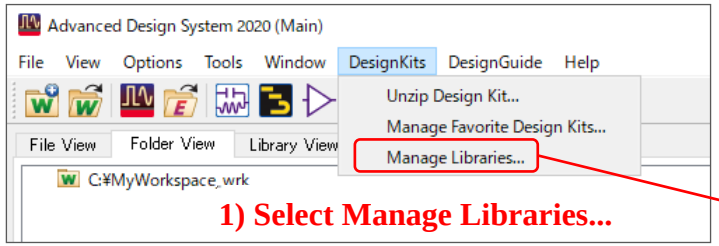
product/type		model type
multilayer ceramic chip capacitor	temperature compensation type	tolerance model
	high dielectric type	DC bias model
inductors	for high frequency circuit	tolerance model
	for standard circuit	DC superimposition model /frequency model(*)
	for decoupling circuit	
	for power circuit	DC superimposition model
chip beads		frequency model
3–terminal filters		
common mode filters		
varistors		voltage–current model
chip protectors		
pulse transformers		frequency model

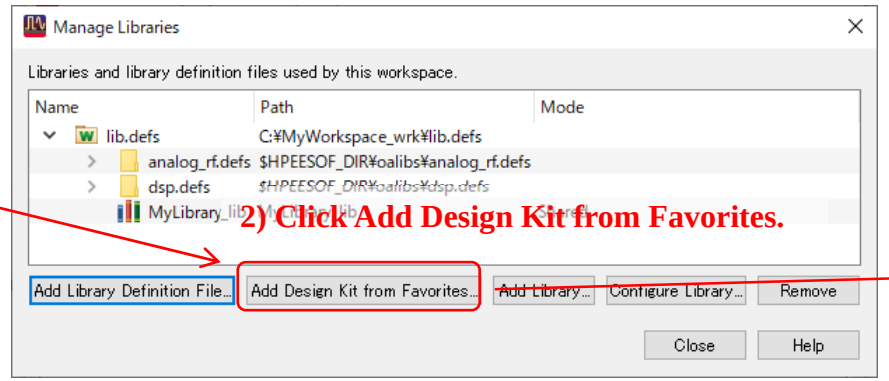
*Model type depends on the product. Please refer the product list for detailed information.

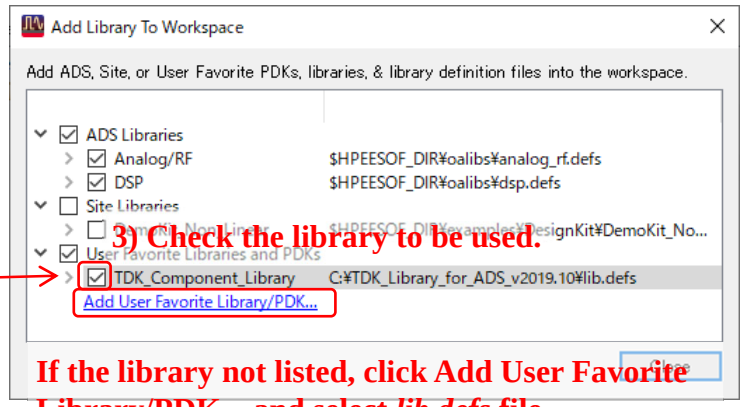
How to setup

- **Preparation of the library file**
  Unzip the library data file (e.g. tdk_library_for_ads_v202207.zip) and save it at any directory.

- **Adding the library to an existing workspace**
  1) Open a workspace and select Manage Favorite Design Kits... from DesignKits menu.
  2) Click the Add Design Kit from Favorite button.
  3) Put a check mark at the library to be used. If the library is not listed, click the Add User Favorite Library/PDKs and select the lib.defs file that is in the unzipped folder.

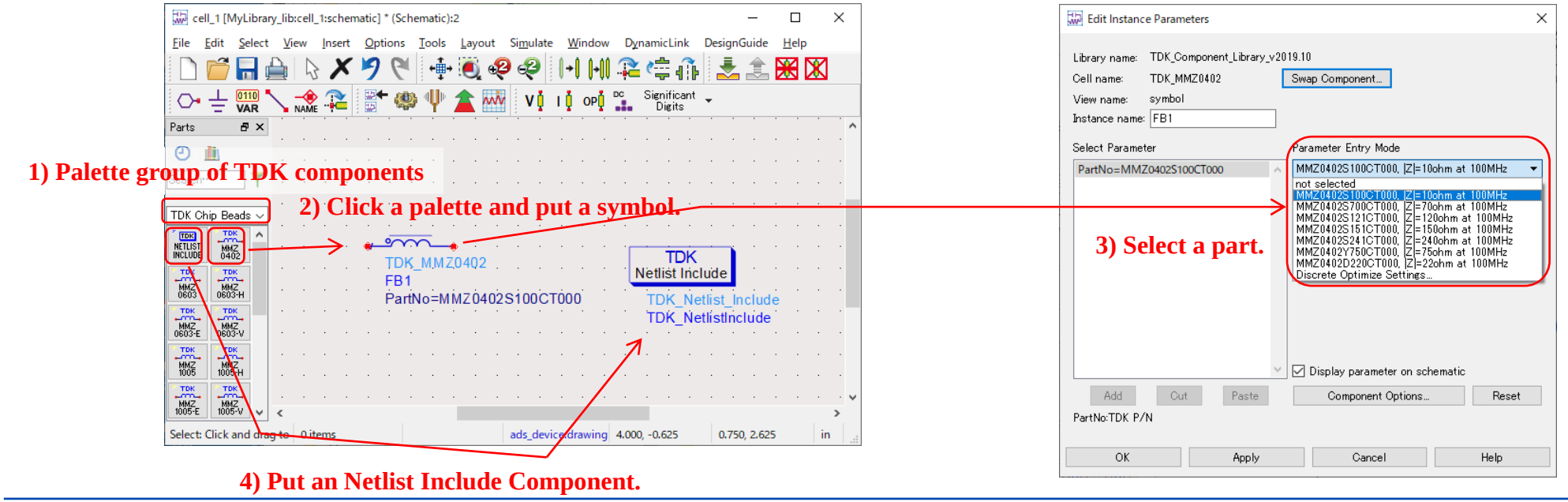






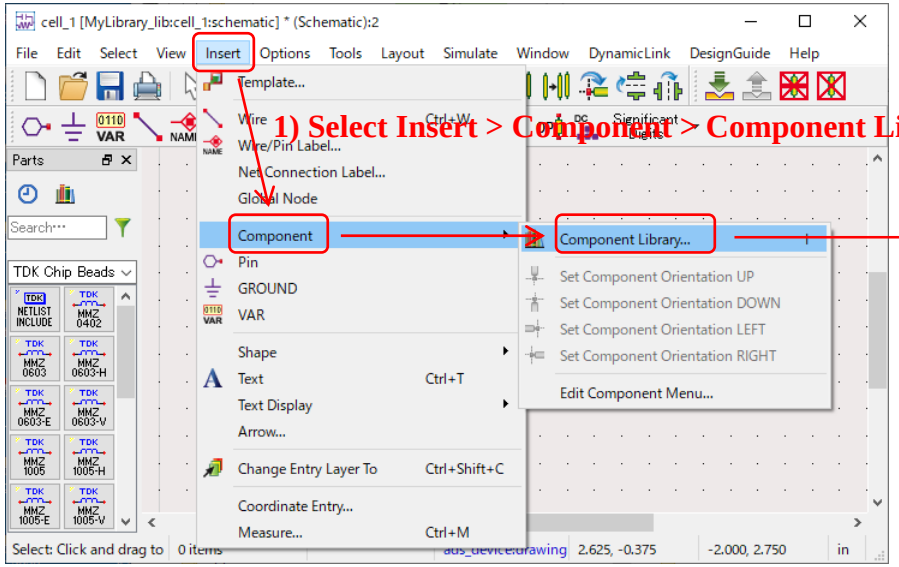
How to use the library (1)

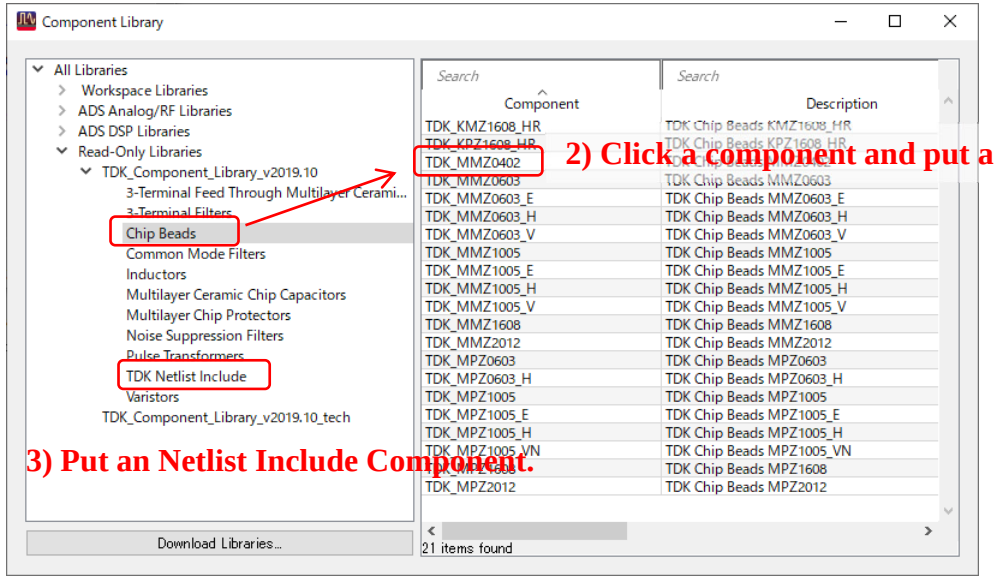
- **Putting a component from Palette List**
  1) Palette group of TDK components is added in component palette list after installation.
  2) Click a palette of component to be used and put a symbol on the schematic.
  3) Double-click the symbol of the component and open the component setting window. Select a part on the dropdown list of the setting window.
  4) An Netlist Include Component is necessary to use TDK components. Click the palette of the Component and put it on the schematic.



How to use the library (2)

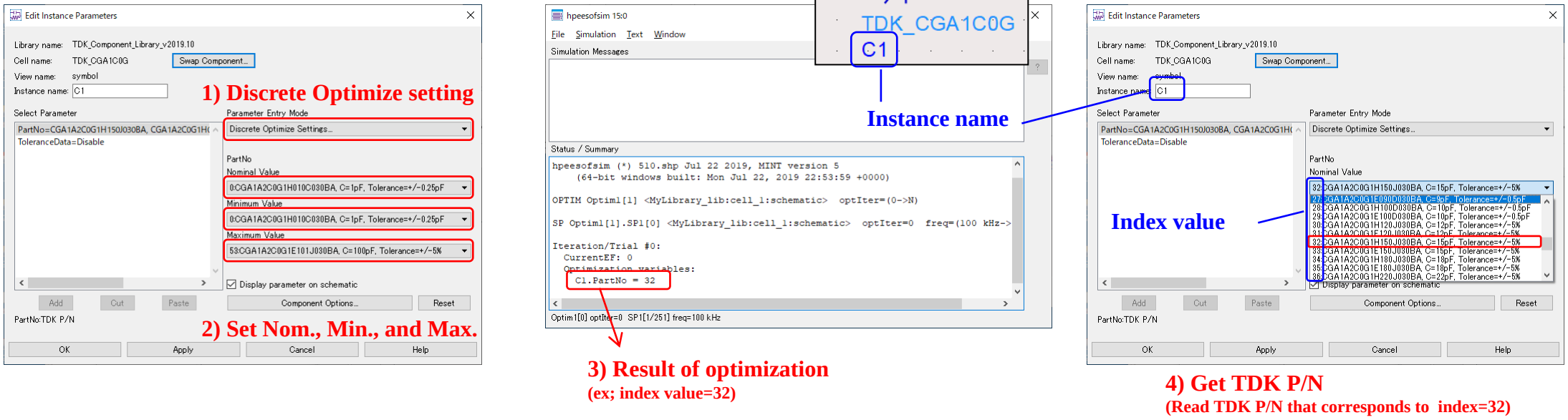
- **Putting a component from Component Library**
  1) Click the icon of Display Component Library List and open the Component Library window.
  2) Click a component to be used and put a symbol on the schematic.
  3) An Netlist Include Component is necessary to use TDK components. Click the component and put it on the schematic.





How to use the library (3)

- **Discrete optimize**
  1) Select Discrete Optimize setting listed in the last of the dropdown list in the component setting window.
  2) Set Minimum, Nominal, and Maximum and start simulation.
  3) A result of optimization is shown with a format of "<instance name>.Name = <index value>" in a status window.
  4) Instance name corresponds to the one that is shown in a schematic window. Index value of components are shown in left side of the dropdown list in the component setting window. Get TDK part number that corresponds to the index value from the list.



DC superimposition model / DC bias model

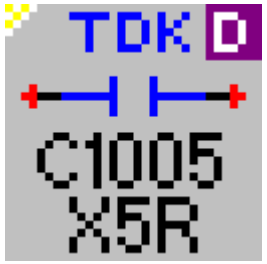
- **Abstract of the DC superimposition model and the DC bias model**

  Power-use inductors have a property called “DC superimposition property” which means the change of the inductance by the DC current applied to the inductors. Ceramic capacitors have a property called “DC bias property” which means the change of capacitance by the DC voltage applied to the capacitors. Those properties can be considered in the simulation from the library version 2015.05. The components in which those models are used have “D” mark on their icon.

frequency model

DC superimposition model

DC bias model

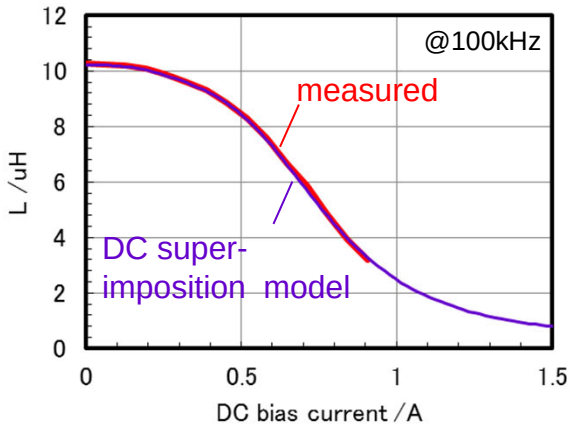


DC superimposition model

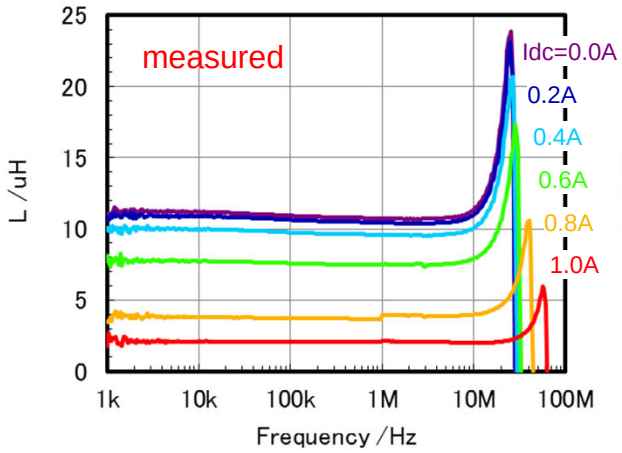
- **Comparison between DC superimposition model and measured data**

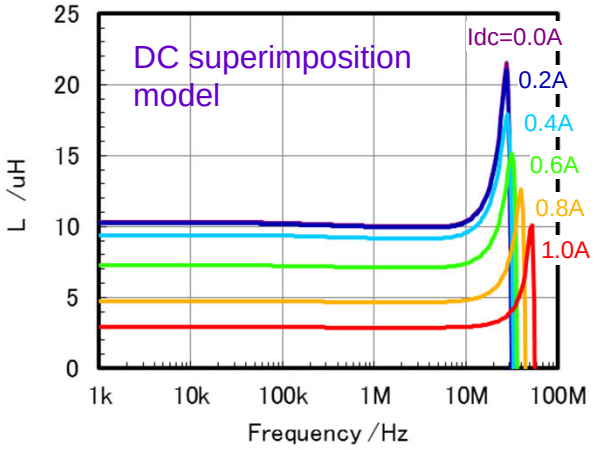
Part No.: VLS3010ET-100M

inductance vs. DC superimposition current



frequency dependence of inductance for various values of DC superimposition current





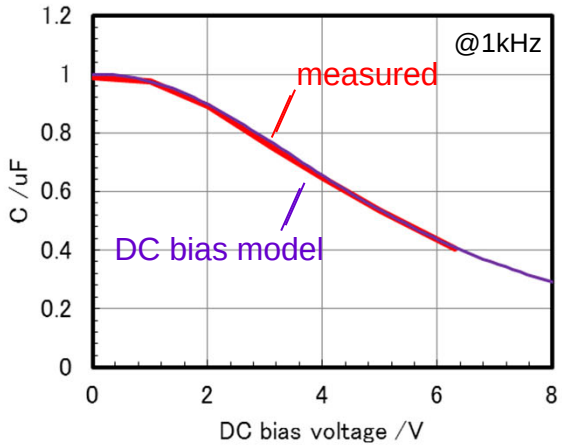
- The change of inductance by DC superimposition current is modeled in the **DC imposition model**.
- In the **DC superimposition model**, the frequency dependence of impedance is also modeled.

DC bias model

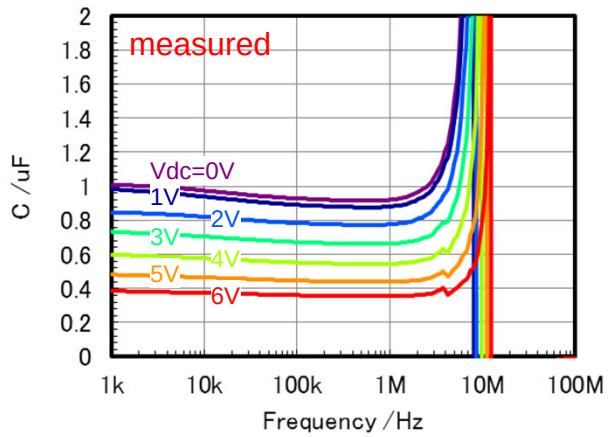
- **Comparison between DC bias model and measured data**

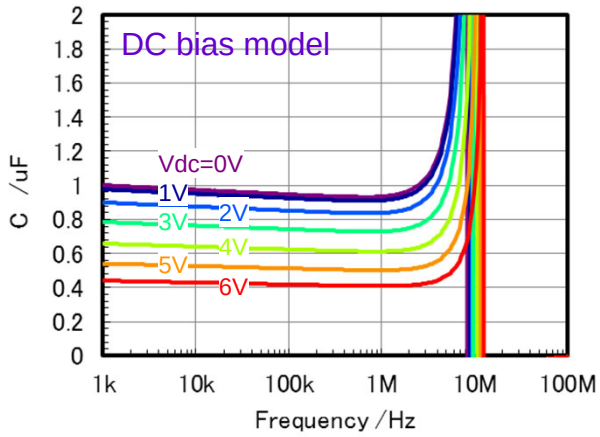
Part No.: C1005X5R0J105K050BB

capacitance vs. DC bias voltage



frequency dependence of capacitance for various values of DC bias voltage



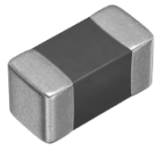


- The change of capacitance by DC bias voltage can be simulated in the **DC bias model**.
- In the **DC bias model**, the frequency dependence of impedance is also modeled.

Voltage-current model

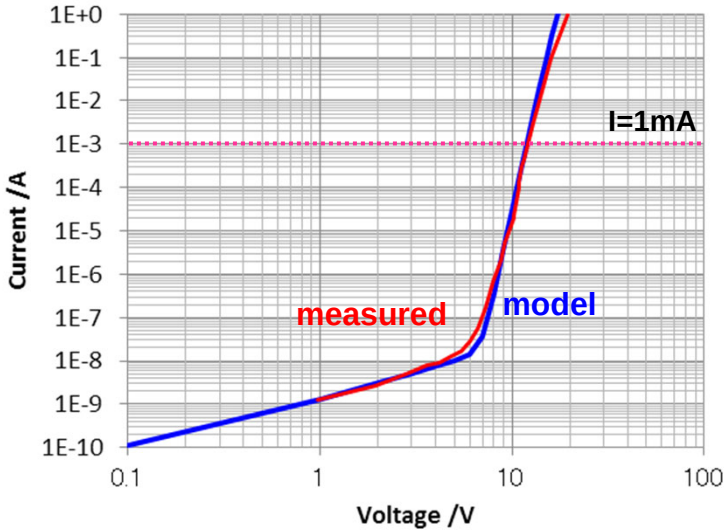
- **Chip varistor**

AVR-M1005C120MTAAB

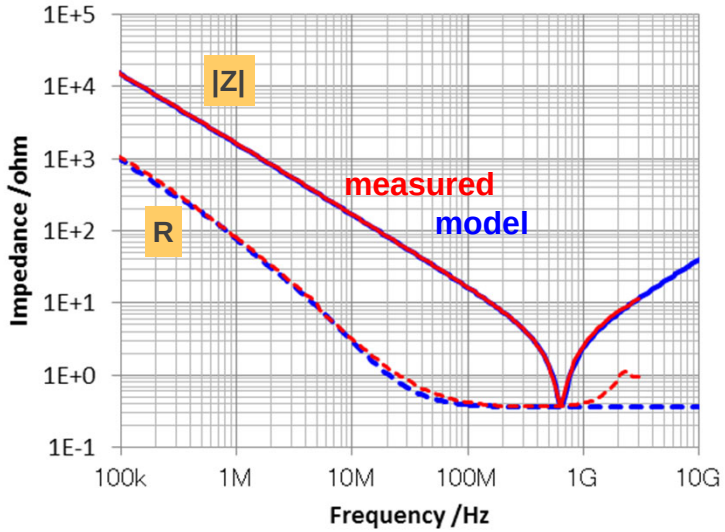


・varistor voltage V1mA=12V
・capacitance C=130pF

voltage-current property

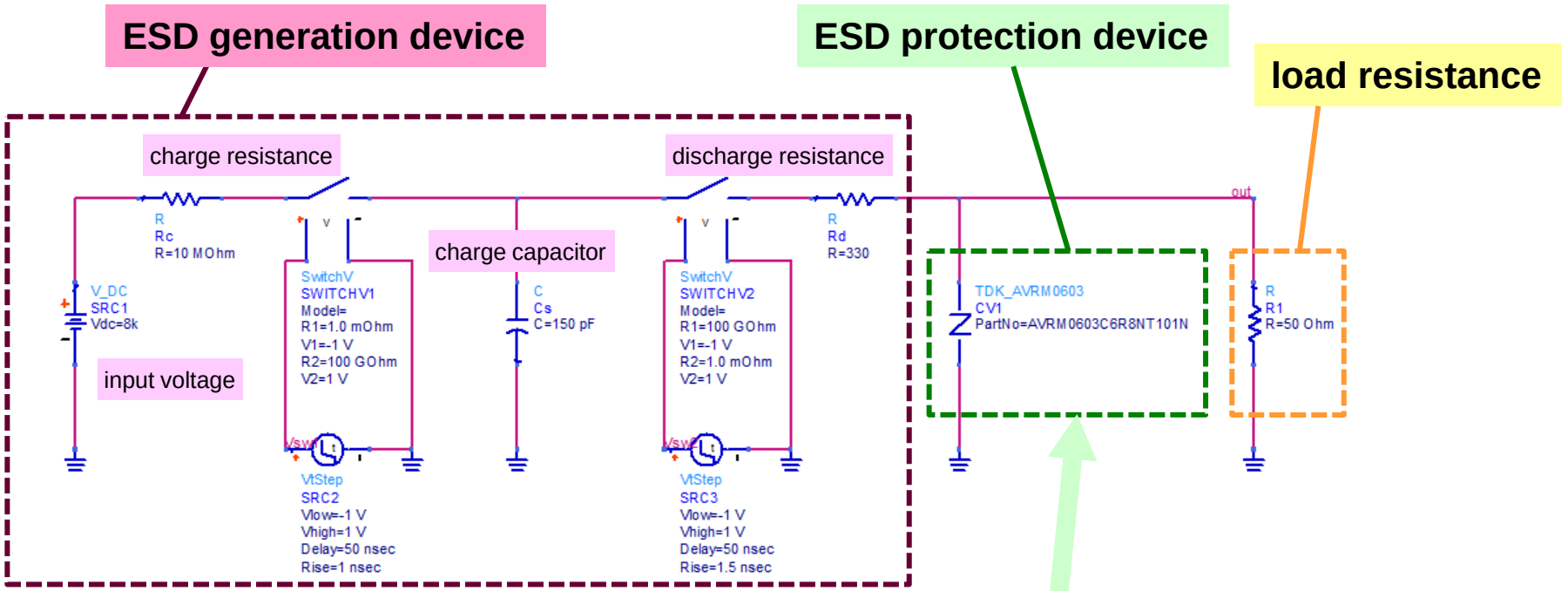


frequency property



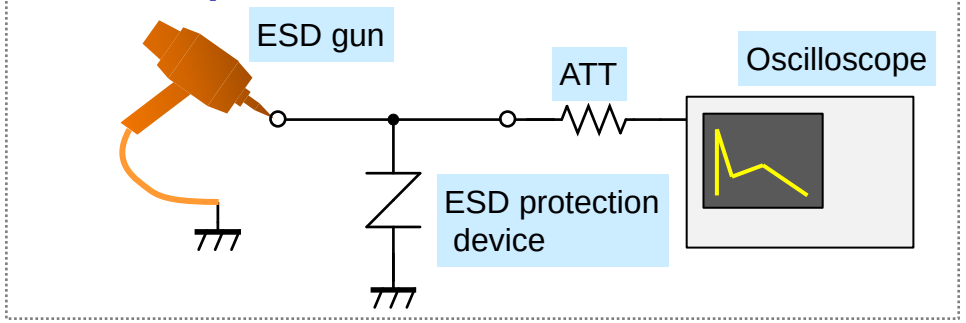
Both the voltage-current property and the frequency property are modeled.

simulation circuit for ESD absorption



chip varistor
voltage-current model
※included in the component library for ADS

the setup for measurement



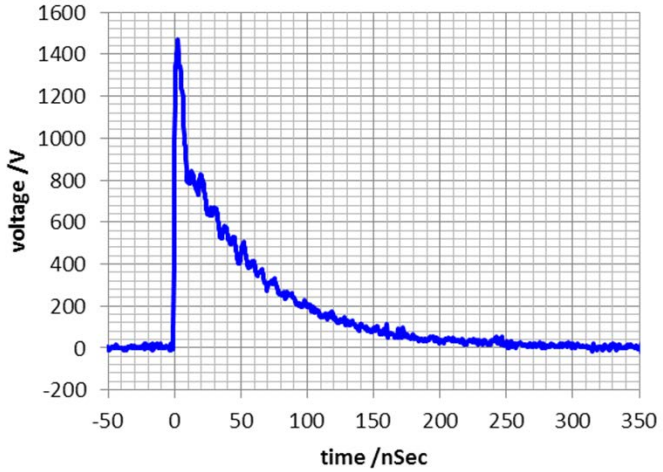
simulated results for ESD absorption

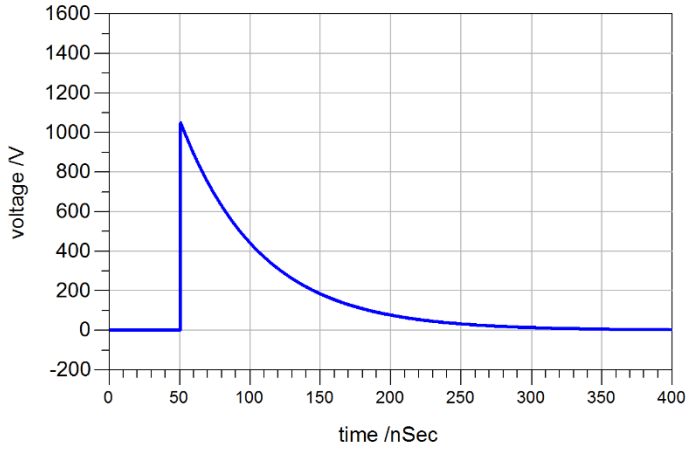
Attracting Tomorrow

measurement

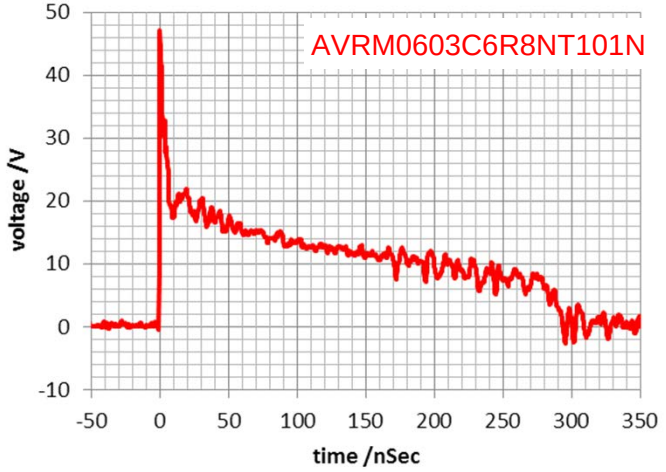
simulation

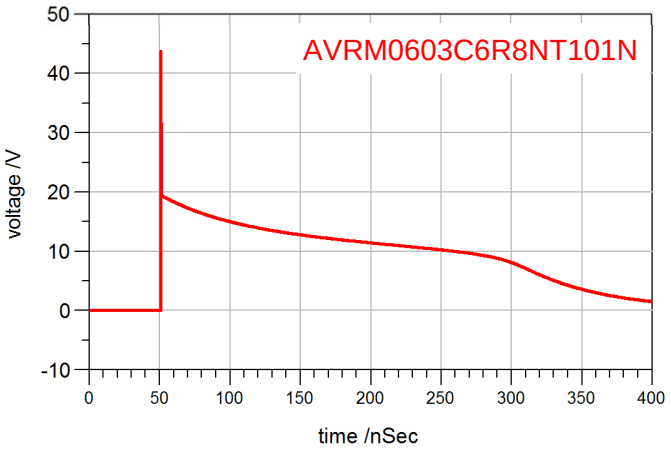
without protection device





with chip varistor





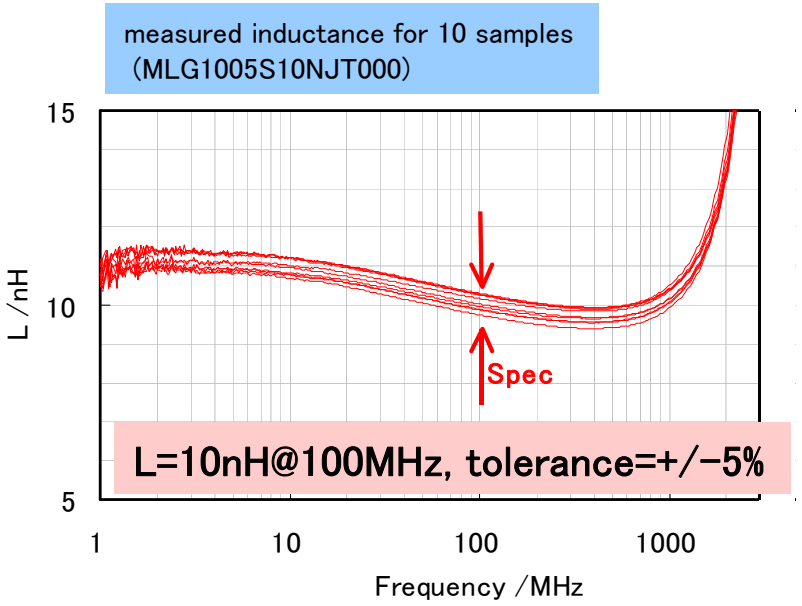
ESD absorption property can be simulated using the voltage-current model.

About the property tolerance model

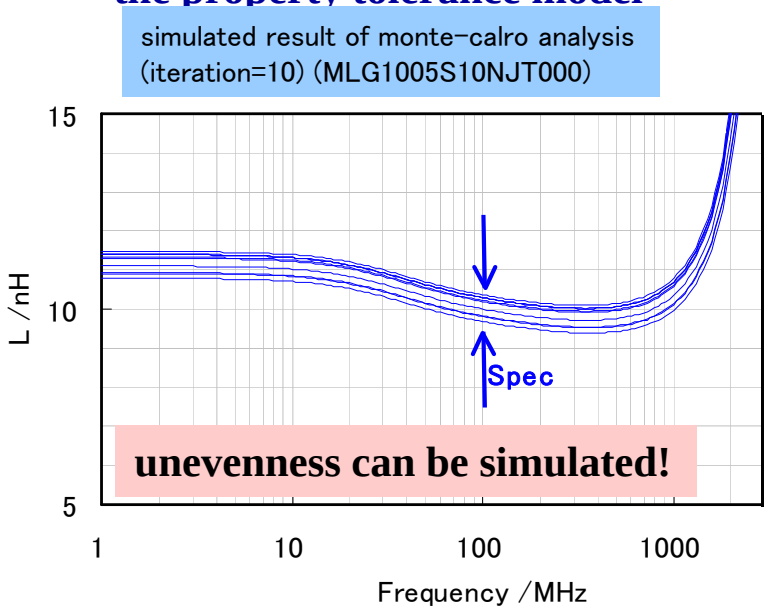
- **Abstract of the property tolerance model**

  The property tolerance model is a new type of simulation model that includes electrical tolerance data of products. The property of actual electric parts have unevenness within tolerance of the product due to unevenness of manufacturing. Simulated result of this model varies at random within the tolerance of the product. Hence, the model can be used in worst case analysis of your circuits.

unevenness of property for an inductor



simulated results of the property tolerance model

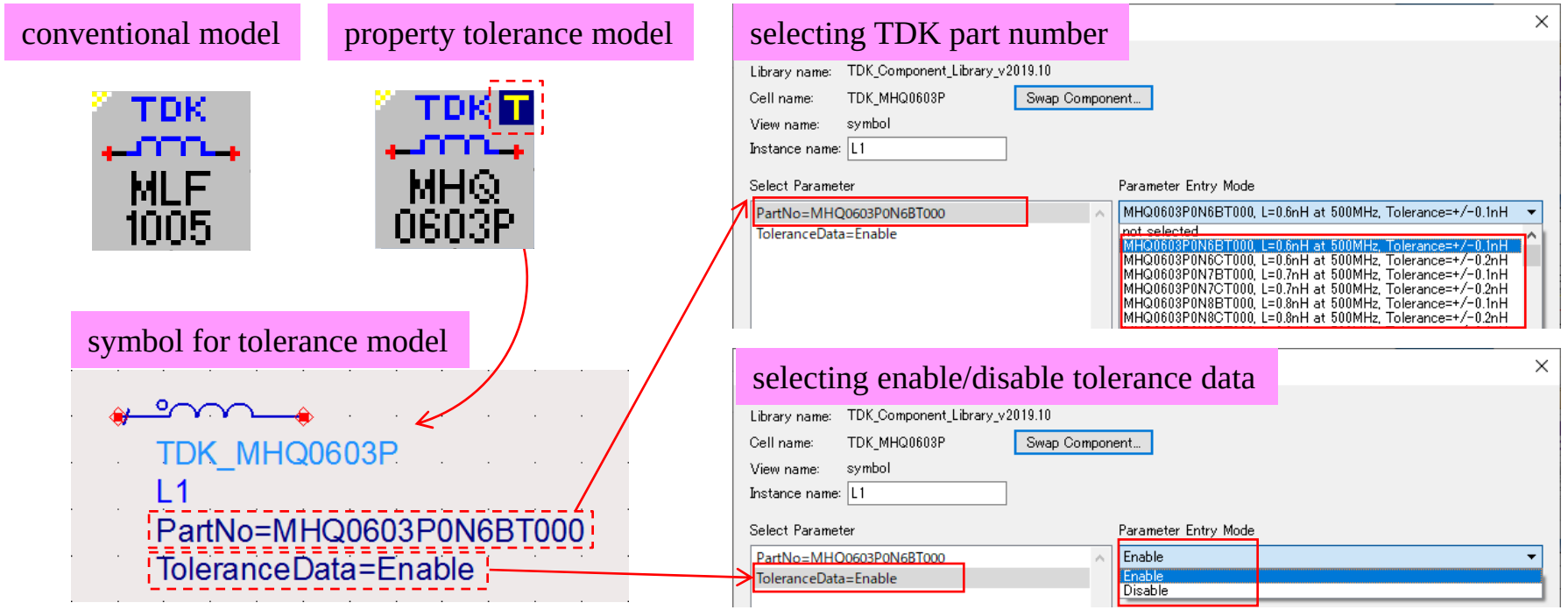


property unevenness of electronic parts can be taken into account in the circuit simulation

How to use the property tolerance model (1)

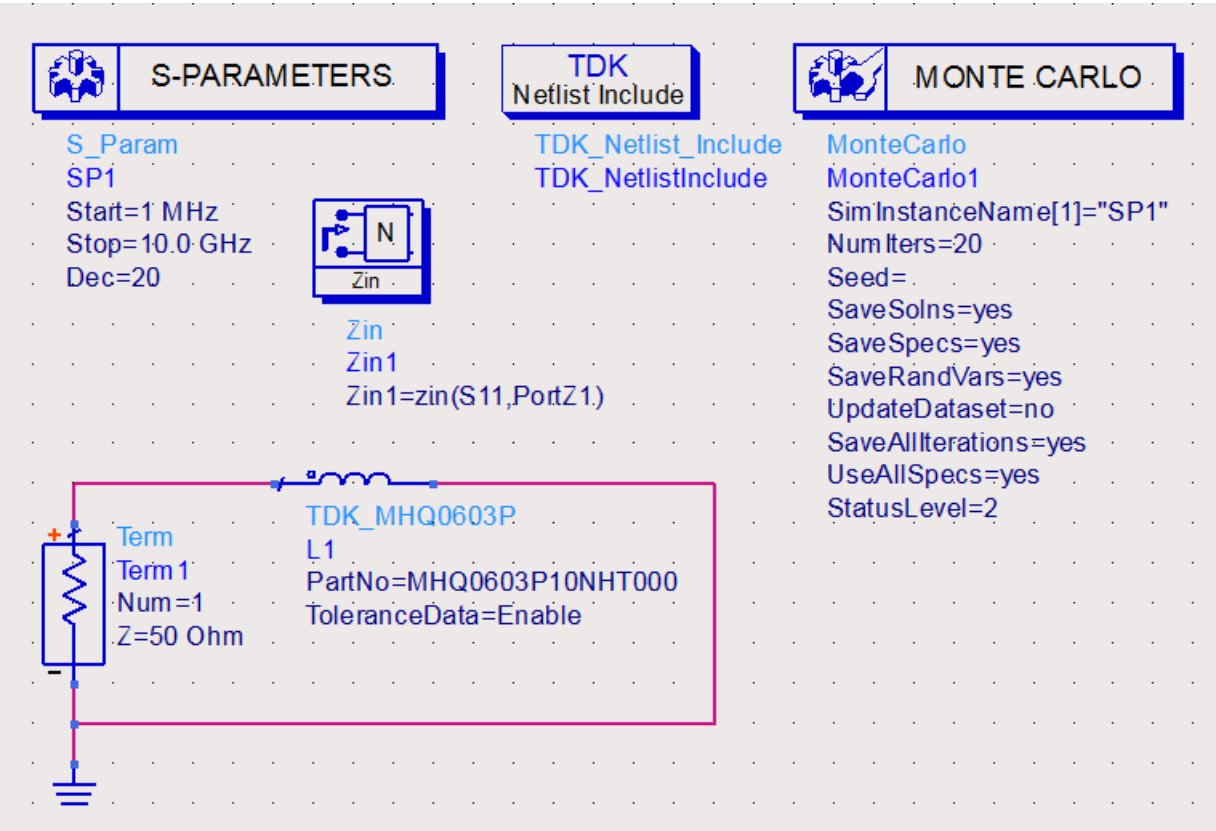
- **Putting and setting the component**

  The property tolerance model is provided for temperature compensation type multilayer ceramic capacitors and inductors for RF circuits. (The component that has “T” mark in the icon is the property tolerance model.) Click the component icon of the property tolerance model, and put a symbol on the schematic window. Double-click the symbol, then TDK part No. and use/no-use of tolerance data can be selected in the setting window. If ToleranceData=Disable, the model works just same as the conventional model that shows typical property of product.



How to use the property tolerance model (2)

- **Setup of Monte Carlo analysis**
  In order to simulate property unevenness, the Monte Carlo analysis function is used. The following is an example of circuit to simulate unevenness of impedance property.




TDK