NL5 Circuit Simulator

User's Manual

Rev. 1.61

Rev. 1.61 11/01/2010 nl5.sidelinesoft.com

VERSION

This version of User's Manual is current for NL5 version 1.6, build 1 (11/01/2010).

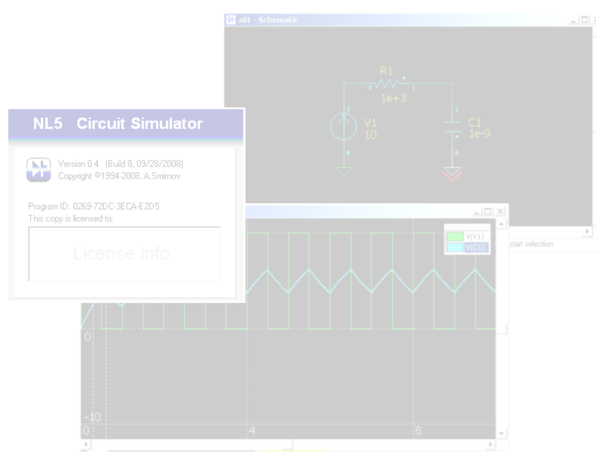
LIMITED LIABILITY

NL5, together will all accompanying materials, is provided on a "as is" basis, without warranty of any kind. The author makes no warranty, either expressed, implied, or stationary, including but not limited to any implied warranties of merchantability or fitness for any purpose. In no event will the author be liable to anyone for direct, incidental or consequential damages or losses arising from use or inability to use NL5.

COPYRIGHTS

Table of Contents

I. Quick Start



Install and Run NL5

Install NL5

One of NL5 advantages is that it consists of only one file: `nl5.exe`. NL5 does not require special installation, simply copy `nl5.exe` into any directory. You may have several copies of `nl5.exe` in different directories. To move NL5 to another computer, just copy `nl5.exe`, that's all.

The following files are not required, but, if used, should be located in the same directory as `nl5.exe`:

- `nl5.chm` – NL5 help file.
- `nl5.nll` – NL5 license file.

To create NL5 icon in the desktop, **right-click** on `nl5.exe` in the Windows Explorer and select **Send To | Desktop (create shortcut)** command.

Latest revision of NL5 can be found at nl5.sidelinesoft.com.

Run NL5

To run NL5 double click on `nl5.exe` or on the NL5 icon: . "Splash screen" window with version, date, and license information will show up:

The window disappears in a few seconds.

NL5 can also be started from command line, with or without switches and parameters.

Switches. Switch is a text starting with '-' or '/' symbol. The following switches can be used in the command line:

`-http` - start HTTP server.

For example:

```
>nl5.exe -http
```

Parameters. Parameter is a file name. One or more file names of different types can be used as parameters. For example:

`>nl5.exe rc.nl5`	– download schematic `rc.nl5`
`>nl5.exe tran.nlt`	– download transient data from `tran.nlt`
`>nl5.exe rc.nl5 pref.nlp`	– download schematic `rc.nl5` and preferences from `pref.nlp`

File with "txt" extension being used as a parameter in the command line is considered to be a script. Script will be executed immediately. For example:

`>nl5.exe script.txt`	– download and run script from `script.txt`

NL5 License

Without a license, NL5 operates as a **Demo version**. Demo version has all full-function features available, however the total number of components in the schematic is limited to 20. Although Demo version does not allow entering components above the limit, it still can read and simulate schematics with unlimited number of components, created by licensed version of NL5.

Several full-function license types with different limitations and protection methods can be ordered at nl5.sidelinesoft.com. All licenses (except Portable) are using license file `nl5.nll`. This file should be located in the same directory as application file `nl5.exe` (or `NL5LicenseServer.exe` for network license).

Single PC License

Single PC License provides full functionality on single PC only. The license is tied to specific PC hardware information ("PC fingerprint"). When NL5 starts, it compares "fingerprint" information stored in the license file with current PC information, and runs successfully only if "fingerprints" are identical. The PC "fingerprint" can be obtained in the **Support** dialog box (**Help | Support**), and then entered (pasted) into "fingerprint" window on the license ordering form.

Single PC License is offered free of charge for limited period of time as a **Trial License**. After trial period expired, **Permanent** Single PC License can be purchased. Please visit nl5.sidelinesoft.com for details.

Portable License

Portable License uses USB device - **dongle** - to store license information. NL5 operates as full-function version on any PC with the dongle connected to the USB port. Due to "driverless" dongle technology no problems using USB ports on different PC models are expected.

Personal License

Personal License allows unlimited non-expirable use of NL5 on any PC. The license is issued to a person, and can be installed only on computers where the license owner is using NL5 on regular basis.

Network License

Network License is installed on the "License Server" – a computer running a Windows operating system. The license consists of the `NL5LicenseServer.exe` Windows application and network license file `nl5.nll`. NL5 can work on any computer that has access to the server computer through the network. When NL5 starts, it obtains license information from the server. The number of simultaneously running NL5 applications (number of "seats") is unlimited.

Create and Simulate Your First Schematic

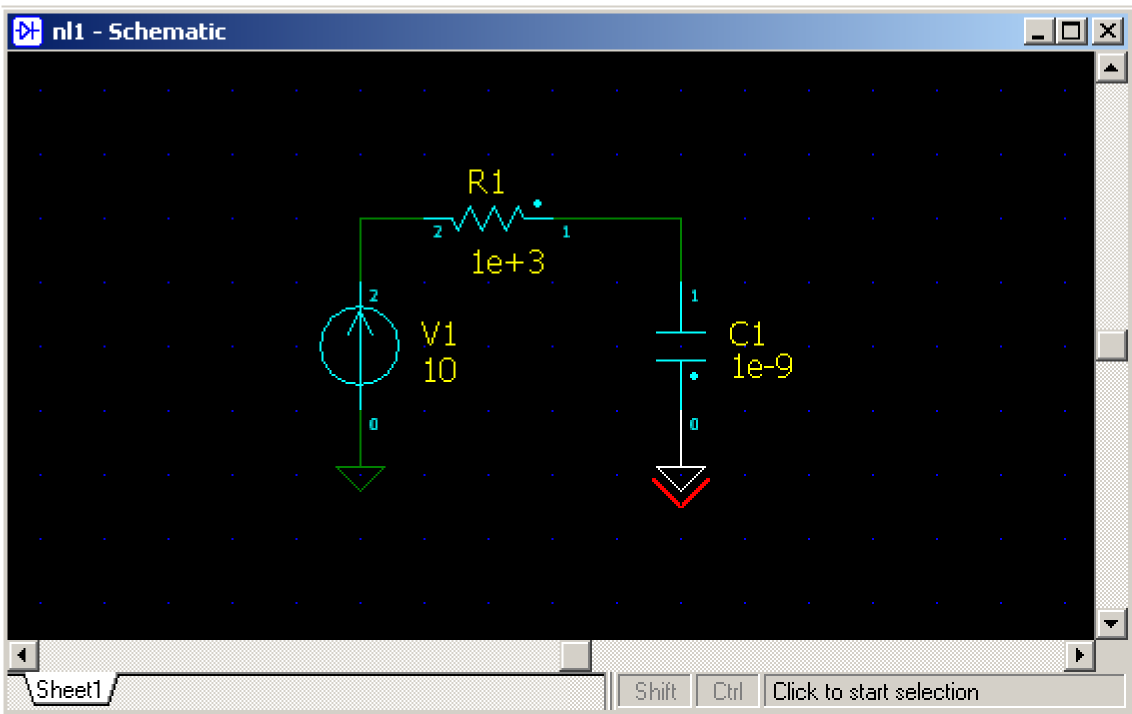
Enter schematic

Entering and editing can be done using keyboard keys, mouse, or both. Here are step-by-step instructions how to enter simple schematic using keyboard.

When NL5 opens, an empty schematic is created. A red cursor is located in the middle of the screen and is directed to the right.

- Press **Space** to switch to drawing mode.
- Press **Arrow Down** several times to draw short wire downward.
- Press **V** key and then press **Enter** to place a voltage source.
- Press **G** key to place a ground. Now cursor is switched back to selection mode.
- Press **Arrow Up** several times to move cursor back to the starting point.
- Press **Arrow Right** to change direction; then press **Space** to switch to drawing mode.
- Press **Arrow Right** several times to draw a short horizontal wire.
- Press **R** key and then press **Enter** to place a resistor.
- Press **Arrow Right** several times again; then press **Arrow Down** several times.
- Press **C** key and then press **Enter** to place a capacitor.
- Press **G** key to place a ground. Schematic is ready.

Here is what you are expected to see:

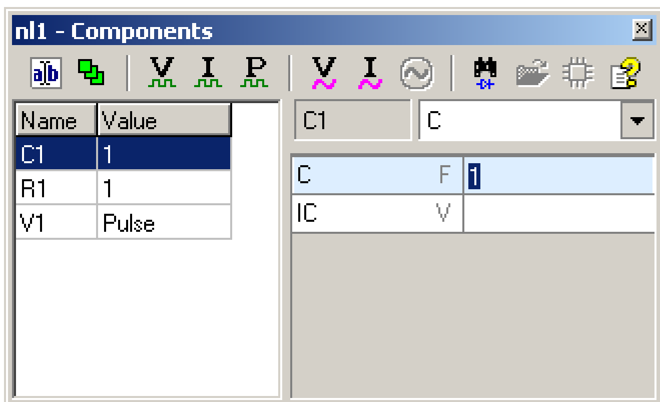


Edit component parameters

Now you will use mouse to select a component and keyboard to enter parameters.

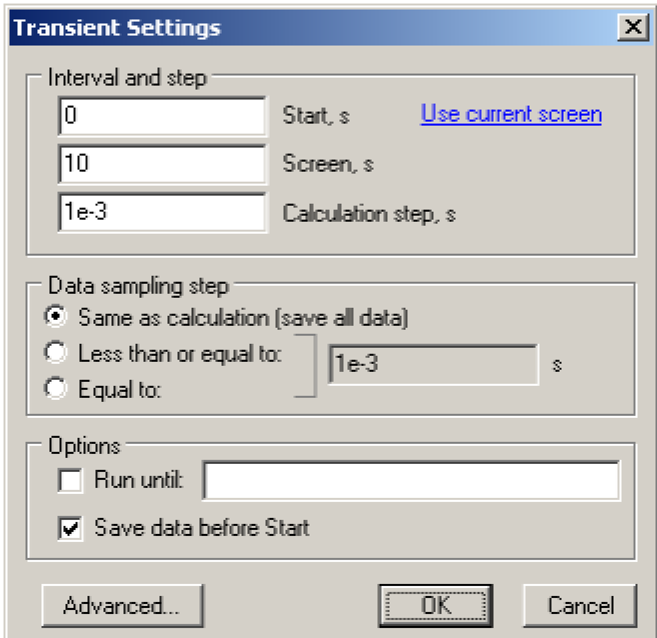
- Double click on the voltage source **V1**. The **Components** window will show up. On the left pane V1 should be selected. Right pane shows component name (**V1**), model (**V**) and parameters (just one parameter, "V").
- Click  button right to the model name. A drop down window will show available models for the voltage source.
- Select **Pulse**.
- Double click on the resistor **R1** on the left pane. A resistance value "1e+3" will be selected on the right pane.
- Press **1 ("one")** key, resistance will change to 1 Ohm.
- Double click on the capacitor **C1** on the left pane, then change capacitance "1e-9" to "1".

Components are ready. Here is a result:



Transient settings

Click **Transient settings** Toolbar button, or select **Transient | Settings** command in the Main Menu. You don't need to change anything here, but you can, if you wish. Click **OK** button.

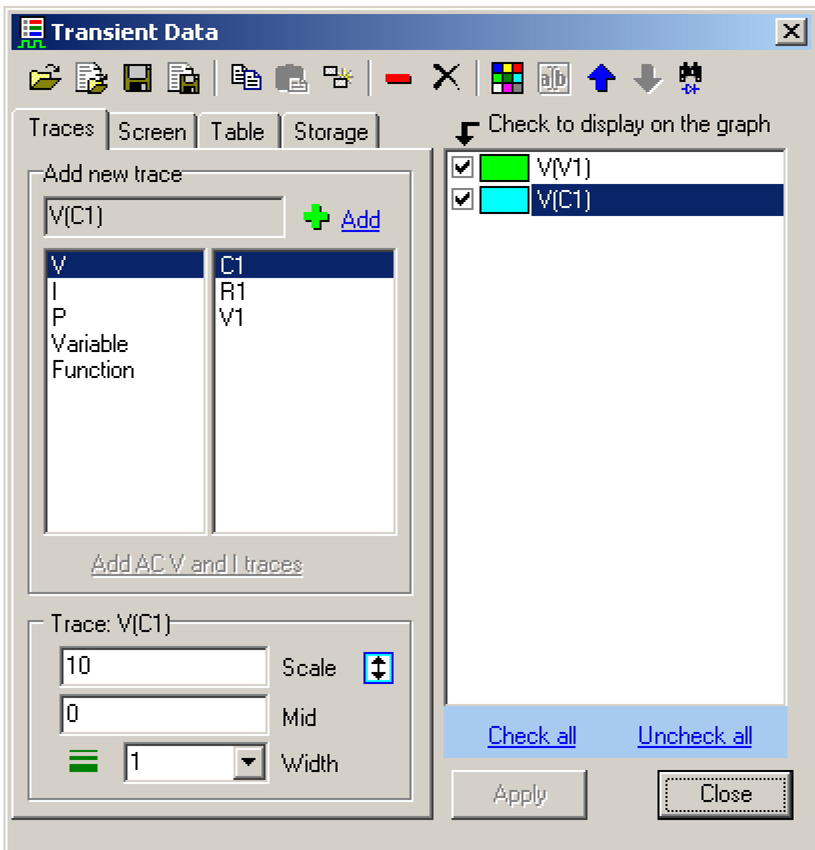


Transient data

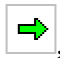
Click **Transient data** Toolbar button, or select **Transient | Data** command in the Main Menu. Make sure **Traces** tab is selected.

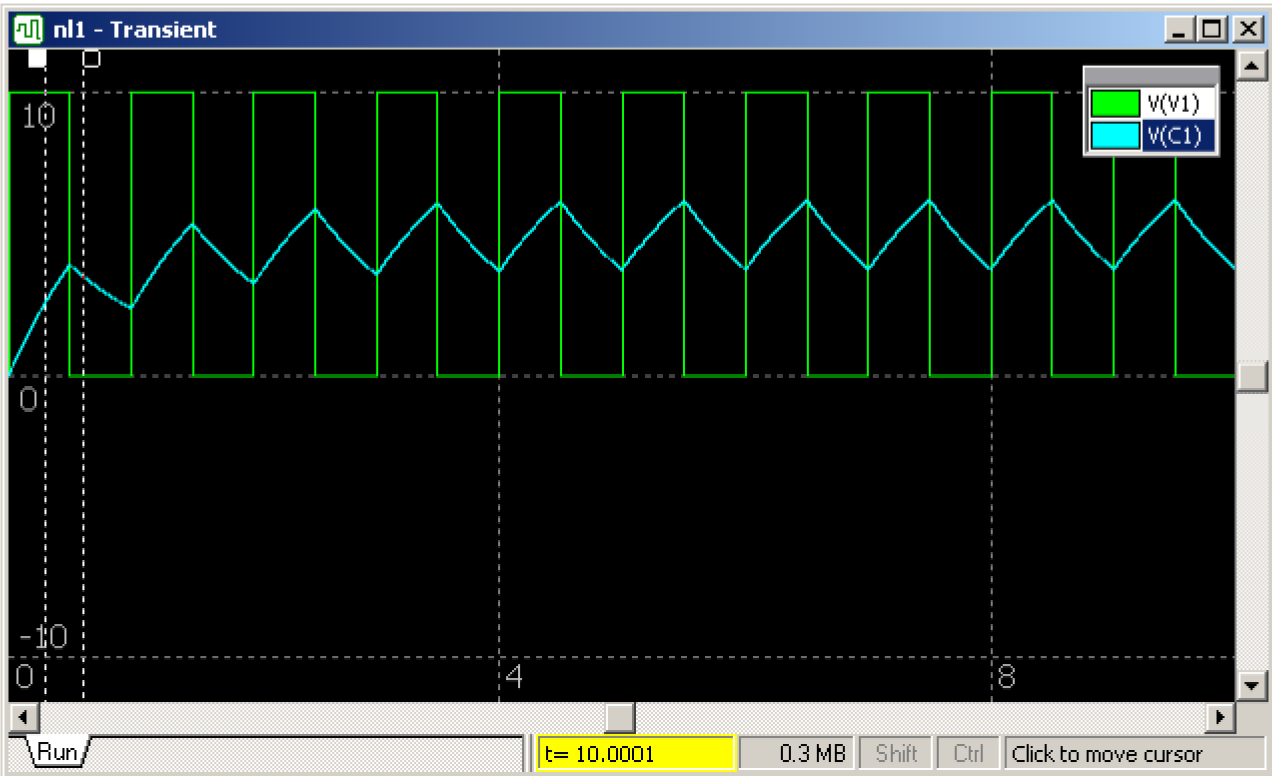
- In the **Add new trace** box select **V** (voltage trace).
- Double-click on **V1** and **C1** in the components list. Voltage traces will be added to the traces list.
- Click **Close** button.

Here is window view (before closing):



Run transient

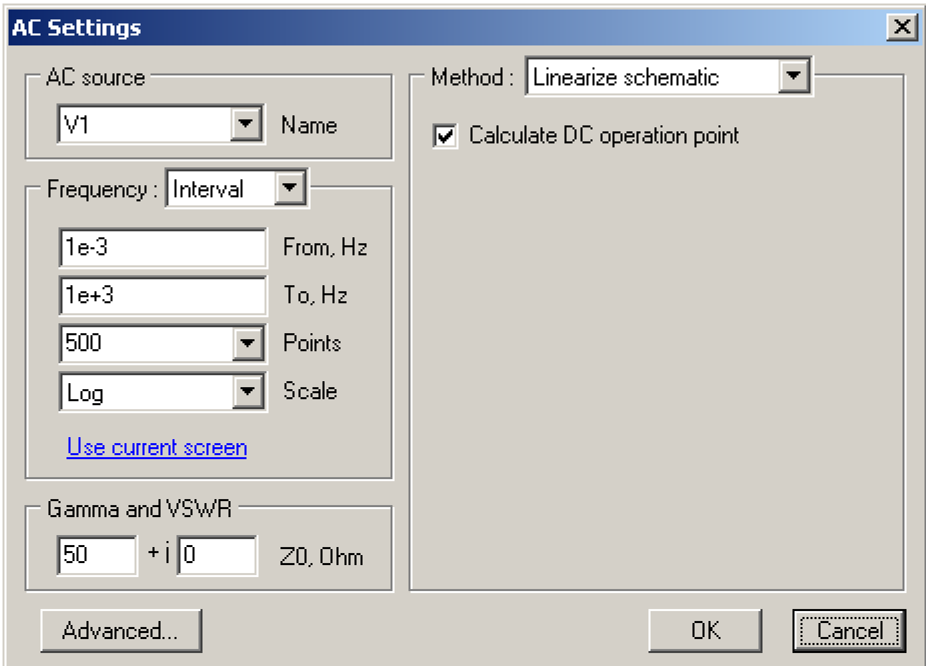
Click **Start transient** Toolbar button , or select **Transient | Start** command in the Main Menu. A transient will be calculated and displayed:



AC settings

Click **AC settings** Toolbar button , or select **AC | Settings** command in the Main Menu.

- Click on the **Name** drop-down list in the **AC source** box and select **V1**.
- Click **OK** button.

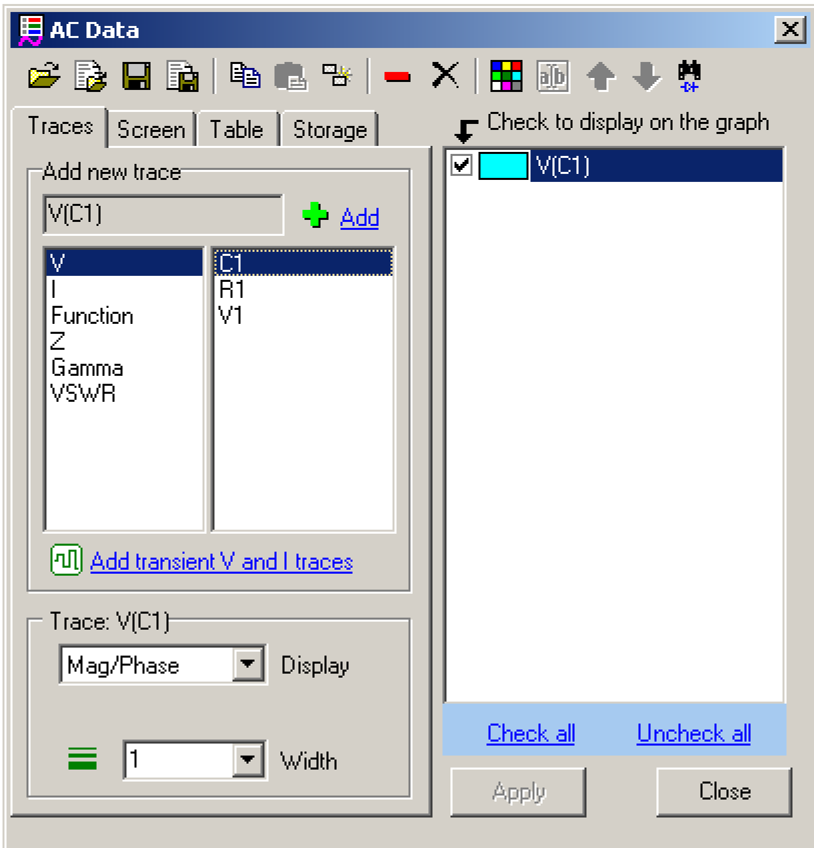


AC data

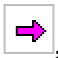
Click **AC data** Toolbar button , or select **AC | Data** command in the Main Menu. Make sure **Traces** tab is selected.

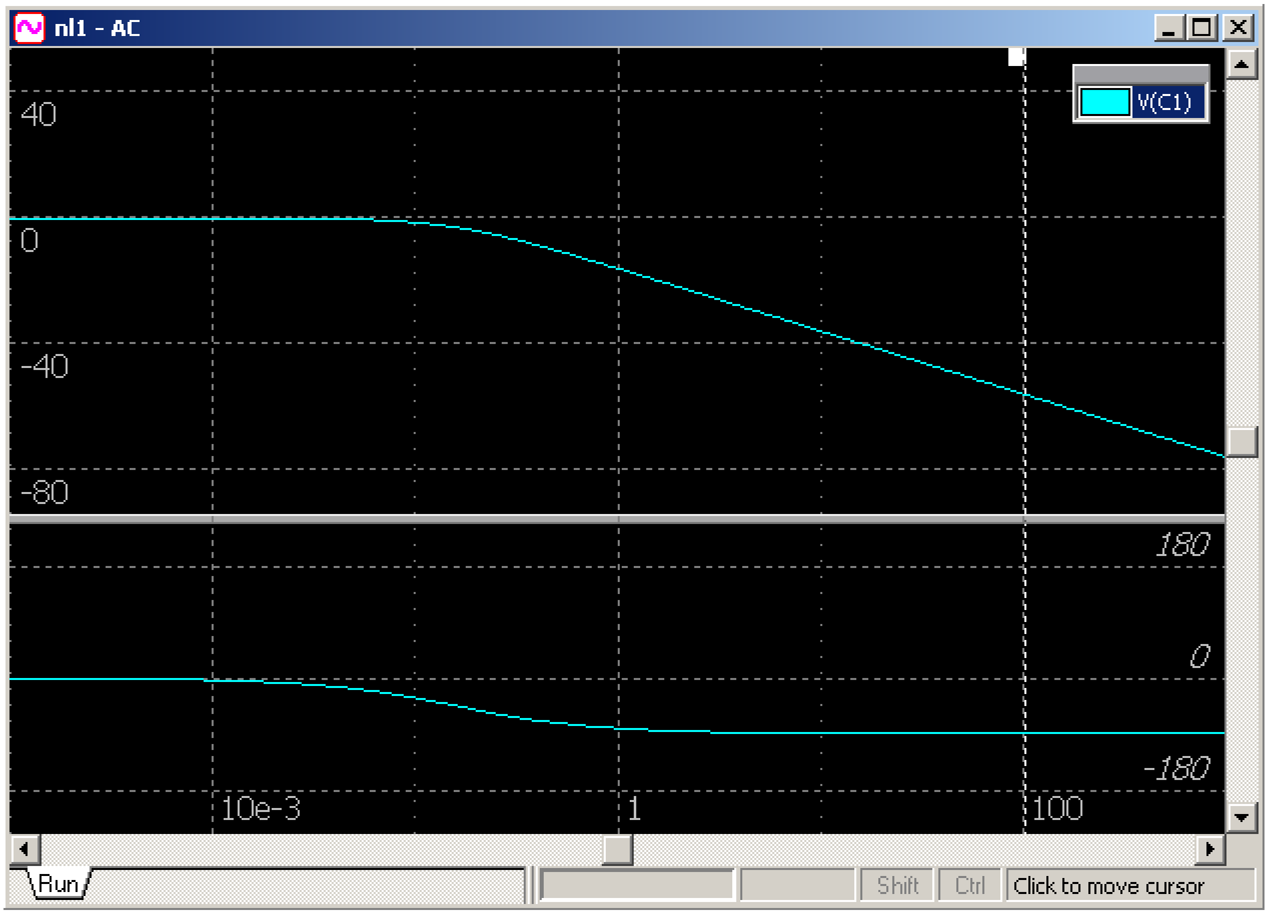
- In the **Add new trace** box select **V** (voltage trace).
- Double click on **C1** in the components list. AC voltage trace will be added to the traces list.
- Click **Close** button.

Here is window view (before closing):

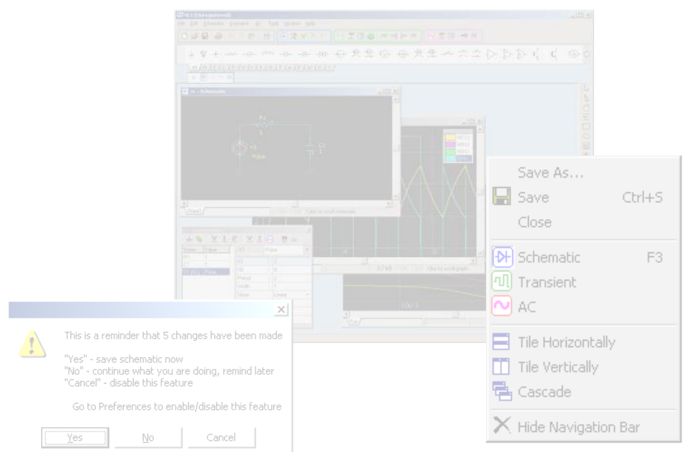


Run AC

Click **Start AC** Toolbar button , or select **AC | Start** command in the Main Menu. An AC response will be calculated and displayed:



II. User Interface



Data format used in NL5 mostly complies with common engineering and scientific practice. It makes it easy to learn and use by any person, familiar with other popular tools. NL5 is using several dedicated **File Types** for schematic and analysis data. NL5 **Graphical User Interface** (GUI) is based on standard Microsoft Windows Multi-Document Interface (MDI) architecture. It consists of different interface components, such as windows, dialogs, menus, toolbars, etc. NL5 supports many commands and shortcuts that are commonly used in Windows applications, for instance: **Edit | Copy** (**Ctrl-C**), **Edit | Paste** (**Ctrl-V**), **Window | Tile,** using of **Ctrl** key along with mouse for select/copy operation, using window scrollbars, etc. Other commands are very intuitive, so that it would not take long to start working with schematic. **Preferences** are used to customize application "look and feel" and default parameters. **Printing** allows convenient layout and formatting of windows to be printed.

Data format

Case-insensitivity.

All the text data in the NL5, such as component names, variables, functions, commands, etc., are **case-insensitive**, unless otherwise stated. Lower case and upper case letters are considered the same. For example:

```
Rin = RIN = rin
sin(45) = SIN(45)
```

Numbers

Number format in NL5 is very flexible and complies with many commonly used styles and standards. A number can use exponential multipliers **E** or **e**, and **case-sensitive** letter multipliers:

Letter	Multiplier
T	10^{12}
G	10^{9}
M, mg	10^{6}
k, K	10^{3}
m	10^{-3}
u, mk	10^{-6}
n	10^{-9}
p	10^{-12}

For example:

```
1.3e+3   47E-9   100k   0.33mk   2.2M
```

Letter multiplier can be followed by any text, which is considered as units and will be ignored:

```
1.3kOhm   47nF   0.1mkH   333ps
```

Any text that does not begin with letter multiplier is considered as units and will be ignored:

```
1.3Ohm   0.001F   0.1H   333apples
```

Letter multiplier and units (with or without letter multiplier) can be used instead of a decimal point:

```
1k3   5n6   3nF3   47F0  2s2
```

Zero before decimal point or letter multiplier can be omitted. For example:

```
.47   n47   uF5
```

A number can be entered in decibels, using suffix db (case insensitive). It will be automatically converted to a standard number (allowed for positive numbers only):

```
20db = 100
3.01dB = 1.41416472507
6DB02 = 1.99986186963
```

Infinite value is denoted by:

```
inf
```

Imaginary part of a complex number has **lower case** letter 'j' at the end of a number. Letter 'j' cannot be used alone, only as a suffix:

```
50+45j
1+1e-3j = 1+.001j
30j
1+j  - wrong! Correct format: 1+1j
```

The following predefined constants (case insensitive) can be used in expressions:

```
PI = Pi = pi = 3.14159265359
RAD = Rad = rad = 180/pi = 57.2957795131
LOW = Low = low – low logical level, V
HIGH = High = high – high logical level, V
```

Constant `RAD` can be used to convert degrees to radians and radians to degrees:

```
Degrees = Radians*RAD
Radians = Degrees/RAD
```

where `Degrees` is value in degrees, and `Radians` is value in radians.

Being entered in a variety of formats available, a number is automatically converted and stored in the floating point (**double**) format. When number is displayed, an engineering notation, with exponential multiplier and power of ten to be multiple of three, is used:

Entered	Displayed
1k3	1.3e+3
47e-8	470e-9
5600000	5.6e+6

Names

Component. When a new component is created, it is assigned a default name: 'letter' plus number:

```
R1, V2, C123
```

Then the component can be renamed. The name is case-insensitive and may contain any characters and symbols. When used in formula or function, the name is enclosed in quotes:

```
"R out", "V pulse", "+12V"
```

However, if the name starts with letter and contains letters and numbers only, it can be used without quotes:

```
Rout, V123, Plus12V
```

If the component has been renamed, its name will be automatically modified in all appearances of the component name in trace names, formulas and functions.

To access component's parameter in the formula, function, or script expression, use component name followed by dot '**.**' and parameter name:

```
R1.R, V2.slope, C123.IC, "R out".R
```

If parameter name is not specified, a first parameter of the component will be used:

```
R1 = R1.R
C2 = C2.C
```

To access component which is part of the subcircuit use subcircuit component's name followed by dot '**.**' and component's name in the subcircuit. A nesting level is unlimited: components inside subcircuit, which in turn is part of subcircuit, can be accesses by similar notation:

```
X1.R2
X1.F1.V3.period
```

where `X1` and `F1` are subcircuits.

To access component's model name (in a script or command-line) use component name followed by dot '**.**' and "model":

```
V1.model=pulse
```

Variable. Variable name has the same format as a component, except it does not have parameters. For example:

```
Freq, "max limit", X1.var
```

Please note that this applies only to a **schematic** variable: a variable defined in the **Variables window**. Variables used in C code (script, Code component) comply with C language standard.

Trace. The basic name of transient or AC trace that holds current simulation data consists of the letter specifying type of the trace (**V**, **I**, **P**), followed by component's name in parentheses:

```
V(R1), I(C2), P(L3)
```

The trace with basic name can't be renamed. When trace is duplicated, loaded from file, or pasted, it can be renamed to an arbitrary text:

```
"Copy of V(R1)"
"Old trace of R1"
"V pulse"
```

The name of the trace of **Function** type is the function itself. Renaming the trace will change the function:

```
"V(r1)*V(r1)/r1"
```

Trace and cursors data. Trace and cursors data shown in the transient or AC table (value at cursor, min, max, mean, etc.) and trace value at specified time can be used in the script expressions. The trace should be added to the Transient or AC Data, but does not need to be displayed on the graph or in the table. To access trace data use the following notation:

- **Cursors (screen)**

`left`	– position of the cursor, or left edge of the screen
`right`	– position of right cursor, or right edge of the screen
`delta`	= `right-left`

- **Transient amplitude**

`V(R1).(1.2)`	– value at t=1.2
`V(R1).left`	– value at the left edge of the screen or left cursor
`V(R1).right`	– value at the right edge of the screen or right cursor
`V(R1).delta`	= `V(R1).right-V(R1).left`
`V(R1).min`	– minimum
`V(R1).max`	– maximum
`V(R1).pp`	– peak-to-peak (max–min)
`V(R1).mean`	– mean
`V(R1).rms`	– RMS
`V(R1).acrms`	– RMS of the signal with subtracted mean value

- **AC magnitude**

`V(R1).(1.2)`	– value at f=1.2
`V(R1).left`	– value at the left edge of the screen or left cursor
`V(R1).right`	– value at the right edge of the screen or right cursor
`V(R1).delta`	= `V(R1).right-V(R1).left`
`V(R1).min`	– minimum
`V(R1).max`	– maximum
`V(R1).pp`	– peak-to-peak (max–min)

`V(R1).slope` – slope of the gain, `dB/dec`

- **AC phase**

`V(R1).phase.(1.2)` – value at f=1.2
`V(R1).phase.left` – value at the left edge of the screen or left cursor
`V(R1).phase.right` – value at the right edge of the screen or right cursor
`V(R1).phase.delta` = `V(R1).right-V(R1).left`
`V(R1).phase.min` – minimum
`V(R1).phase.max` – maximum
`V(R1).phase.pp` – peak-to-peak (max–min)

Operators

NL5 supports the following arithmetic and logical operators:

```
=        &&       ==       %
+=       ||       !=       ^
-=       <        +        ++
*=       <=       -        --
/=       >        *
?:       >=       /
```

See full operators list, description, and examples in the **Attachment 2.**

Functions

NL5 offers many standard and NL5-specific mathematical functions. For the convenience of users, there may be several names used for the same function (for example `log10` and `lg`), so that the user can use the name he or she is more comfortable with. The following functions are available:

```
sin        exp         mag, abs     im
cos        ln, log     sign         par
tan, tg    lg, log10   db           random, rand
sqrt       lb, log2    min          limit, lim
sqr        asin        max          islow
sq         acos        int, round   ishigh
pow        atan        phase
pwr        atan2       re
```

See full functions list, description, and examples in the **Attachment 3.**

Expressions

Expression may consists of:

- Numbers.
- Predefined constants (PI and RAD).
- Names of components, parameters and variables.
- Local script or C-code variables.
- Operators.
- Functions.
- Parentheses with unlimited nesting level.

For example:

```
2*2
2^10-1
sin(2*PI*f)          // "f" is a schematic variable
max(R1,R2,R3)
1/((R1+R2)*C1)
```

Please note that NL5 operates only with floating point (**double**) data type. Even if the number is entered as integer (without decimal point), it is converted to a floating point format. Arguments and results of all operators and functions are floating point as well. In addition, many operators and functions can be used with complex numbers.

Expression can be used instead of number in most entry fields in the dialog boxes, and for some component parameters. When **Enter** key is pressed, or **OK** or **Apply** button (if exists) is clicked, the expression will be immediately evaluated and replaced with the numerical value.

C language

Simplified C-language is implemented in the NL5. It is used in the script and in the **C** model of **Code** component. Although not all standard C features are supported, it is quite sufficient for many tasks. Briefly, the difference between NL5 implementation and standard C is the following:

- The only supported data type is **double.**
- bitwise operators not supported.
- Structures and unions not supported.
- Pointers not supported.
- **goto** statement not supported

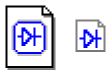
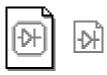
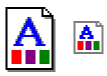
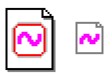
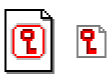
The following keywords, statements, and operators are available:

```
double          default         continue
if…else         for             break
switch          while           return
case            do…while
```

C operators and statements with examples are listed in the **Attachment 4.** Please refer to publicly available resources for general C language syntax description and reference.

File types

There are several file types registered for NL5. Each type has designated extension and icon.

Icon	Extension	Description
	`nl5`	`Schematic.`
	`nl5~`	`Schematic backup.`
	`nlp`	`Preferences.`
	`nlt`	`Transient data (binary).`
	`nlf`	`AC data (binary).`
	`nll`	`License file.`

If NL5 is started from command line, one or more files of those types can be used as parameters. For example:

`>nl5.exe rc.nl5` – download schematic `rc.nl5`
`>nl5.exe tran.nlt` – download transient data from `tran.nlt`
`>nl5.exe rc.nl5 pref.nlp` – download schematic `rc.nl5` and preferences from `pref.nlp`

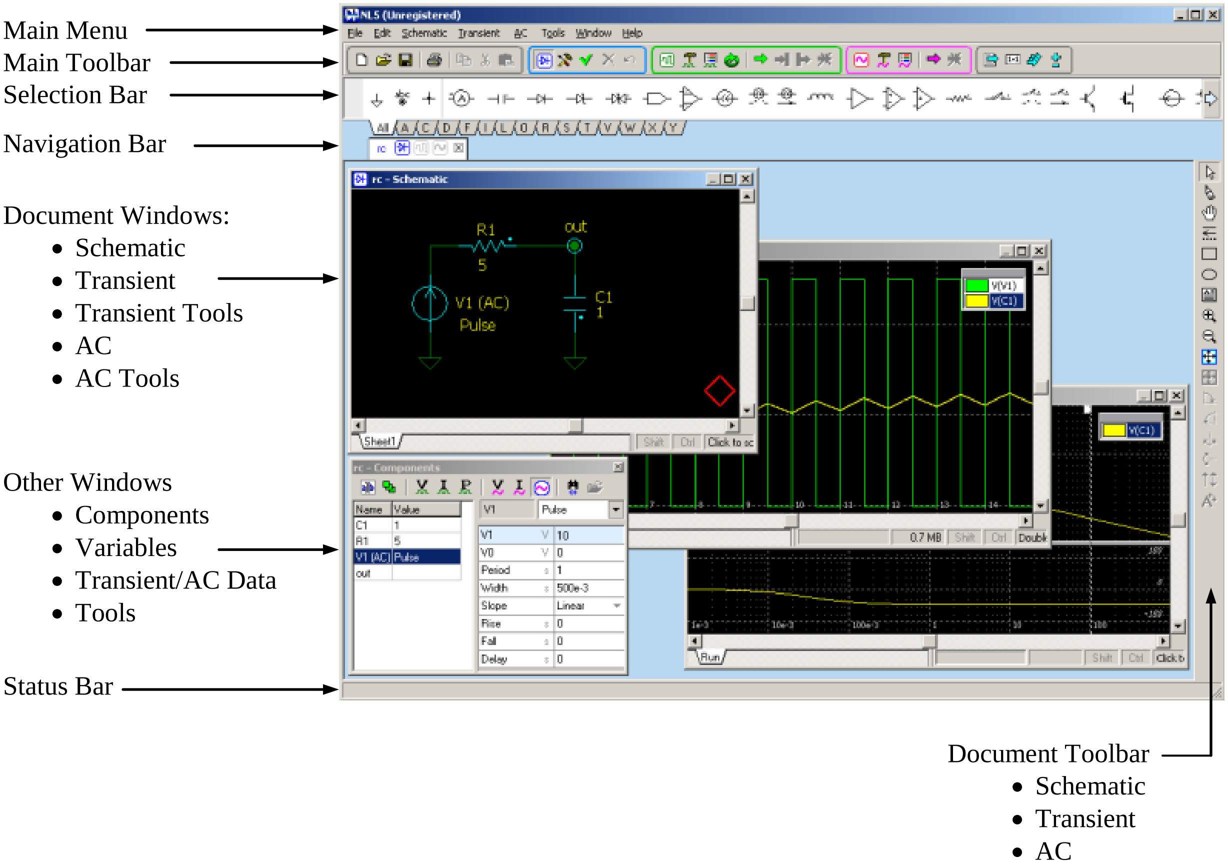
File with "txt" extension being used as a parameter in the command line is considered to be a script. Script will be executed immediately. For example:

`>nl5.exe script.txt` – download and run script from `script.txt`

Graphical User Interface

Main Window

NL5 Main Window and its components are shown below:

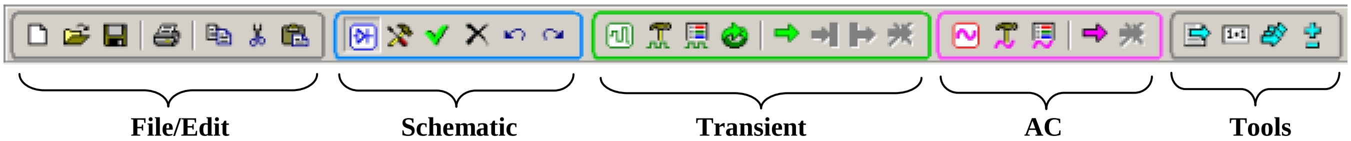


Main Menu

Main menu contains standard Windows menus (such as **File**, **Edit, Window, Help**), and NL5 specific (**Schematic, Transient, AC, Tools**).

Main Toolbar

Main Toolbar provides fast access to often used commands and contains 5 groups of buttons:



Move mouse pointer over the button to see a hint with button description.

Status Bar

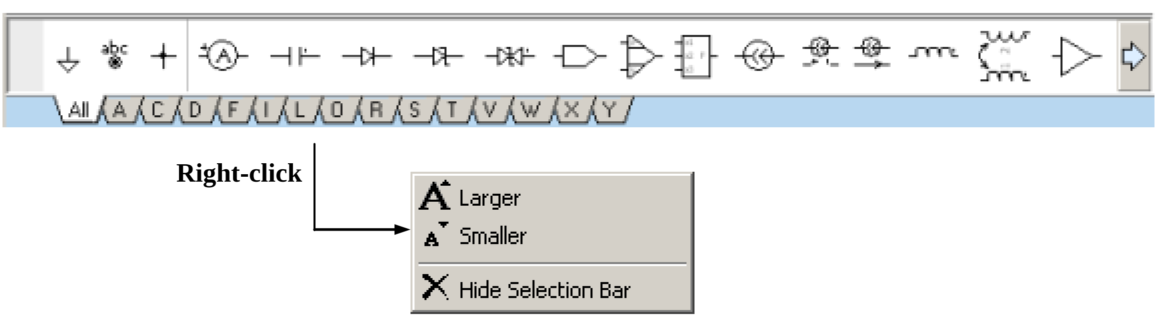
Status Bar shows some application-related messages, such as:

- Opening documents.
- Saving documents.
- Checking for updates.

Select **Window | Status Bar** menu command to show/hide Status Bar.

Selection Bar

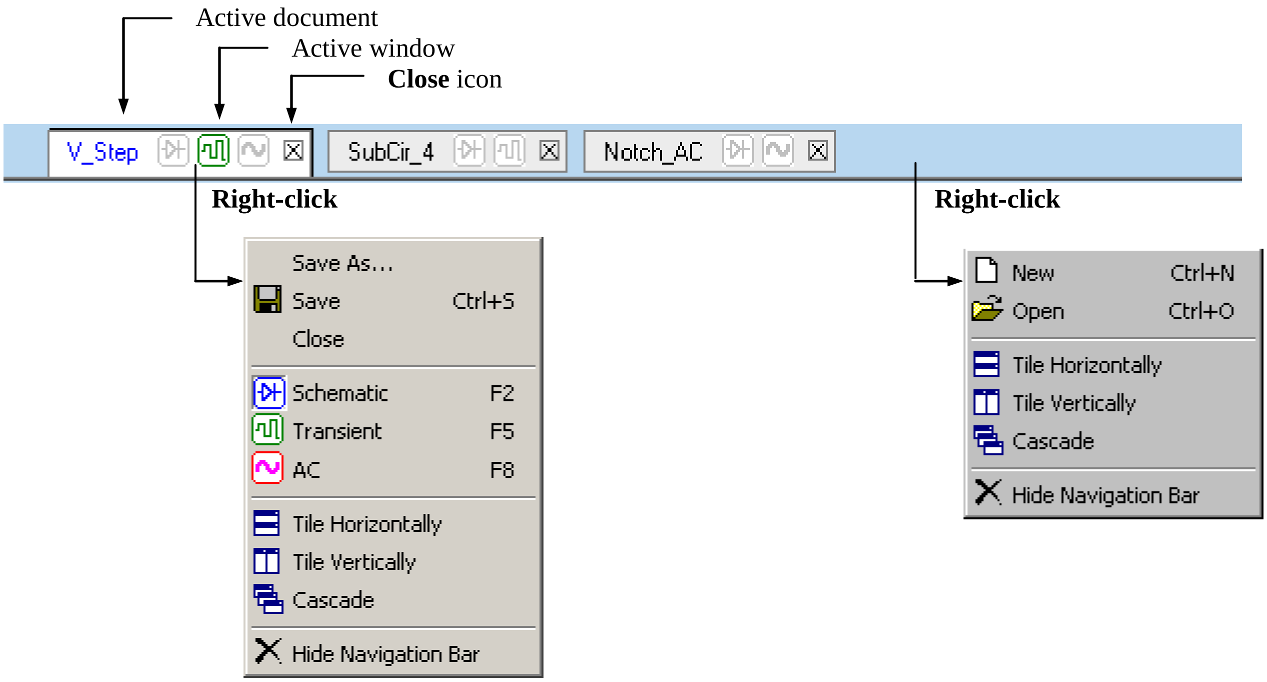
Selection Bar consists of tabs, one per letter (only if components are available for this letter). Each tab contains symbols of components of the "letter" type, and 3 common schematic elements: ground, label, and connection point. The tab "All" contains symbols of all components.



- If some components are not visible, click on left/right arrow images to scroll.
- Move mouse pointer over component symbol to see a hint with short description of a component.
- Click on the symbol to place component on the schematic.
- Right-click on the bar to see context menu with relevant commands.
- Select **Window | Selection Bar** menu command to show/hide Selection Bar.

Navigation Bar

Navigation Bar displays all opened documents and windows, and indicates active document and active window with highlighted icon.



- If some tabs are not visible, click on left/right arrow images to scroll.
- Move mouse pointer over icons to see a hint.
- Click on the tab to activate the document.
- Click on the window icon to activate the window.
- Click on the **Close** icon to close the document.
- **Right-click** on the document tab or empty space of the Navigation Bar to see context menu with relevant commands.
- Select **Window | Navigation Bar** menu command to show/hide Navigation Bar.

Document Windows

NL5 document may have several windows opened at the same time:

- Schematic
- Transient
- Transient Tools
- AC
- AC Tools

Schematic, Transient, and AC Windows are part of standard Multi-Document Interface, and basically behave similar to other Windows applications. Document Windows:

- Can be minimized and maximized.
- Can be arranged within Main Window (**Window | Tile**, **Window | Cascade**).
- Are listed under **Window** menu.
- Have a related Document Toolbar displayed at the right side of the Main Window.
- Closing Schematic window will automatically close the entire document.
- Use Navigation Bar or **Window** menu to navigate between these windows and arrange them on the screen.

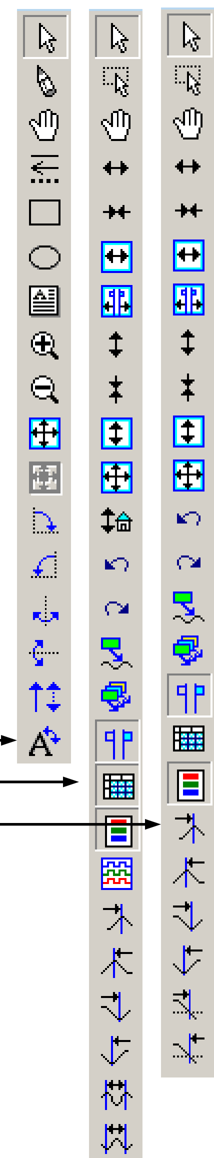
Transient Tools and AC Tools Windows are not part of the Multi-Document Interface and behave different. Tools Windows:

- Are always "on top" of other windows.
- Are listed under **Transient | Tools** and **AC | Tools** menus.

.

Document Toolbars provides fast access to commands related to active Document window. There are 3 types of Document toolbar:

- Schematic toolbar
- Transient toolbar
- AC toolbar



Only one toolbar corresponding to active Document Window is visible at a time.

Other Windows

Other Windows are not part of the Multi-Document Interface, however they remain open all the time and do not need to be closed to switch to another window. These windows always show information related to current active document. Switching between documents automatically updates information in these windows. Those windows include:

- Components Window (**Window | Components**, or **F3**)
- Variables Window (**Window | Variables**, or **F4)**
- Transient and AC Data **(Transient/Data, AC/Data)**
- Tools (**Tools | Script, Tools | Sweep,** etc.)

Dialog boxes

Unlike Windows, Dialog boxes must be closed to return to the Main Window. Typically, Dialog boxes have **OK** and **Cancel** buttons, and some have **Close** button. Examples of the Dialog box are:

- Preferences **(Edit | Preferences).**
- Schematic Tools **(Schematic | Tools).**
- Transient Settings **(Transient | Settings).**
- …and more.

Help

NL5 help file `nl5.chm` should be located in the same directory as `nl5.exe`. The file contains quick reference information, such as description of operators, functions, commands, components, and models. For detailed information refer to this Manual. To open **Help** select **Help | Help** Main menu command. For context Help, click **F1** hot key anywhere in the program, or **Help** button , which is available in some windows and dialog boxes.

Hot keys

- **F1** - Help
- **F2** - Show schematic window
- **F3** - Show/hide components window
- **F4** - Show/hide variables window
- **F5** - Show transient window
- **F6** - Start transient
- **F7** - Continue transient
- **F8** - Show AC window
- **F9** - Start AC

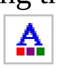
Preferences

NL5 preferences are used to customize different features of the application, such as "look and feel" (fonts, colors, formats), default parameters, memory management, etc. Preferences apply to the whole application, not to the particular document (schematic). Changing preferences does not affect simulation results.

Preferences are located in the same directory as nl5.exe, in the file called `nl5.nlp`. Preferences are saved into the file every time **Apply** or **OK** button in the Preferences dialog box is clicked, and on exiting NL5. At start-up, NL5 loads last saved preferences from the file.

Preferences can also be saved in the custom preferences file (extension "nlp"), and then opened back from the file. This feature allows having different profiles for different tasks and switch between them easily.

Open Preferences dialog box by Main menu command **Edit | Preferences**. Many context menus do have **Preferences** command as well, usually the bottom one in the list:

Selecting this command opens Preferences dialog box directly at context-related page. **Preferences** button  is also available in some dialog boxes and windows.

The Preferences dialog box consists of several of pages. Select the page by clicking on the page name in the tree-view selection window. When any of parameters changed, **Apply** button is enabled. Then click:

- **OK** – accept changes and close the dialog box.
- **Cancel –** cancel changes and close the dialog box.
- **Apply –** accept changes without closing the dialog box.

Preferences

Save/open preferences to/from a file and select color scheme.

Preferences

- **Save preferences**. Save preferences to a file.
- **Open preferences**. Open preferences from a file.
- **Reset preferences to default**.

Color scheme. Color scheme is applied to all Document windows (Schematic, Transient, Transient Tools, AC, AC Tools). Changing color scheme also changes colors of transient and AC traces.

- **Color with black background**.
- **Color with white background**.
- **Black and white**. This scheme can be temporary used to save black and white schematic or graph image in the file or copy to clipboard.

Application

Set application options.

- **Automatically check for updates.** Can be set in the range "Never"…"Every 90 days". NL5 can automatically check for updates on the NL5 website. NL5 does not download and install updates: it only notifies if a new update is available. If your PC has anti-virus or/and firewall service active, you may be asked for granting permission to access NL5 website. If NL5 version, revision and build are current, a message will be displayed in the Status bar. If new update is detected, the dialog box with information about update and release notes will be displayed.

- **Most Recently Used files.** Can be set in the range 0…10. This is a maximum number of most recently used files displayed under **File** menu.

- **Subcircuit Library path.** A path to "Subcircuit Library" directory. If subcircuit (**SubCir** model) is located in the "Subcircuit Library" directory, a short name (without path) can be used as subcircuit file name.

- **Beep on errors and messages.** Produce sound signal when error or message window is displayed. This option does not affect sounds generated by system messages (such as "file not found", "file already exists", etc.).

Document

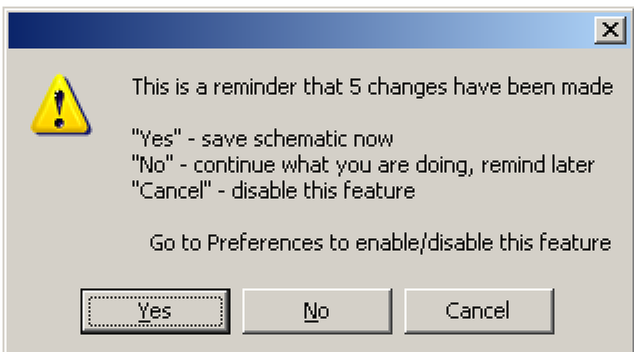
Set default properties of a new schematic and autosave/backup options.

Properties. This is default information to be set in the properties of a new schematic. To view and edit schematic properties, select **File | Properties** command.

- **Author**.
- **Organization**.

Autosave and backup

- **Create backup when saving first time**. If schematic was loaded from the file, edited, and then is being saved first time, the file it was loaded from will be converted into backup file with extension "nl5~". This prevents from schematic loss by mistakenly overwriting the file.
- **Save automatically when analysis starts**. If selected, NL5 will save schematic automatically every time transient or AC analysis starts.
- **Show a reminder when NNN changes are made**. If selected, NL5 will show a reminder after specified number of schematic changes are made:



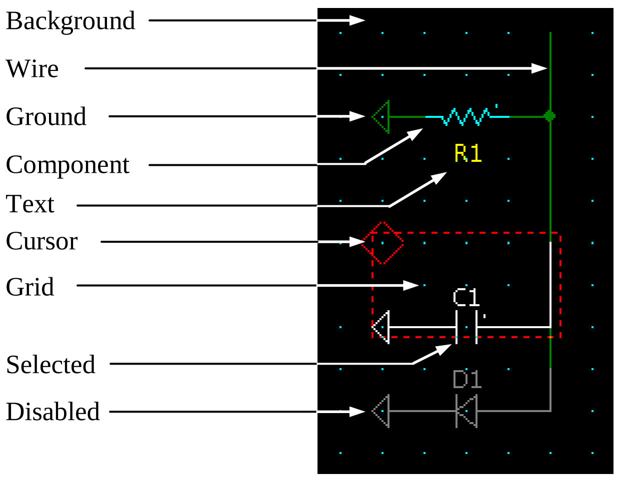
Undo

- **Max number of Undo steps.** Although Undo/Redo buffer can be unlimited, its maximum size may be specified as well. When new Undo information is being added and buffer size exceeds specified size, the earliest data will be removed from the buffer.
- **Clear Undo buffer on schematic save.** If selected, Undo buffer will be cleared when schematic is saved into file. Otherwise, all operations since opening or creating the schematic can be reversed.

Schematic

Set properties of schematic. New properties will be applied to all new and existing schematic elements, except elements with customized (formatted) properties.

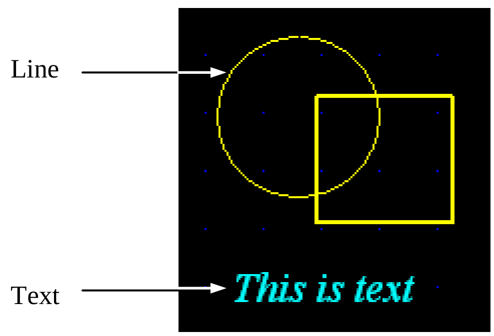
- **Font size.** Set font size of component name and value.
- **Font.** Select font of component name and value.
- **Line width.** Set width of a line (wires and components).
- **Cursor width.** Set width of a line of cursor image.
- **Attributes grid.** Select grid size for component attributes location (relative to schematic grid size).
- **Show grid points.** If selected, show schematic grid points.
- **Show node numbers.** If selected, show schematic node numbers.
- **Show hints**. If selected, show component name, model, and parameters in the hint box, when mouse pointer is moved over component.
- **Colors.** Double-click on the item in the list to change the color:



Drawings

Set default properties of a new drawing. Default properties will be applied only to new drawings. Existing drawings are not affected by default properties.

- **Font size.** Set font size of a text.
- **Font.** Select font of a text.
- **Line width.** Set width of a line, rectangle, and oval.
- **Drawings grid.** Select grid size for drawings (relative to schematic grid size).
- **Colors.** Double-click on the item in the list to change the color:



Mouse (Schematic)

Set mouse wheel action and properties.

Mouse wheel action. Select action to be performed on mouse wheel rotation, along with **Ctrl** and/or **Shift** key state.

- ***no key held*** - just mouse wheel rotation, no key held.
- **Ctrl -** mouse wheel rotation, and **Ctrl** key held.
- **Shift -** mouse wheel rotation, and **Shift** key held.
- **Ctrl+Shift -** mouse wheel rotation, and **Ctrl** and **Shift** keys held.

Select action from:

- **None**
- **Zoom**
- **Hor scroll**
- **Vert scroll**

- **Invert Zoom –** invert Zoom In/Out operation.

Components

Set properties of components and variables window and default parameters of a new component.

Components and variables window.

- **Font size.** Set font size of components and variables window.
- **Font.** Select font of components and variables window.
- **Show units with NNN color.** Select color to display units of the parameter in the components window:
  - **None** – do not show units
  - **Grey** – show with grey color
  - **Black** – show with black color

New component. Set default value of a new component:

- **R, Ohm** – resistor
- **C, F** – capacitor
- **L, H** – inductor
- **Vd (Diode), V** – diode forward voltage

Symbols

Select symbols of some component types.

- **Voltage source.** Symbol of voltage source, voltage controlled and current controlled voltage sources.
- **Current source.** Symbol of current source, voltage controlled and current controlled current sources.
- **Controlled source.** Symbol of controlled voltage and current source.
- **Resistor.** Symbol of resistor, potentiometer, voltage and current controlled resistors.

Warnings

Select warnings to be detected and shown during schematic check.

- **Warnings.** Unselect to disable all warnings.
- **Floating pins.** Check for any floating pin of a component.
- **Non-connected components.** Check for components with all pins disconnected.
- **Overlapping components and wires.** Check for overlapping component with another component and component with wire.
- **Possibly floating schematic.** Check if schematic has at least one ground or label with voltage source model.

Graphs

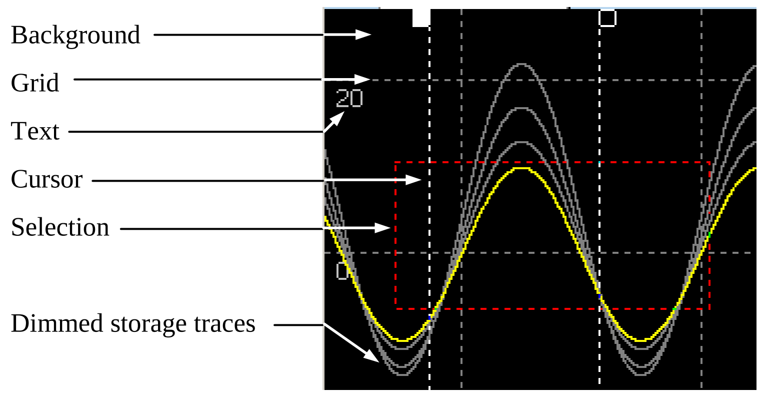
Set properties of Transient, Transient Tools, AC, and AC Tools windows.

- **Font size.** Set font size of axes numbers.
- **Font.** Select font of axes numbers.
- **Default trace width.** Set width of a new trace.
- **Markers width.** Set markers width.

Gridlines interval (pixels)

- **Vertical gridlines.** Set approximate interval between gridlines in pixels.
- **Horizontal gridlines.** Set approximate interval between gridlines in pixels.

- **Numbers alignment.** Select position of vertical scale numbers.
- **Colors.** Double-click on the item in the list to change the color:



Table

Set properties of the transient and AC data table.

Text. Set properties of the text in the table, other than phase in AC data table.

- **Font size.** Set font size of the text.
- **Font.** Select font of the text.

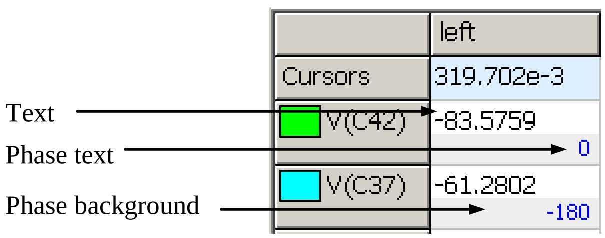
Phase. Set properties of the phase text in AC data table.

- **Font size.** Set font size of the text.
- **Font.** Select font of text.
- **Alignment.** Select alignment of the text.

Significant digits. Set number of significant digits for data table and Markers Tool.

- **Time/Frequency.**
- **Data.**

- **Colors.** Double-click on the item in the list to change the color:



Legend

Set properties of the Legend window.

- **Legend font size.** Set font size of a legend window.
- **Max width.** If unselected, the width of the Legend window will be automatically adjusted to fit names of the traces. If selected, the width of the legend window will be automatically adjusted, but only up to specified width (NNN * height of legend window font).

Annotation

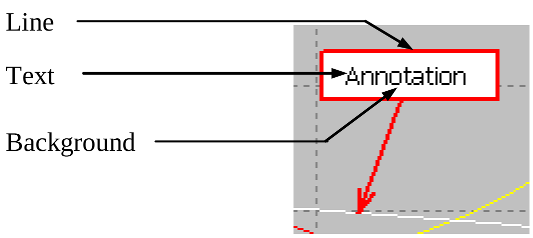
Set properties of annotations in Transient and AC windows.

- **Font size.** Set font size of annotation text.
- **Font.** Select font of annotation text.
- **Line width.** Set width of a line for an arrow pointer and a rectangle.
- **Arrow.** If selected, draw arrow pointer.
- **Draw line with trace color.** If selected, use trace color for an arrow pointer and rectangle.
- **Draw text with trace color.** If selected, use trace color for annotation text.

Significant digits. Set number of significant digits for time/frequency and data display.

- **Time/Frequency.**
- **Data.**

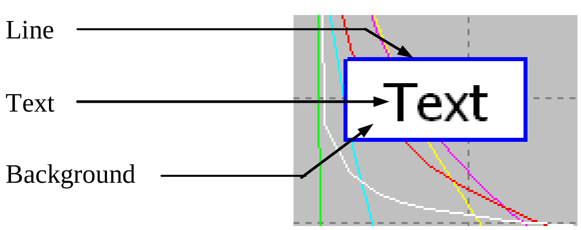
- **Colors.** Double-click on the item in the list to change the color:



Text

Set properties of a text in Transient and AC windows.

- **Font size.** Set font size of a text.
- **Font.** Select font of a text.
- **Line width.** Set width of a line for an arrow pointer and a rectangle.
- **Colors.** Double-click on the item in the list to change the color:



Mouse (Graphs)

Set mouse wheel action and properties.

Mouse wheel action. Select action to be performed on mouse wheel rotation, along with **Ctrl** and/or **Shift** key state.

- ***no key held*** – just mouse wheel rotation, no key held.
- **Ctrl –** mouse wheel rotation, and **Ctrl** key held.
- **Shift –** mouse wheel rotation, and **Shift** key held.
- **Ctrl+Shift –** mouse wheel rotation, and **Ctrl** and **Shift** keys held.

Select action from:
  - **None**
  - **Zoom** (both horizontal and vertical)
  - **Hor zoom**
  - **Vert zoom**
  - **Hor scroll**
  - **Vert scroll**

- **Invert Zoom –** invert Zoom In/Out operation.

Transient

Set transient simulation and memory options.

- **Status update interval, ms.** Update transient status with specified interval.

Memory.

- **Max memory per trace, MB.** Set maximum amount of memory allowed per one trace. If trace memory exceeds this limit, the beginning portion of the trace will be deleted, and warning message *"One or more traces have been truncated"* will show up in the transient status bar.
- **Max memory per delay and transmission line components, MB (warning).** Set maximum amount of memory allowed for delay and transmission line components. If estimated required memory exceeds specified limit, a warning message will show up, with the option to continue or stop simulation**.**

Export traces.

- **Approximate number of points.** When opening Transient Export/View dialog box, time step value is automatically selected, so that number of points in the table is close to specified number.
- **Max number of points.** Export/View time step cannot exceed this number. It will be automatically limited. This number also applies to the script command "*tracename*".

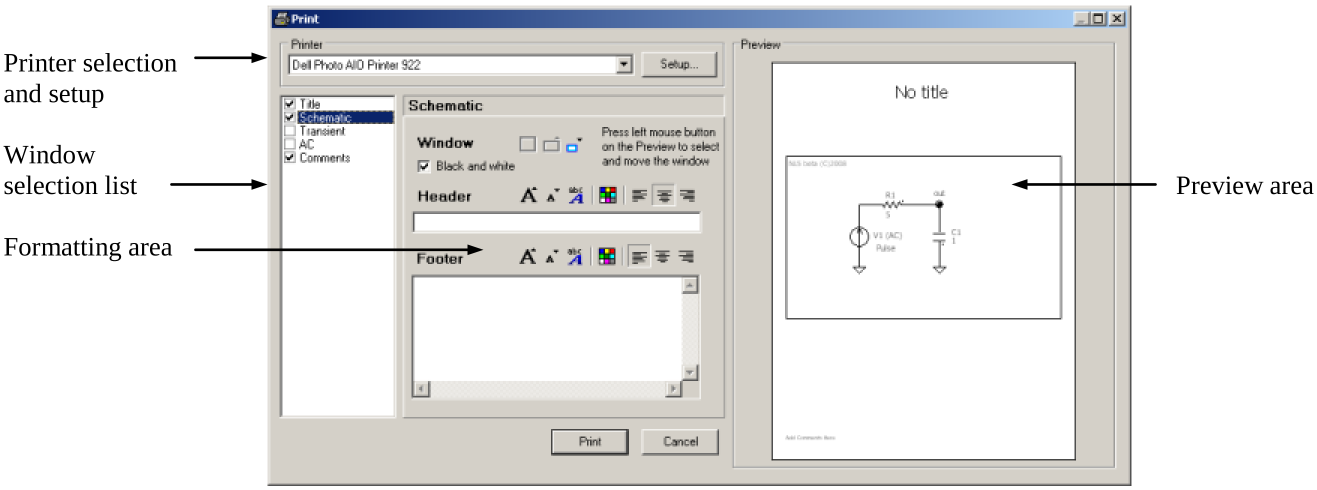
HTTP Server

Set HTTP Server options.

- **Max number of log lines.** Enter maximum number of log lines, or select from drop-down list. When number of lines exceeds specified maximum, the very first lines will be removed.

Printing

Click **Preview and print** Toolbar button or select **File | Preview and print** command to open **Print** dialog box. Typical view of the dialog box and its main components are shown below:

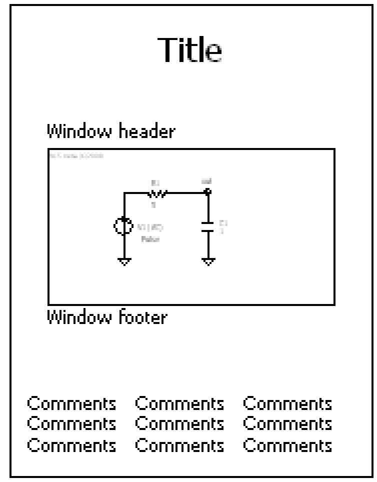


- Select printer from drop-down **Printer selection** list. Click **Setup** button for printer setup.
- Select windows to be printed in the **Window selection list**. The list contains **Title**, **Comments**, and windows of active document available for printing (opened):
  - **Schematic.**
  - **Transient.**
  - **Transient Tools.**
  - **AC.**
  - **AC tools.**

  Select checkboxes of windows to be printed.

  Please note: if Schematic, Transient, or AC window is maximized, then only that window is available for printing. In this situation warning message will be displayed below selection list.
- Select window name in the list and format window in the **Formatting area**.
- Edit windows layout in the **Preview area**. Click on the window image to select window. Click on the window image and drag to move window on the sheet.
- Press **Print** button to print, or **Cancel** button to close dialog box.

Format and layout



In the typical print layout, **Title** is displayed on the top of the sheet, and **Comments** on the bottom. However, they can be moved anywhere on the sheet. One ore more **Windows** can be displayed anywhere on the sheet, window size is adjustable in the **Window** format section. Each window may have individual **Header** and **Footer**. Header is one line text, and it is formatted in the **Header** section. Footer may have many lines, and it is formatted in the **Footer** section.

The following format buttons are available:

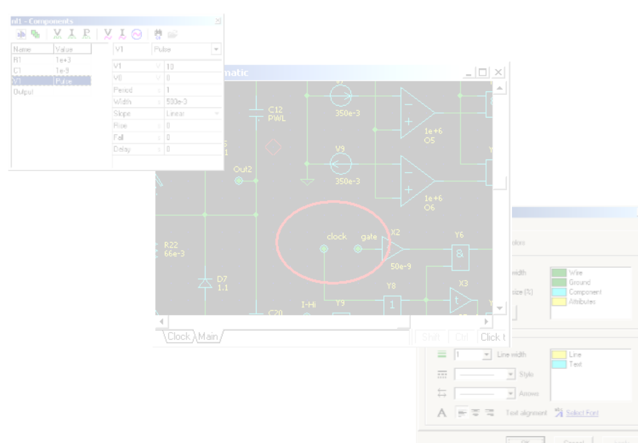
Window

- **Maximize**.
- **Larger**.
- **Smaller**.
- **Black and white.** Select to preview and print color window in black and white format.

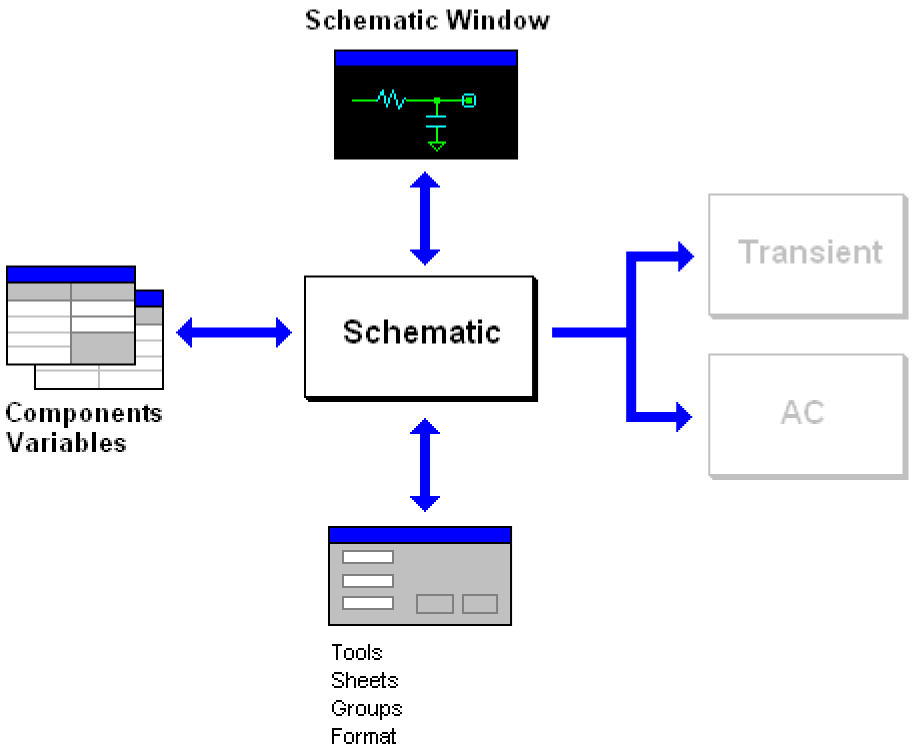
Title, Comments, Header and Footer

- **Larger font.**
- **Smaller font.**
- **Select font.**
- **Select color.**
- **Align left.**
- **Center.**
- **Align right**.

III. Schematic



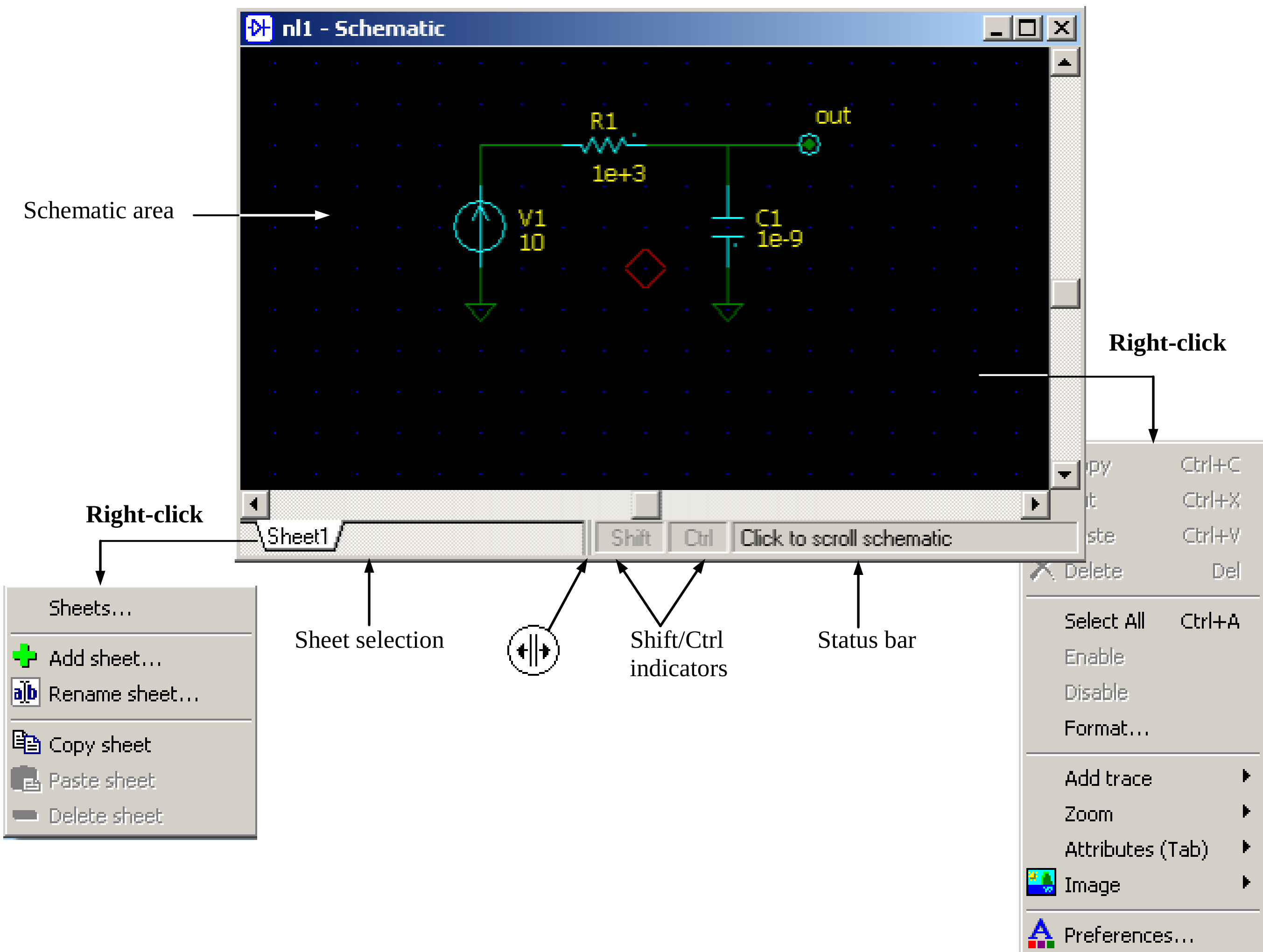
The following simplified diagram shows schematic structure and operations:



Schematic is shown and can be edited in the **Schematic window**. Any document must have Schematic window: closing Schematic window will automatically close the entire document. Components and variables are shown and can be edited in the **Components Window** and **Variables Window**. Several Dialog boxes, such as **Tools**, **Sheets**, **Groups**, **Format**, are used to perform other operations on the schematic and Schematic Window. Schematic data is used for Transient and AC analysis.

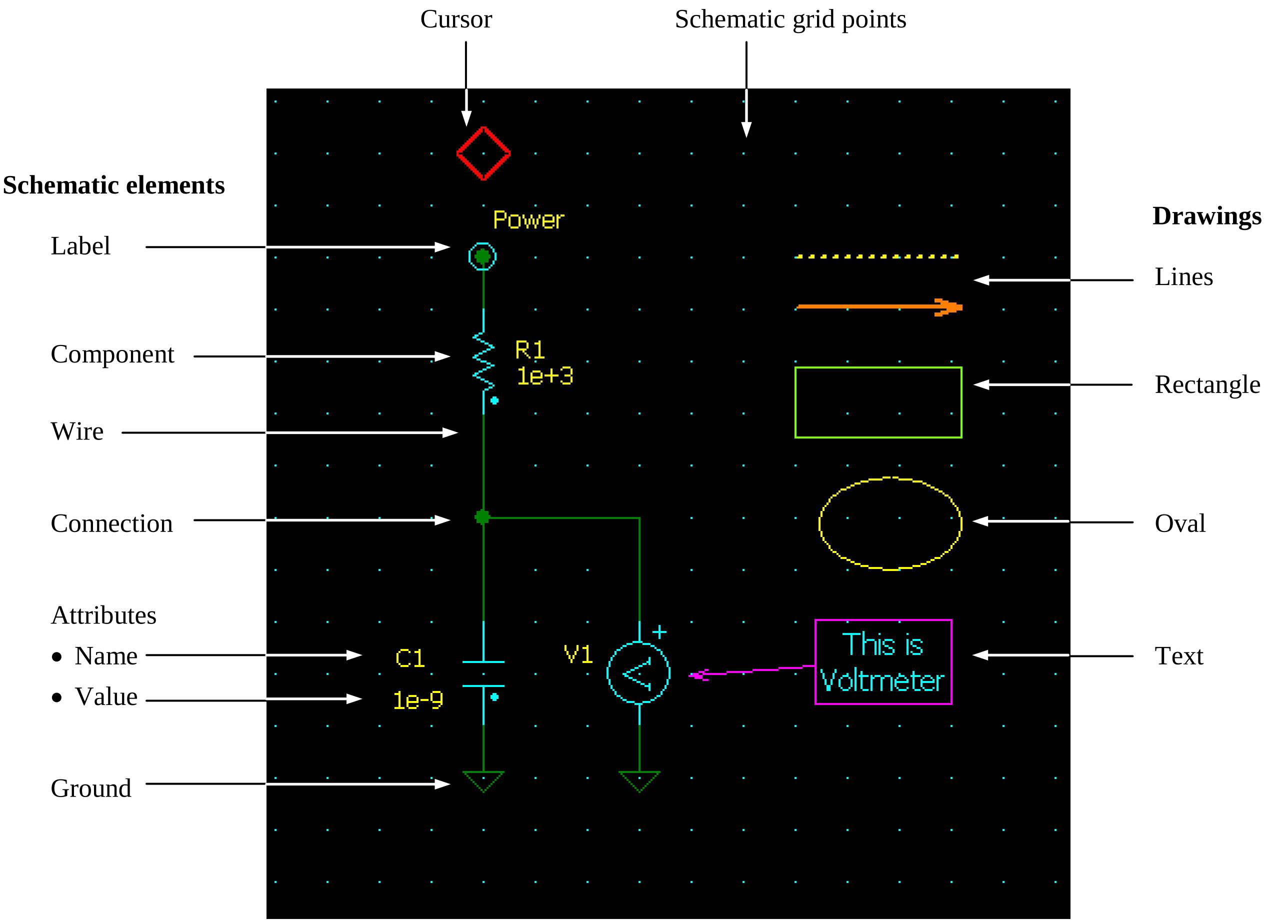
Schematic window

Typical view of schematic window and its main components are shown below:



- Schematic is shown in the **Schematic** area.
- **Sheet selection** area contains sheet tabs. Click on the tab to select sheet.
- **Right-click** on the Sheet selection area to see context menu with relevant commands.
- **Shift/Ctrl indicators** are highlighted when **Shift** and/or **Ctrl** key are depressed.
- **Status bar** shows hint related to current position of mouse pointer and **Shift/Ctrl** state.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize Sheet selection area.
- **Right-click** on the schematic to see context menu with relevant commands.

Schematic area contains schematic elements, drawings, grid points, and cursor.



- **Schematic** elements include wire, connection, ground, label, and component. Schematic elements represent "electrical" part of the schematic used for simulation.
- **Drawings** include line, rectangle, oval, and text. Drawings are used for comments and notes.
- **Grid points** are reference points for cursor and schematic elements.
- **Cursor** is used to place/select schematic elements and can be placed at grid points only.

Most of editing commands apply both to schematic elements and drawings. Unless otherwise stated, the word "drawings" is omitted in the description of those commands.

All elements are initially placed on the schematic with default properties (color, width, style, font, etc.), defined on **Schematic** page of **Preferences** dialog box. Properties of any element can be customized by formatting. Changing default properties of schematic elements applies to all existing elements on the schematic, except elements with custom properties. Changing default properties of drawings does not affect existing drawings.

Editing schematic

Schematic editing and navigation can be performed by commands available in the Main Menu, schematic context menus, Main Toolbar, Schematic toolbar, shortcuts, keyboard keys, and mouse. NL5 supports many commands and shortcuts that are commonly used in Windows applications (such as **Edit | Copy** (**Ctrl-C)**, **Edit | Paste** (**Ctrl-V)**, and more), using of **Ctrl** key with mouse for select/copy operation, using window scrollbars, etc. Other commands are very intuitive, so that it would not take long to start working with schematic.

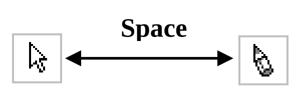
Very often, the same operation can be performed by different ways. For instance, selecting and placing new component on the schematic can be done using keyboard keys only, mouse only, or both. It is user's choice to select the most effective and convenient one. A complete commands list is provided.

There are 6 schematic editing modes:

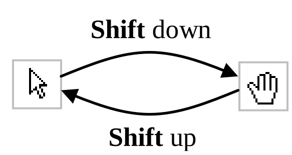
- ***Selection*.** Select elements, blocks**;** place components.
- ***Wire***. Draw wire, place components.
- ***Scrolling***. Scroll schematic.
- ***Line***. Draw line.
- ***Rectangle***. Draw rectangle and square.
- ***Oval***. Draw oval and circle,

The editing mode can be selected by clicking the button on the schematic toolbar. Also, there are some quick ways to switch between most often used modes:

- Press **Space** to switch between ***Selection*** and ***Wire*** modes:



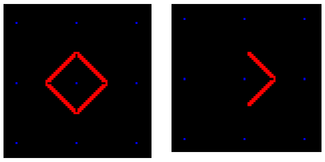
- Point mouse pointer to empty space, press and hold **Shift**, click and drag mouse to scroll schematic. Release **Shift** to return to ***Selection*** mode:



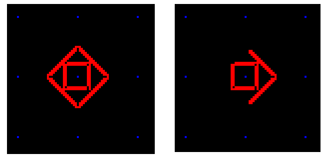
- Press **Esc** or **Space** in all modes to switch to ***Selection*** mode:

Cursor

Cursor is used as a marker to place a new schematic element: wire, ground, connection, component, or label. Cursor is used in two modes: ***Selection*** and ***Wire***:



Selection mode



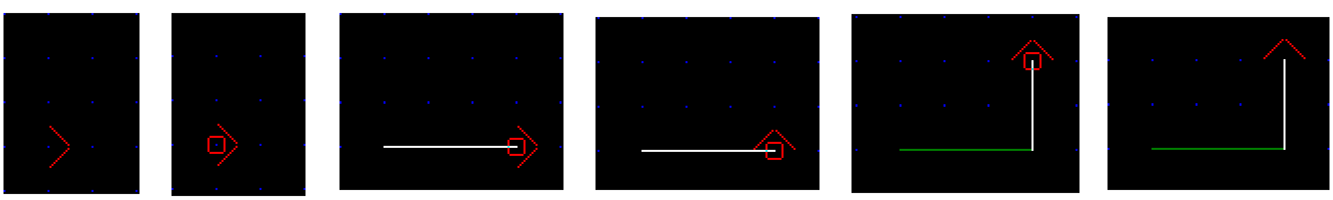
Wire mode

- Use **Left**, **Up**, **Right**, **Down** keys to change cursor direction and move cursor.
- To change cursor direction by mouse, click close to the corner pointing to a new direction.
- Click on the schematic to move cursor to a new point.
- Press **Home** to center cursor on the screen.
- Press **Space** to switch between ***Wire*** and ***Selection*** modes.
- In ***Selection*** mode, move cursor on the element and click to select the element.
- In ***Wire*** mode, move cursor to draw a new wire.
- When cursor reaches the edge of schematic window, the window will scroll automatically.

Wire

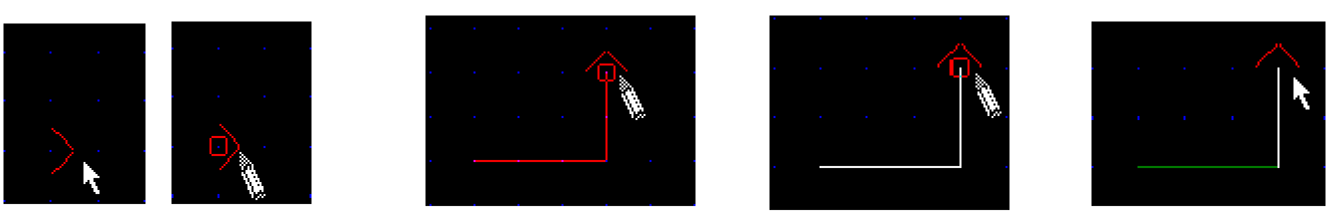
The following examples show how to place wire using keyboard and mouse.

Keyboard. Move cursor to the starting point using arrow keys (**Left**, **Up**, **Right**, **Down)**. Press **Space** to switch to ***Wire*** mode, then move cursor using arrow keys. A new wire appears in selected state. To complete the wire, press **Space** to switch back to ***Selection*** mode, or change cursor direction and continue new wire in another direction:



Press **Space** Press **Right…** Press **Up** Press **Up…** Press **Space**

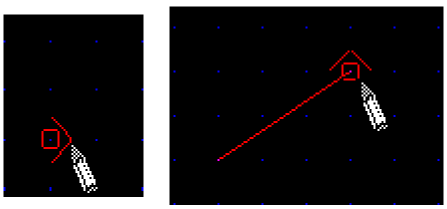
Mouse. Click **Wire** button  to switch to ***Wire*** mode. Click on the wire starting point, hold mouse button, and drag to the ending point of the wire, then release mouse button. You can do two orthogonal pieces of wire at once. Click **Selection** button  to switch back to ***Selection*** mode:



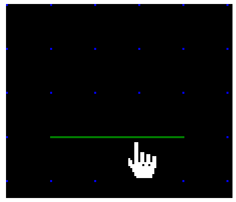
Click **Wire** button Click and drag right, then up Release left button Click **Selection** button

You can use both keyboard and mouse to draw wires. For instance, use **Space** key to switch between ***Selection*** and ***Wire*** modes, and use mouse to draw wires.

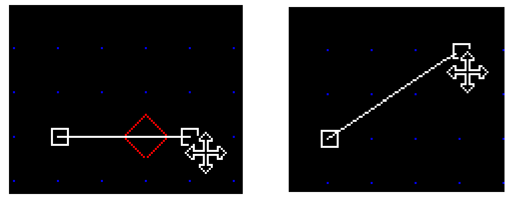
To draw diagonal wire hold **Ctrl** key while dragging and releasing mouse button. Another way to make diagonal wire is to select existing wire, then click and drag the end of the wire:



Hold **Ctrl** key, click and drag



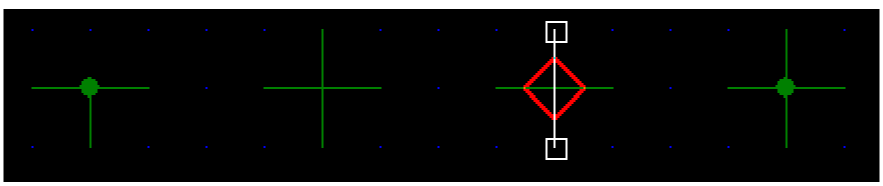
Click to select



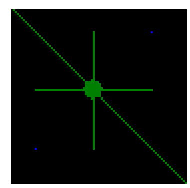
Click and drag the end

Connection

Three wires coming to one point are always connected. Connection point will be automatically placed here during schematic check. Two crossing wires are not connected by default. To connect these wires, place connection point: move cursor to the crossing and press **'.'** (**dot**) key, or move mouse pointer over crossing and click on **Connection** image ( ) in the Selection Bar.



Automatic connection | No connection | Press '.' | Connection



Diagonal wire not connected!

All the unnecessary connection points will be automatically deleted during schematic check. **Warning**: diagonal wire may be not connected to other wires even if connection point is placed at the crossing. Try to avoid connection of two diagonal wires at one point.

Ground

To place a ground press '**G**' key, or click on **Ground** image ( ) in the Selection Bar. The ground is common for entire document, including all sheets and all subcircuits.

Component

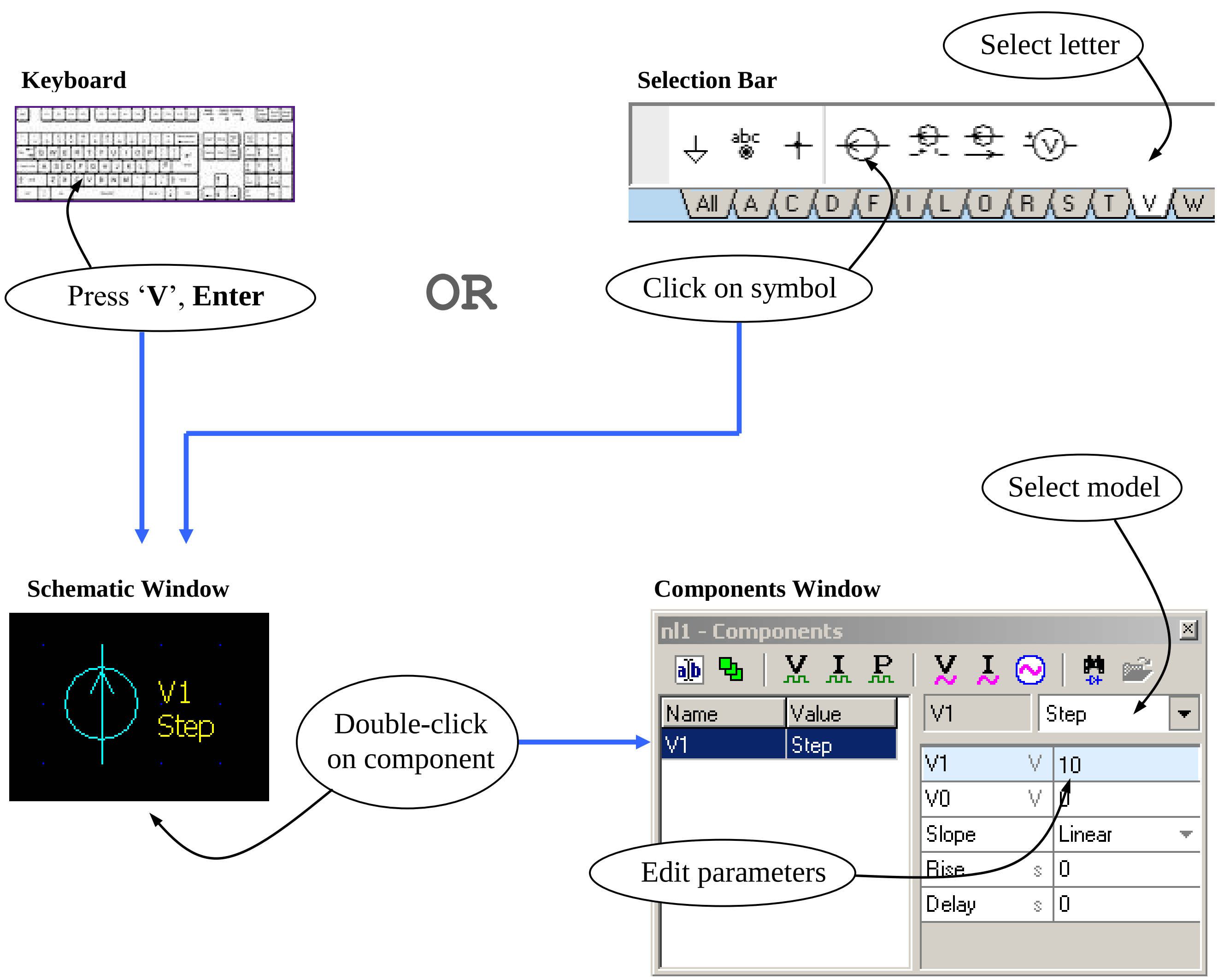
Each component type has a designated **letter** and **symbol.**

Letter identifies functional group of a component. For instance, all component types with letter '**S**' are switches. Selection Bar has a tab designated for each letter. When placing component by keyboard, the letter key is used to select component type. Default name of a component begins with this letter.

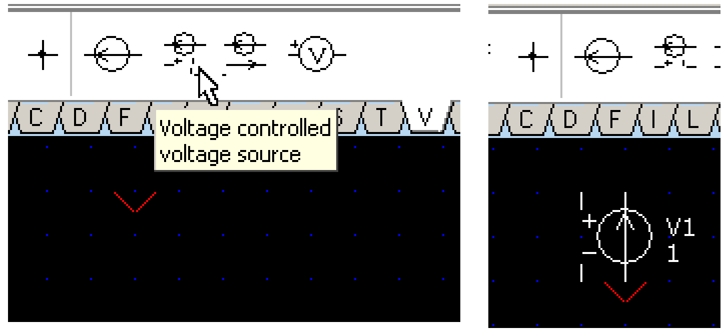
Symbol is an image of a component type: how component is displayed on the schematic**.** Selection Bar contains symbols of all available component types.

Some component types are "customized": symbol of those components, as well as number of pins, pins location and names, can be edited in the **Edit Component** dialog box. The dialog box shows up automatically at the moment when component is being placed, and can be opened any time later. See *Editing customized component* chapter for details.

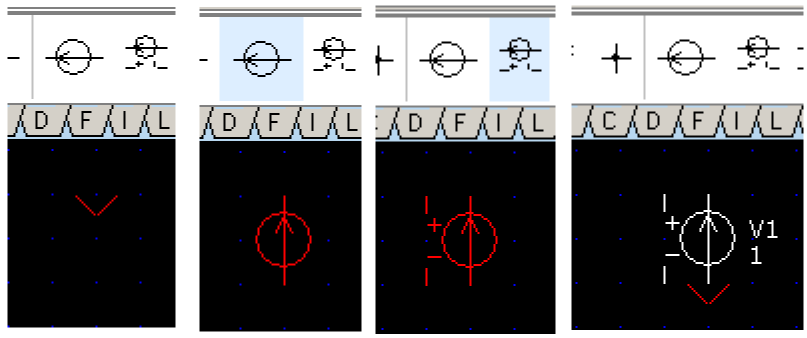
The following diagram and example show the process of placing component.



Selection Bar. Select tab with required component type (**V**), then click on component symbol to place component ("Voltage controlled voltage source"). New component shows up in "selected" state, so you can immediately rotate, mirror, flip, or select view as needed.

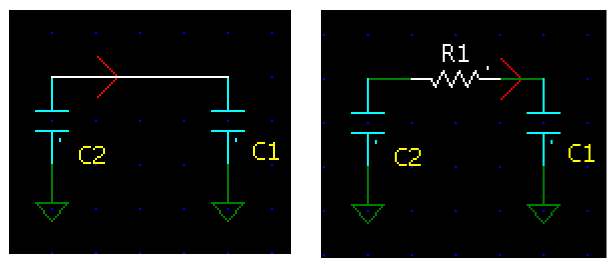


Keyboard. Press key with the letter designated to the component type (**V**). If several components types are available for this letter, press letter key again until desired type of a component shows up. Use arrow keys to move a new component, or click and drag new component by mouse. To place component press **Enter**. To cancel, press **Delete** or **Esc**.



Schematic with a new component, which is not set yet and is shown in cursor's color is sometimes called to be in ***New component*** mode.

When component is placed above existing wire, a piece of the wire underneath the component is automatically removed, so that no editing of the wire is required.



A new component has automatically generated name. The name begins with type specific letter, followed by unique number. Then, the component can be renamed in the **Components** window.

When component is placed on the schematic, its image can be modified (flipped, mirrored, rotated) to fit schematic better. In addition, some component types may have several **views.** Commands that modify image and change views are also applied to a new component while placing component using keyboard, before pressing **Enter**.

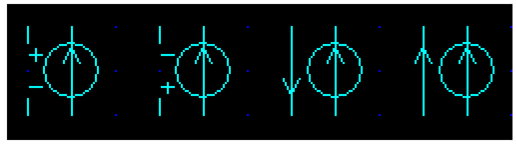
Along with schematic, a new component will show up in the **Components** window. In this window, you can see all the **models** available for component type. When model is selected, **parameters** of the model are shown and available for editing. See *Components Window* chapter for details.

To switch to **Components Window** from schematic, place cursor on the component and press **Enter,** or **double-click** on the component. In this case, if you finish editing parameters by pressing **Enter** or **Esc**, you will switch back to the schematic.

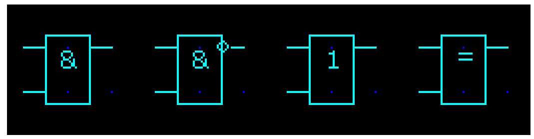
Component View

Some component types have several **views**: almost identical images with slight modification. Different views may have different pinouts, or indicate some functionality difference. A few examples of different views:

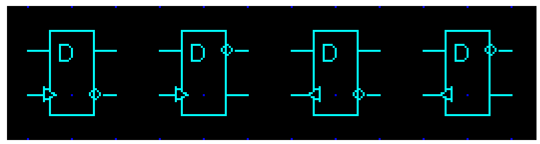
- Polarity of control signal and source:



- Logical function of logical components:



- Inverted and non-inverted inputs/outputs:



To change view of the component use the following buttons and shortcuts:

- **Next view.** Select next view of a component with multiple views.
- '+', '-' keys. Select next/previous image of a component by changing view, mirror and/or flip component, whichever is applicable.

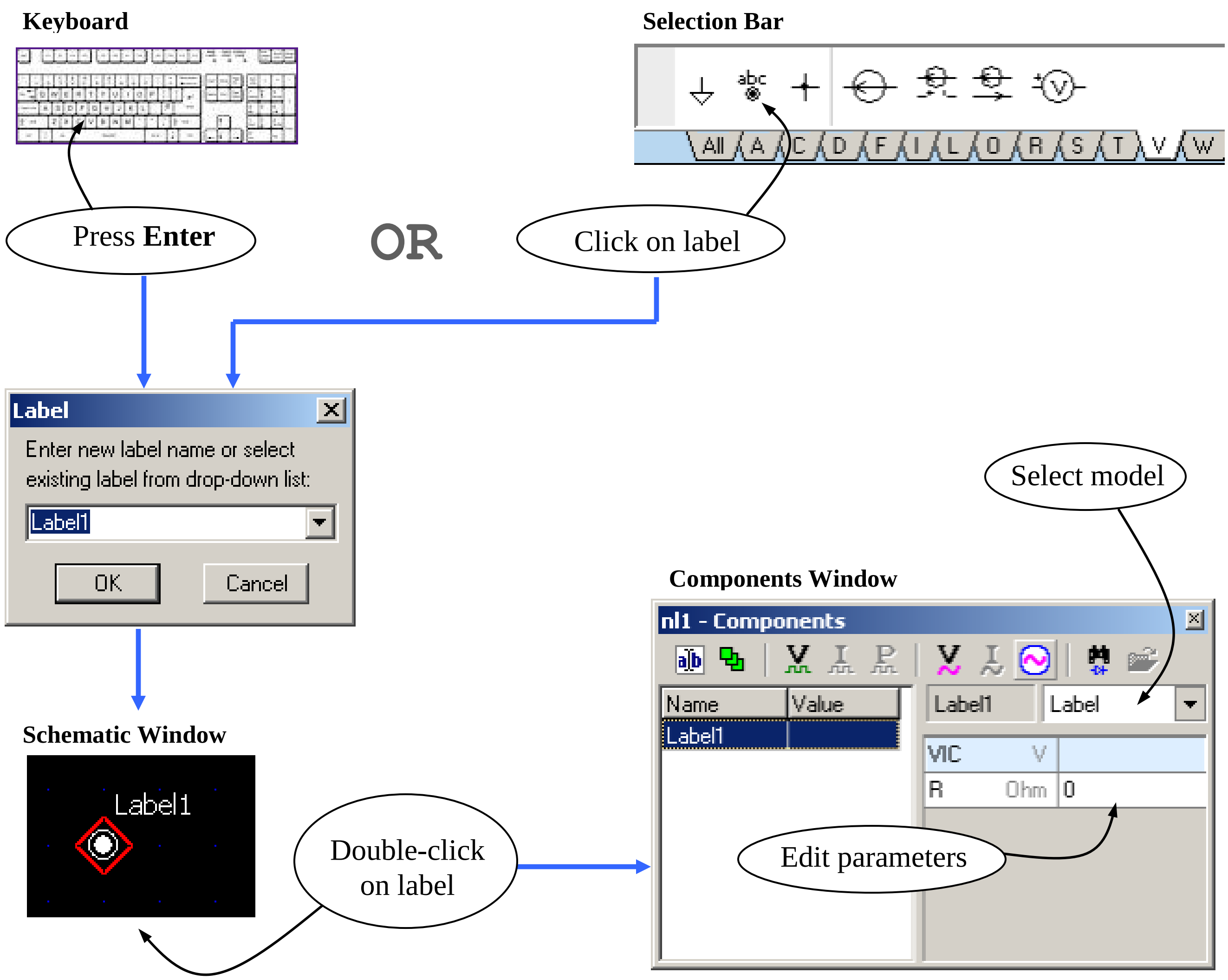
These commands can also be applied to a new component while placing component using keyboard, before pressing **Enter**.

Label

Label is similar to component, except that there can be many labels with the same name in the schematic. All labels with the same name are electrically connected. (Labels in the subcircuit are local to the subcircuit and are not connected with the main schematic). Labels can be used:

- To connect different points of the schematic without wire.
- To connect parts of the schematic located on different sheets.
- As a simulation "probe" (V trace).
- As a voltage source.

The following diagram and example show the process of placing label.

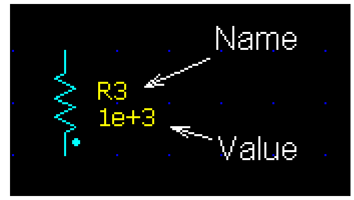


To place label press **Enter** on the wire or empty space, or click on **Label** image (abc) in the **Selection Bar**. **Label** dialog box will show up. Enter new label name or select existing label from drop-down list and click **OK**. A label will immediately show up on the schematic and in the Components Window. To set up model and parameters of the label, place cursor on the label and press **Enter**, or **double-click** on the label to switch to **Components** window.

Attributes

Attributes of the component and label include **Name** and **Value**. The following attributes display modes are available:

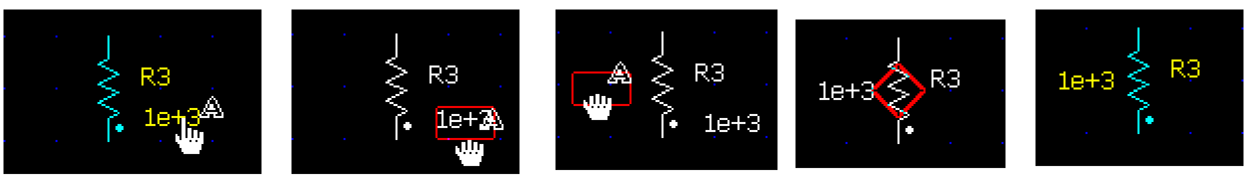
- No attributes
- **Name** only
- **Value** only
- **Name** and **Value**



Press **Tab** key to toggle attributes display mode, or select attributes under **Schematic | Attributes** Main Menu item.

Attributes can be placed with resolution higher than schematic grid. The "attributes grid" can be set up in the range "1/1" down to "1/32" of schematic grid. Attributes grid can be changed on **Schematic** page of **Preferences** dialog box.

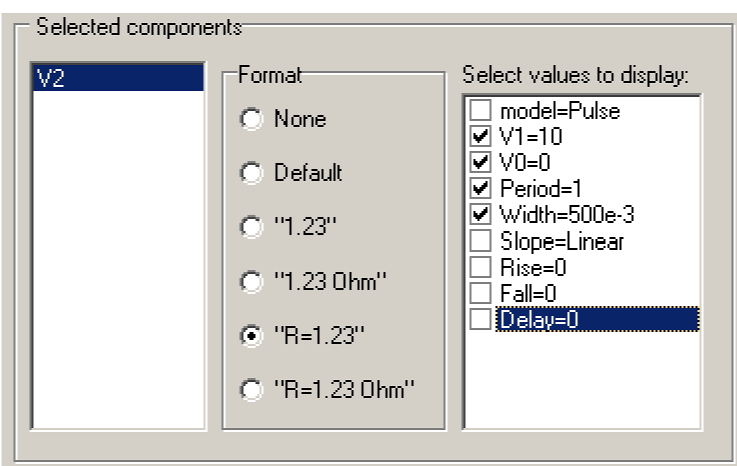
To move attribute, click on the attribute and drag:

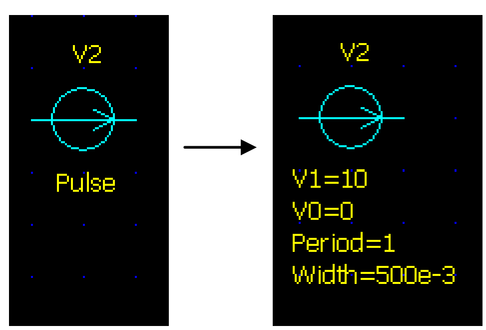


Click… Drag… Release… Unselect

To change attributes orientation select component and or click **Rotate attributes** button  , or press **Ctrl-T**.

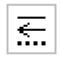
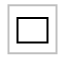
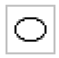
By default, **Value** is either first parameter of the component's model, or model name. List of parameters displayed for specific component can be customized in the **Format** dialog box. **Right-click** on the component, select **Format** command in the context menu, then select **Attributes** page. For the component highlighted in the list, select format and values to display.



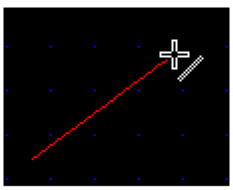


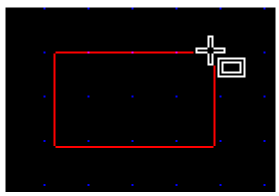
Drawings (line, rectangle, oval)

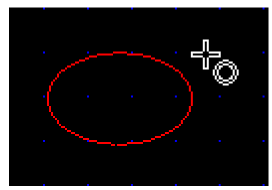
Drawings (line, rectangle, oval) do not affect functionality of the schematic and are used solely as a "decoration". Drawings can be placed with resolution higher than schematic grid. The "drawings grid" can be set up in the range "1/1" down to "1/32" of schematic grid. Drawings grid can be changed on **Drawings** page of **Preferences** dialog box.

- To place line, click **Line** button  to switch to ***Line*** mode.
- To place rectangle, click **Rectangle** button  to switch to ***Rectangle*** mode. .
- To place oval, click **Oval** button  to switch to ***Oval*** mode.

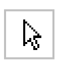
Click on the starting point, hold mouse button, and drag to ending point of the drawing, then release mouse button:

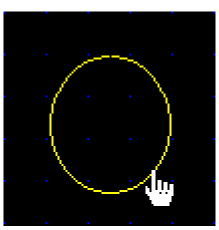

Line

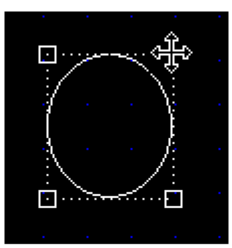

Rectangle (square)

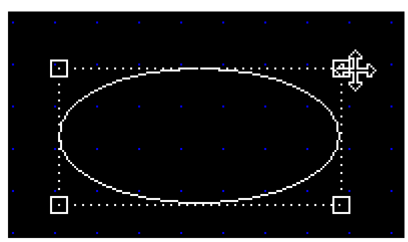

Oval (circle)

To draw square or circle, hold **Ctrl** key while dragging and releasing mouse button.

Click **Selection** button  to switch back to ***Selection*** mode. To change size and/or shape of the drawing select drawing, then click square marker and drag:


Click to select

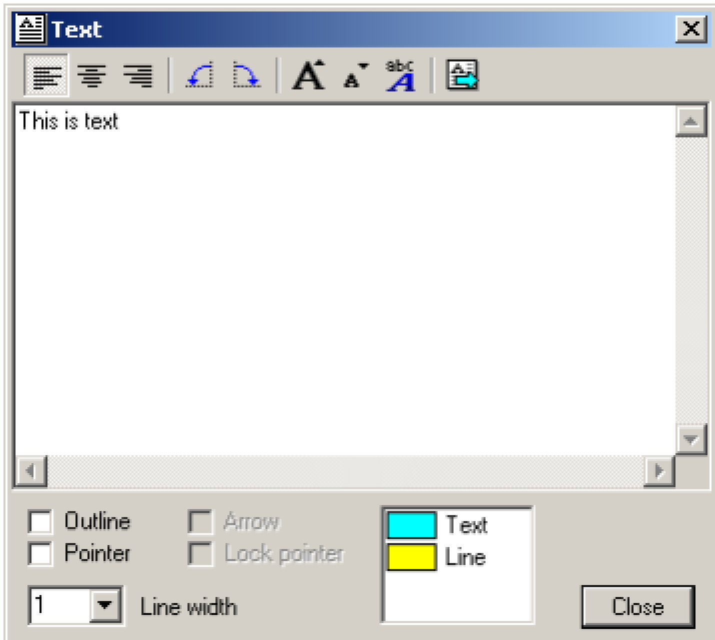

Click and drag



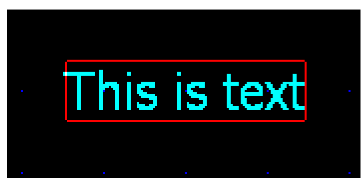
When placed, drawings show up in a default format. To change color, line width and type of drawings, select drawings, then select **Edit | Format** command from Main menu, or right-click on the drawings and select **Format** command from context menu. To format just one drawings element, simply **double-click** on the element.

Text and Variables

To enter text click **Insert text** button . **Text** dialog box will show up:



Enter text in the text box. The text will be simultaneously shown on the schematic:



The text can be formatted using toolbar buttons and controls:

Alignment. Set alignment of multi-line text.

- **Align left.**
- **Center.**
- **Align right**.

Orientation. Change orientation of the text.

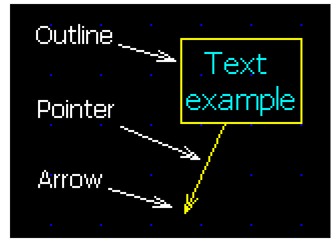
- **Rotate left.**
- **Rotate right.**

Font. Change size of the font or select specific font type and options.

- **Larger font.**
- **Smaller font.**
- **Select font.**

- **Run script.** Run script from the text (the text is considered to be a script, and will be executed).

Outline and pointer options

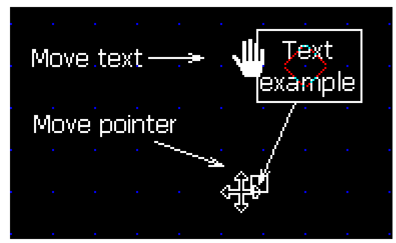


- **Outline.** Draw outline rectangle.
- **Pointer.** Draw pointer line from the text to specified point.
- **Arrow.** Draw pointer line with arrow.
- **Lock pointer.** Lock the end of the pointer: the end of the pointer will not move even when text is being moved.

- **Line width.** Specify line width of the outline and pointer.
- **Color.** Double-click on the item in the list to change the color.

Click **Close** button when done to close dialog box.

To edit the text, **double-click** on the text, or **right-click** on the text and select **Edit text** command from context menu. The same **Text** dialog box will show up.



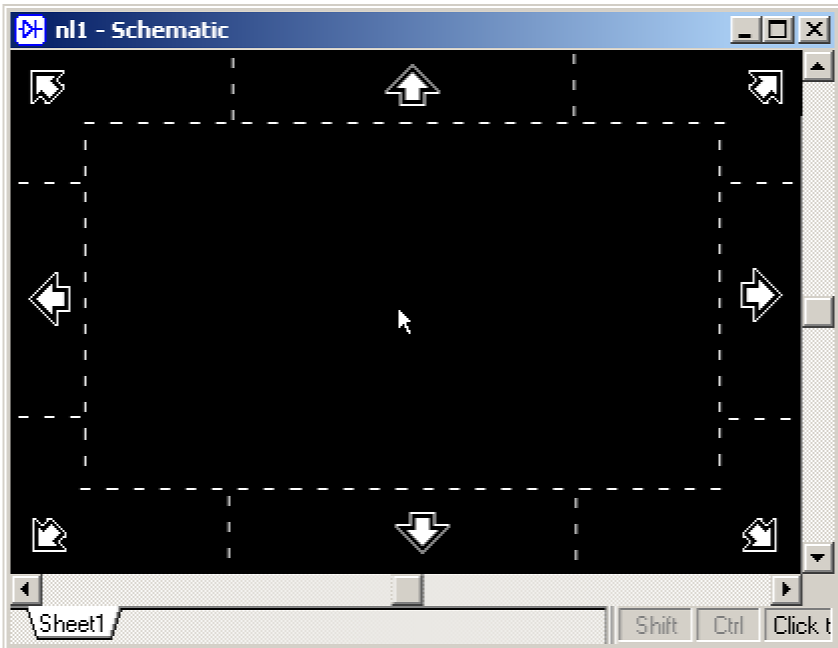
To move the text, click on the text and drag. If pointer is locked, only text will move. To move the pointer only, click on the text to select, then click and drag square marker at the end of pointer.

To place list of variables on the schematic click **Insert variables** button . Variables and their values will be shown similar to a text. The text can be formatted, but cannot be manually edited: it will be automatically updated as variables or their values change.

Scrolling and Zooming

To scroll schematic use any of the following methods:

- Move schematic cursor to the edge of schematic window, the window will scroll automatically.
- Move mouse pointer to the edge of schematic window. Mouse pointer will take "big arrow" shape. Click or hold left mouse button to scroll schematic:



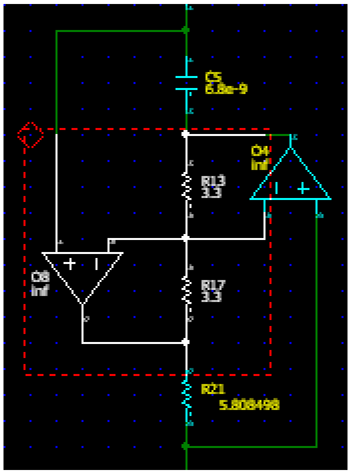
- In ***Selection*** mode , point mouse pointer to empty space, hold **Shift** key, then click and drag schematic. **Shift** key will temporary to ***Scrolling*** mode.
- Hold **Ctrl** key and rotate **mouse wheel** to scroll horizontally.
- Hold **Shift** key and rotate **mouse wheel** to scroll vertically.
- Use **Shift-Up**, **Shift-Down**, **Shift-Right**, **Shift-Left** keyboard shortcuts.
- In ***Scrolling*** mode click and drag schematic, or press **Up, Down, Right, Left** keys.
- Press **Home** to center cursor on the screen
- Point and **double-click** on the schematic to set cursor and center it on the screen

To zoom schematic use any of the following methods:

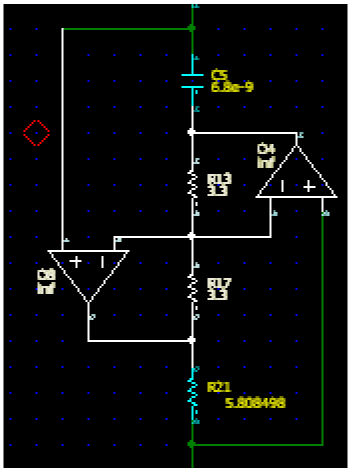
- Rotate **mouse wheel** to zoom-in and zoom-out.
- Click schematic toolbar buttons or use keyboard shortcuts:
  - **PgUp** - zoom-in
  - **PgDn** - zoom-out
  - **Ctrl-Home -** fit all schematic to the screen
  - **Shift-Home** - fit selection to the screen
- **Right-click** on schematic window to open context menu, select **Zoom** item, then select schematic scale in percents (25%…250%).

Select and Unselect

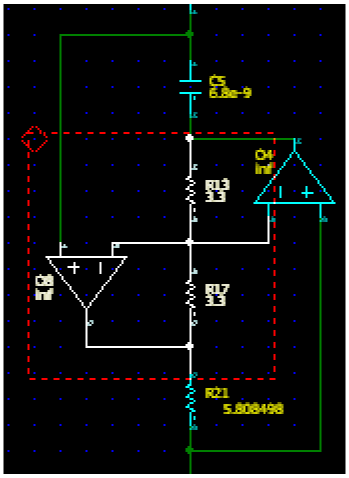
- To select one schematic element, point and click on the element.
- To select a block, point on the empty space, click and drag selection rectangle. Depending on state of **Ctrl** and **Shift** keys at the moment of releasing mouse button, the following selection can be done:
  - No keys depressed. Only components completely located in the selected area; only part of the wires located in the selected area. Selection is bounded by a rectangle.
  - **Ctrl** key is depressed. All components and wires with any part in the selected area. Selection is not bounded by a rectangle.
  - **Shift** key is depressed. Only components and wires completely located in the selected area. Selection is bounded by a rectangle.



No keys depressed



Ctrl key depressed

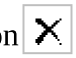


Shift key depressed

- To add new selection to existing one, press and hold **Ctrl** key, then select a new element or a new block.
- To select all elements, press **Ctrl-A**.
- **Right-click** to select element and open context menu.
- **Select Net** command in the context menu selects schematic element with all wires connected to the element either directly, or through labels (including other sheets).
- Moving schematic cursor automatically selects element under the cursor.
- To unselect, point and click on empty space, or press **Esc**.
- To unselect a block, point and click on empty space outside the block, or press **Esc** twice. Pressing **Esc** first time removes block rectangle, pressing second time unselects all elements.

  Schematic with selection bounded by a rectangle is sometimes called to be in ***Block selected*** mode.

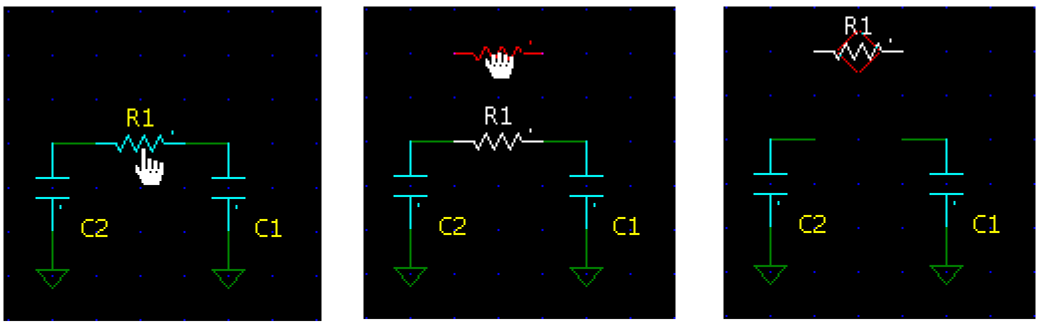
Delete

To delete elements, select elements or block, then press **Del**, or click **Delete** button  .

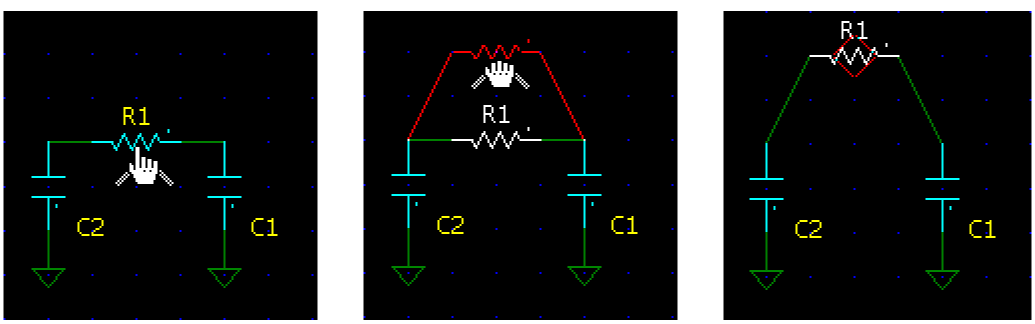
To delete entire sheet, **right-click** on Sheets selection tab, then select **Remove sheet** command in the context menu. The sheet will be deleted permanently and can't be restored. Schematic should contain at least one sheet, so the last sheet cannot be deleted.

Move and Copy

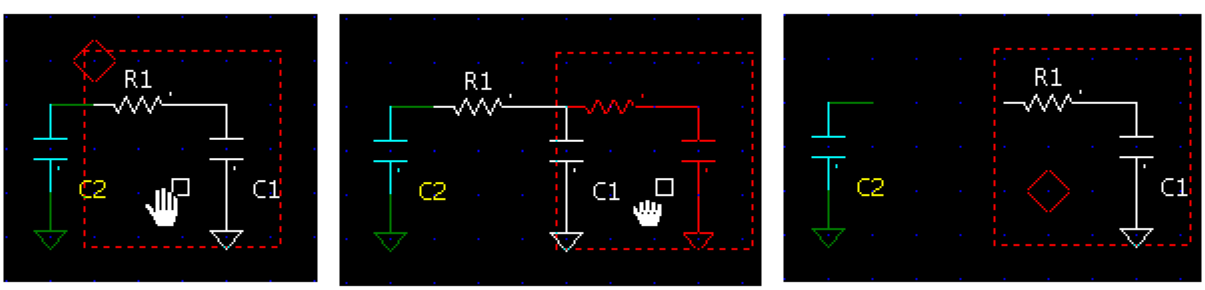
- To move element: click on the element and drag to a new location:



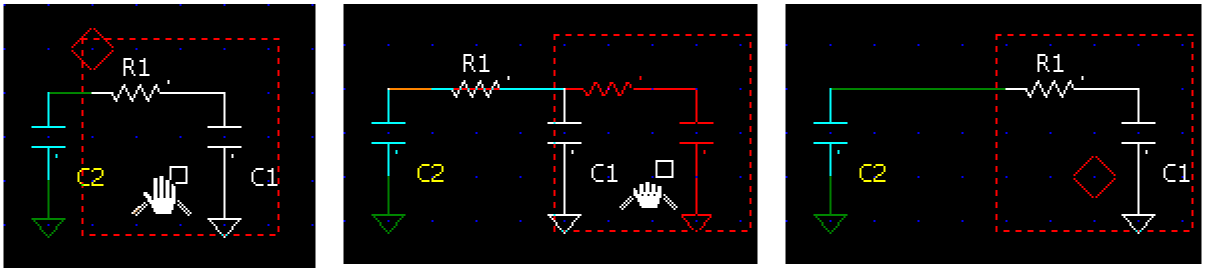
- To move schematic element with rubber bands: hold **Shift** key, click on the element and drag to a new location:



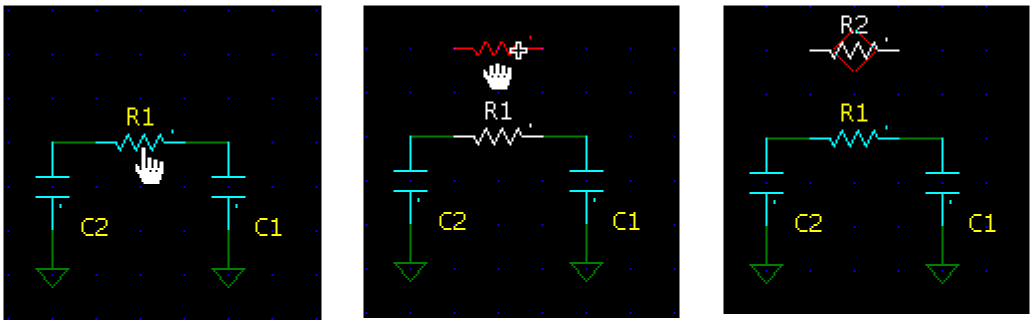
- To move block: click on selection and drag to a new location.



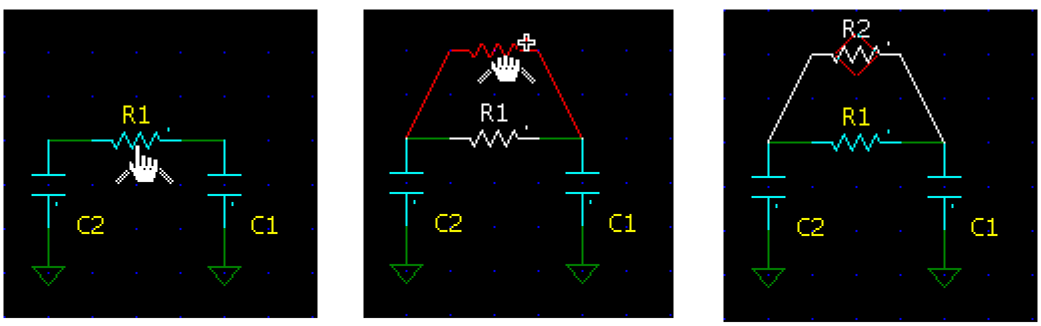
- To move block with rubber bands: hold **Shift** key, click on selection and drag to a new location.



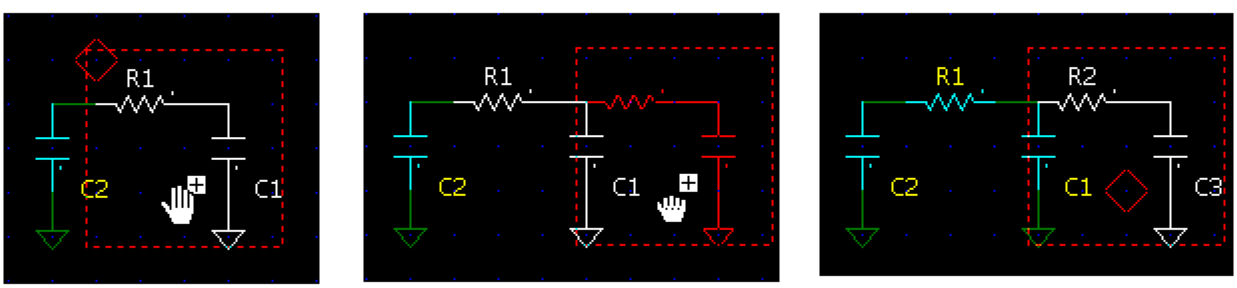
- To copy element: hold **Ctrl** key, click on the element and drag to a new location.



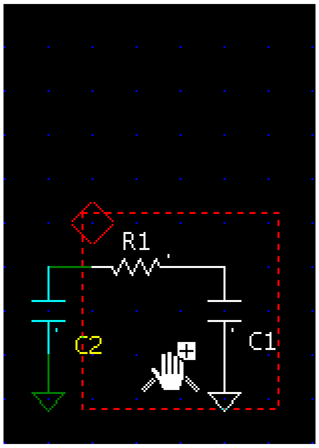
- To copy schematic element with rubber bands: hold **Ctrl** and **Shift** keys, click on the element and drag to a new location.

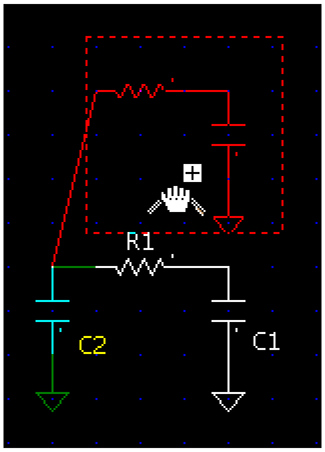


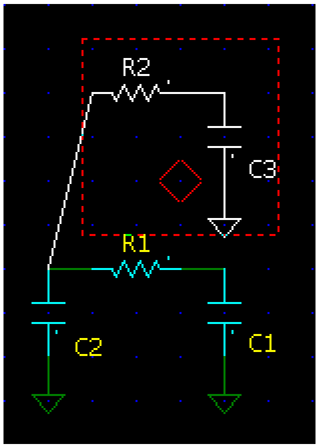
- To copy block: hold **Ctrl** key, click on selection and drag to a new location.



- To copy block with rubber bands: hold **Ctrl** and **Shift** keys, click on the element and drag to a new location.

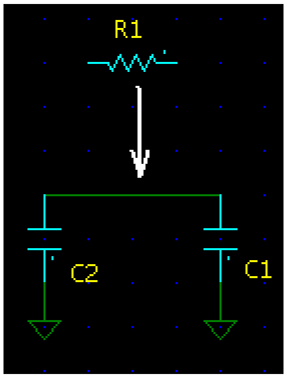


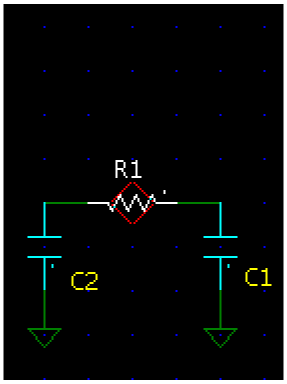




Move/copy operations can also be done with standard commands and shortcuts **Edit | Copy** (**Ctrl-C**), **Edit | Cut** (**Ctrl-X**), and **Edit | Paste** (**Ctrl-V**). Use these commands to move/copy elements to another sheet, document, or NL5 application as well.

When component is moved/copied above existing wire, a piece of the wire underneath the component will be automatically removed, so that no editing of the wire is required:





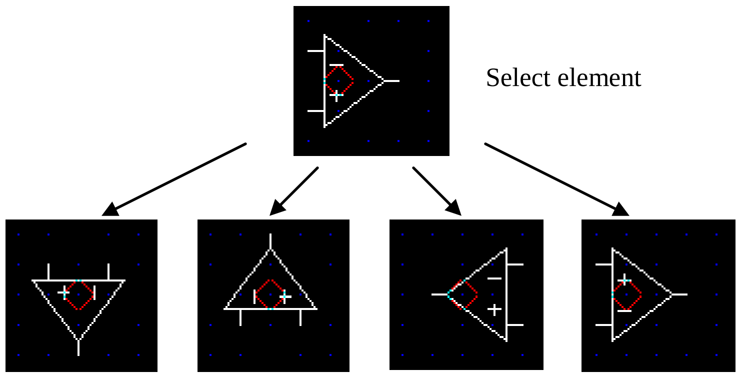
Disable and Enable

Schematic elements can be disabled. Disabled elements are shown in "disabled" color and are not used for simulation. Disabling elements allows temporary exclude elements from simulation without deleting. To disable, **right-click** on selection and select **Disable** command from context menu. To enable, select **Enable** command.

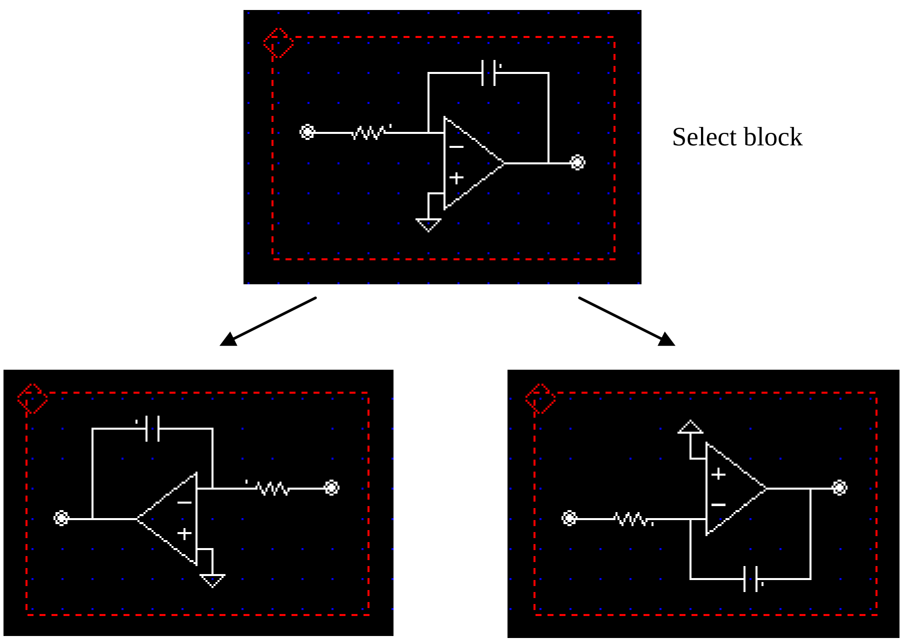
Mirror, Flip, Rotate

To change orientation of a schematic element, use **Rotate**, **Mirror**, and **Flip** commands**:**

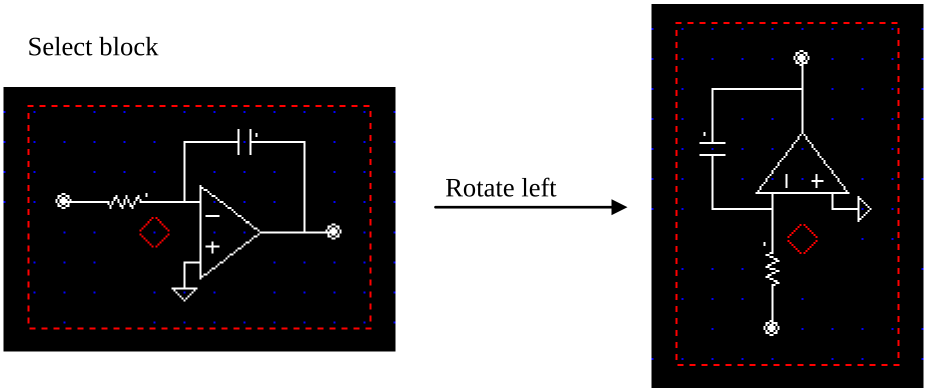
- **Rotate right (Ctrl-R).** Rotate right (clockwise).
- **Rotate left (Ctrl-L).** Rotate left (counterclockwise).
- **Mirror (Ctrl-M).** Mirror (flip around vertical axis).
- **Flip (Ctrl-F).** Flip (flip around horizontal axis).



Those commands can be applied to selected block as well:

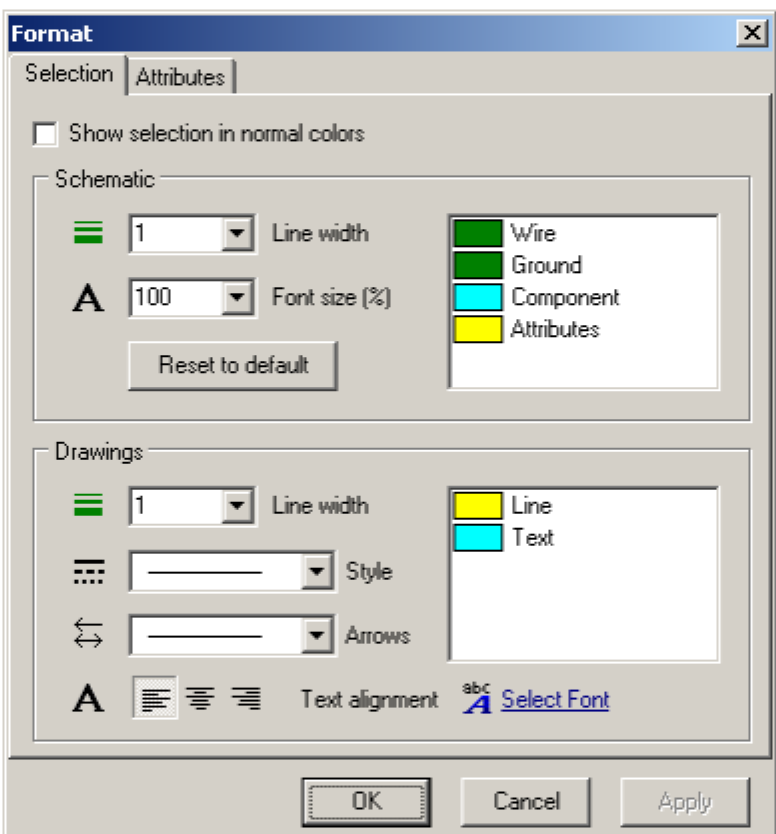


To rotate block, select the block, then place cursor to the center of rotation inside the block:



Format

All elements are initially placed on the schematic with default properties (color, width, style, font, etc.). After that properties of any element can be customized by formatting. To format selected elements, **right-click** on the selection and select **Format** command from context menu, or select **Edit | Format** command from Main menu. **Format** dialog box will show up:



Only properties applicable to selected elements will be enabled. For instance, if only wires are selected, all fields except Schematic Line width and Wire color will be disabled. If selected elements have different value of the same property, corresponding field will be enabled, but left blank. Leave it blank to keep individual values unchanged, otherwise they will be set to the same value.

- **Show selection in normal colors**. When **Format** dialog box opens, all selected elements have the same, "selected" color. Select this checkbox to see all elements in their normal color.

Schematic.

- **Line width**. Line width of all selected schematic elements: wires, components, labels, and grounds.
- **Font size**. Font size of component attributes.
- **Color**. Double-click on the item in the list to change the color.
- **Reset to default**. Click the button to reset custom properties back to default.

Drawings.

- **Line width**. Line width of all selected drawings: lines, rectangles, ovals and texts.
- **Style**. Line style for line, rectangle, and oval.
- **Arrows**. Arrows control for lines.
- **Text alignment**. Alignment of multi-line text.
- **Select font**. Select font of the text.
- **Color**. Double-click on the item in the list to change the color.

- **OK.** Accept changes and close the dialog box.
- **Cancel.** Ignore last changes and close the dialog box.
- **Apply.** Accept changes without closing the dialog box.

Undo and Redo

To undo schematic and component parameters changes, click **Ctrl-Z**, or click **Undo** button . To reverse undo operation, click **Ctrl-Y**, or click **Redo** button . An unlimited number of undo operations may be reversed with the redo command.

Size of the Undo buffer is unlimited, and by default the buffer keeps all changes since the document is created or loaded from the file, and buffer is cleared when schematic is saved into the file. However, for optimal performance the size (number of undo steps) may be specified on the **Document** page of **Preferences** dialog box. Also, unselecting option "Clear Undo buffer on schematic change" allows keeping all changed done since the file was created or loaded from the file, and never clear the buffer.

Undo operation does not apply to component parameters changed during Sweep, Optimization, Script execution, or done from Console tool.

Schematic editing commands

The following schematic editing commands, buttons, and shortcuts are available in the Main Menu, Main Toolbar, Schematic Toolbar, and schematic context menu.

- **Show schematic window (F2).**
- **Schematic Tools**.
- **Check schematic**.
- **Sheets.** Open Sheets dialog box.
- **Groups**. Open Groups dialog box.
- **Parts list**. Show parts list in the **Properties** dialog box.
- **Attributes ►**
  - **Name.** Toggle attribute name display.
  - **Value.** Toggle attribute value display.
- **Image ►**
  - **Copy to clipboard.** Copy image of schematic window to the clipboard.
  - **Save as BMP.** Save image of schematic window to a file in "bmp" format.
  - **Save as JPG.** Save image of schematic window to a file in "jpg" format.
- **Cut (Ctrl-X).** Cut selection (copy to clipboard and delete).
- **Copy (Ctrl-C).** Copy selection to clipboard.
- **Paste (Ctrl-V)**. Paste from clipboard.
- **Delete (Del)**. Delete selection.
- **Undo edit (Ctrl-Z).** Undo schematic and component parameters change.
- **Redo edit (Ctrl-Y).** Reverse undo.
- **Select All (Ctrl-A)**. Select all schematic elements.
- **Select Net**. Select element, and all wires connected to the element either directly, or through labels (including other sheets).
- **Format.** Format selected elements.
- **Preferences.** Open **Preferences** dialog box.
- **Help (F1).** Press **F1** to get help on schematic. To get Help on component, select the component and press **F1**.

Toolbar and context menu

- **Selection.** Switch to ***Selection*** mode.
- **Draw wire**. Switch to ***Wire*** mode.
- **Scrolling**. Switch to ***Scrolling*** mode.

- **Draw line**. Switch to ***Line*** mode.
- **Draw rectangle**. Switch to ***Rectangle*** mode.
- **Draw oval**. Switch to ***Oval*** mode.
- **Insert text.** Enter and place text on the schematic.
- **Insert variables.** Place list of variables on the schematic.
- **Zoom-in (PgUp).** Center cursor and zoom-in schematic.
- **Zoom-out (PgDn).** Center cursor and zoom-out schematic.
- **Schematic to the screen (Ctrl-Home).** Fit schematic to the screen.
- **Selection to the screen (Shift-Home).** Fit selection to the screen.
- **Rotate right (Ctrl-R).** Rotate selected element, block, or new component right (clockwise).
- **Rotate left (Ctrl-L).** Rotate selected element, block, or new component left (counterclockwise).
- **Mirror (Ctrl-M).** Mirror selected component, block, or new component (flip around vertical axis).
- **Flip (Ctrl-F).** Flip selected component, block, or new component (flip around horizontal axis).
- **Next view.** Select next view of selected or new component with multiple views.
- **Rotate attributes (Ctrl-T).** Rotate attributes of selected component.
- **Edit text.** Edit selected text.
- **Run script.** Run script from selected text.
- **Edit component.** Open **Edit component** dialog box. Available for "customized" components.
- **Open subcircuit**. Open subcircuit schematic file. Available for components with **SubCir** model, only if subcircuit file name is defined.
- **Enable.** Enable selected elements.
- **Disable.** Disable selected elements.

- **Add trace ►**
  - **Voltage.** Add transient voltage trace for selected component. If wire is selected, adds label first, and then adds voltage trace for label.
  - **Current.** Add transient current trace for selected component.
  - **Power.** Add transient power trace for selected component.
  - **AC Voltage.** Add AC voltage trace for selected component. If wire is selected, adds label first, and then adds voltage trace for label.
  - **AC Current.** Add AC current trace for selected component.
  - **Set AC Source.** Set selected component as an AC source for AC analysis.

- **Zoom ►**
  - **25%…250%.** Zoom schematic to specified percentage.

Keyboard keys and shortcuts

The following keyboard keys and shortcuts can also be used to edit schematic.

- **Space.**
  - ***Selection*** mode: switch to ***Wire*** mode.
  - Other modes: switch to ***Selection*** mode.
- **Home.** Center cursor on the screen.
- **Tab.** Toggle attributes display (name and value).
- **Enter.**
  - ***New component*** mode: place component.
  - On the component or label: switch to Components Window to edit parameters.
  - On wire or empty space: place label.
- **Esc.**
  - ***New component*** mode: cancel.
  - ***Block selected*** mode: remove block rectangle.
  - While drawing wire, line, rectangle, or oval: cancel.
  - While drugging selection by mouse: cancel.
  - All modes other than ***Selection***: switch to ***Selection*** mode.
  - Unselect all.
- **Right, Up, Left, Down.**
  - ***New component*** mode: move component.
  - ***Block selected*** mode: move selection.
  - ***Selection*** mode: change cursor direction, move cursor.
  - ***Wire*** mode: change cursor direction, draw wire.
  - ***Scrolling*** mode: scroll schematic.
- **Shift-Right, Shift-Up, Shift-Left, Shift-Down.** Scroll schematic.
- **'.' (*dot*) .** Place connection.
- **'G', 'g'.** Place ground.
- **'A'…'Z', 'a'…'z'.** Select new component of "letter" type.
- '+', '-'. Select next/previous image of a component by changing view, mirror and flip component, whichever is applicable.

Mouse operation

The following mouse operation can be used to edit schematic.

- **Click (left button).**
  - On empty space: unselect all, place cursor.
  - On selected block: place cursor.
  - On element: select element.
  - ***Wire*** mode: place cursor.
- **Ctrl-click.** Same as **click**, but do not unselect (add to selection).
- **Right-click.** Same as **click**, plus open context menu.
- **Double-click.**
  - On the component or label: select and switch to Components Window.
  - On the line, rectangle, or oval: format element.
  - On empty space: set cursor and center the screen.
- **Click and drag.**
  - On empty space: select block.
  - On attribute: move attribute.
  - On selection: move selection.
  - ***Wire*** mode: draw wire.
- **Ctrl-click and drug.**
  - On empty space: add block to selection.
  - On selection: copy selection.
  - ***Wire*** mode: draw diagonal wire.
- **Shift-click and drug.**
  - On empty space: scroll schematic.
  - On selection: move selection with rubber bands.
- **Ctrl-Shift-click and drug.**
  - On selection: copy selection with rubber bands.
- **Mouse-wheel.** Zoom-in/zoom-out. Schematic is zoomed relative to mouse pointer position.
- **Ctrl-mouse wheel**. Scroll horizontally.
- **Shift-mouse wheel.** Scroll vertically.

Components

NL5 supports over 60 component **types**. Each component type has a designated **letter**, **symbol**, and **models**.

Letter identifies functional group of a component. For instance, all component types with letter '**S'** are switches. Selection Bar has a tab designated for each letter. When placing component by keyboard, the letter key is used to select component type. Default name of a component begins with this letter.

Symbol is an image of a component type: how component is displayed on the schematic**.** Selection Bar contains symbols of all available component types.

Some component types are "**customized**": symbol of those components, as well as number of pins, pins location and names, can be edited. See *Editing customized component* chapter for details.

Model defines functionality of a component. For instance, voltage source models include **Pulse**, **Sin**, **Step**, **File**, and more. Each model has set of **parameters** specific to the model. Model and parameters of the component can be set up in the **Components window**.

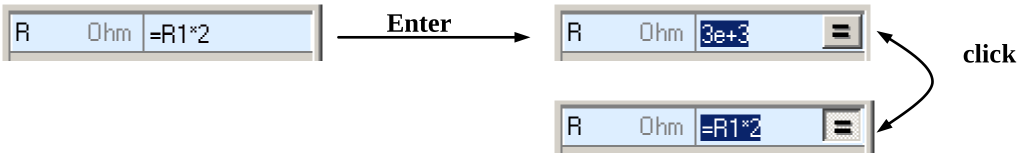
This chapter provides general information, which applies to all components, as well as some specific details on several component types and parameters. See **Attachment 1** for detailed description of all component types, models and parameters.

Formulas

Most of component parameters (numbers) and all variables can be defined as a **formula**. Formula is an expression that may contain other parameters and variables, and is automatically recalculated when any of those parameters change. Formula always starts with equal sign '=':

```
=Var1*2
=R1/2
=max(R1,R2,R3)
```

To enter formula write expression started with equal sign '=' and press **Enter**. The formula will be evaluated and its current numerical value will be displayed along with  button. Click on the button to view/edit formula:



If formula expression has an error and can't be evaluated, `#VAL` text will be displayed instead of a number.

To clear formula, enter new expression or number without equal sign instead of numerical value, or click [=] button to switch to edit mode, erase formula, and press **Enter**.

If formula contains name of a component, and the component has been renamed, the formula will be updated automatically. Formula cannot contain time and values that are changing during calculation, such as voltage, current, and power. Circular references (when some of the parameters in the formula finally point to this formula) are not allowed, and error message will be displayed.

Functions

Some component parameters can be defined as a **function**. Function is an expression, which is recalculated at every transient or AC calculation step. In addition to numbers and names, some functions can also use the following variables:

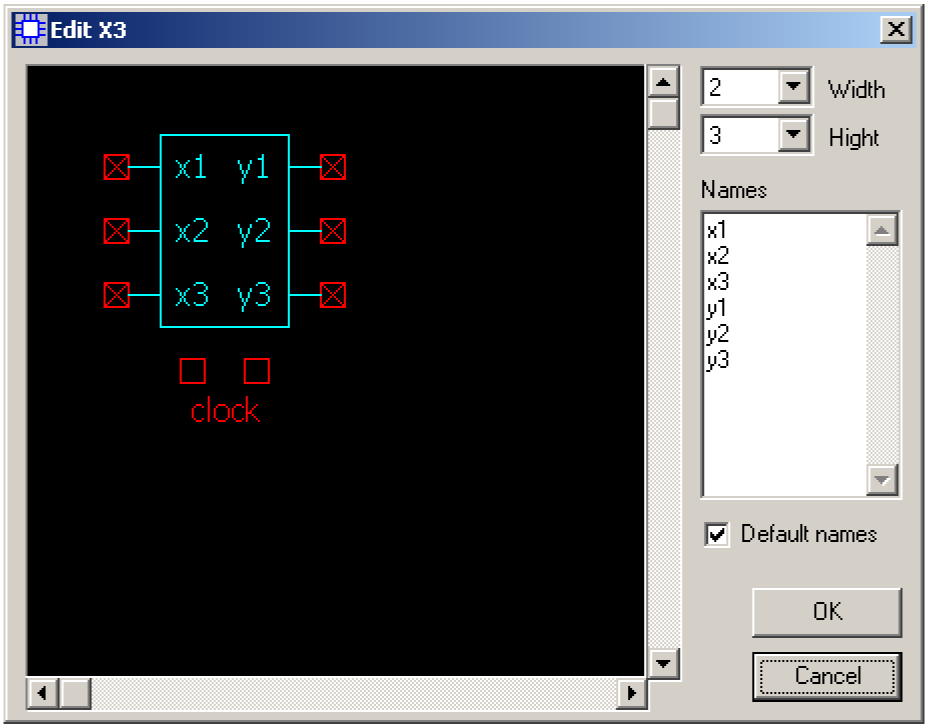
t – current transient time, s.
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s*** = ***p*** = ***j*2πf***.
x, y – input signals for **Function** model.
V(name) – voltage on the component ***name***. V trace should be available for the component.
I(name) – current on the component ***name***. I trace should be available for the component.
P(name) – power on the component ***name***. P trace should be available for the component.

Function is entered as an expression, without equal sign. For example:

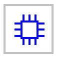
```
sin(t*1000)*(1+cos(t*10))
(t%2>1)?1:-1
mag(x,y)
sq(V(r1))/r1
1/(1+s*R1*C1)
1000*f
```

Editing customized component

When placing a component of **customized** type, an **Edit Component** dialog box shows up:



- Select new dimensions from **Width** and **Height** drop-down boxes to change symbol size.
- Click on rectangle markers to add/remove pins. Depending on component type, position of markers and number of pins allowed may vary.
- Select **Default names** checkbox to create pin names automatically.
- Unselect **Default names** to enter custom pin names in the **Names** box.
- Click **OK** to accept changes and close dialog box, or **Cancel** to ignore changes and close dialog box.

To edit symbol and pins of existing component, select the component, then right-click on the component and select **Edit component** command from context menu, or click **Edit component** button  on the **Components window** toolbar.

Working with Subcircuits

Subcircuit model (**SubCir**) allows creating simple and readable schematic by substituting some part of the schematic with one component (symbol). When simulation starts, the component with **SubCir** model is replaced by actual schematic loaded from subcircuit file. **SubCir** model is available for almost all components. Some **X** components are dedicated for **SubCir** model only: **Block-2**, **Block-3**, **Block-4**, **Block-6**, **Block-8**, and **Custom block**.

SubCir model has the same parameters for all component types:

Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	...		...
	PinN		Name of subcircuit label connected to pin N
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

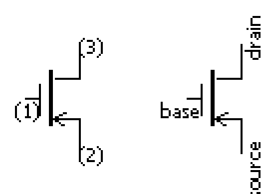
"File" is a file name of subcircuit schematic file. Enter file name manually, or click button and select file name in the dialog box. File extension ".nl5" can be omitted.

Generally, a full path to the file can be used. For example:

```
C:\Program files\NL5\Projects\Modulator\clock_gen.nl5
```

However, if subcircuit file is located in the same directory as a main schematic file, or in the directory defined as "Subcircuit Library" on the **Application** page of **Preferences** dialog box, only "short" file name without path can be used ("`clock_gen`"). The short name is displayed on the schematic. It is recommended to keep subcircuit files in the same directory as a main schematic file or in the "Subcircuit Library", since it allows using short names only and makes the project "portable".

Parameters "Pin1"..."PinN" are used to assign component pins to the labels defined in the subcircuit. If no label name entered for a pin, the pin number in parentheses is displayed on the component image. When label name is defined, it is displayed on the component image. Any of "PinN" parameters can be empty, i.e. not assigned to the label.



"Cmd" is a command string applied to subcircuit schematic when subcircuit is loaded from the file. The string consists of commands in "`name=value`" format, separated by semicolon (";"). This allows using the same subcircuit file with modified values for different components. For example:

```
R1=1k;R2=12k;C1=5n
```

where `R1`, `R2`, and `C1` are subcircuit components.

"IC" is a text string similar to "Cmd" string, but it consists of initial conditions of the subcircuit components. For example:

```
C1.IC=10;O2.IC=0
```

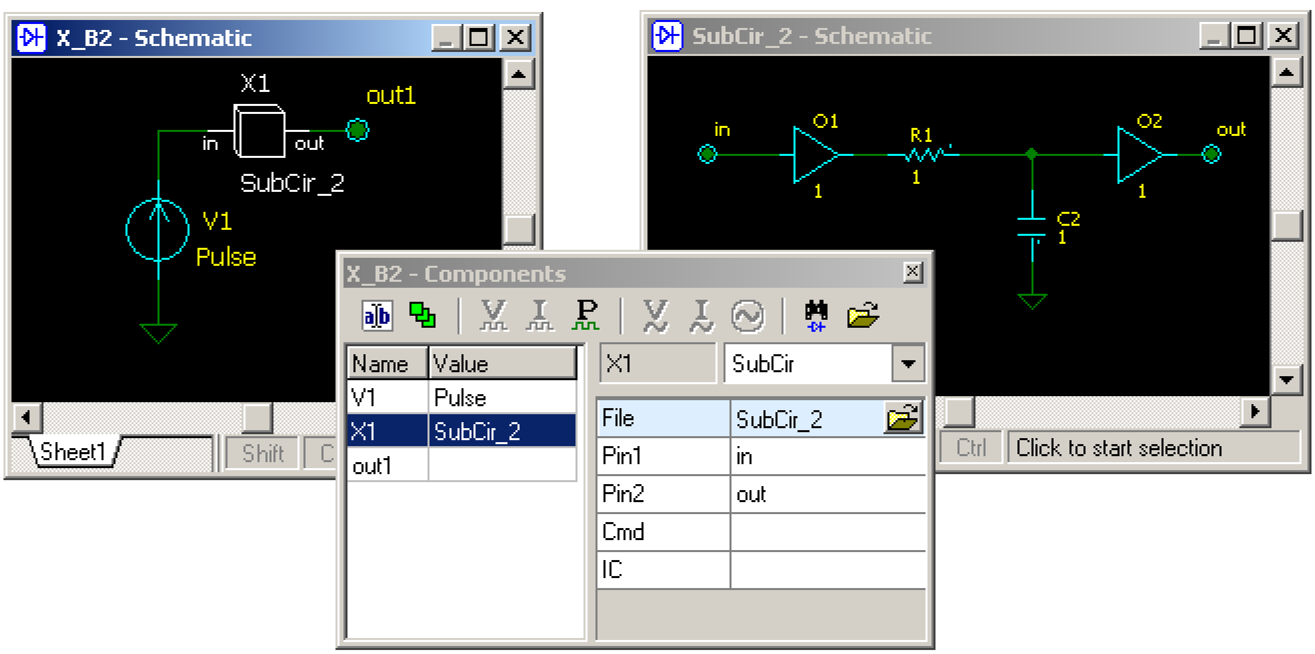
where `C1` and `O2` are subcircuit components. Unlike "Cmd" parameter, "IC" string can be automatically modified by some commands. Command **Transient | Save IC** fills in the string by current IC's of all components in the subcircuit. Command **Schematic | Tools**, **Initial Condition** page, check box **Set subcircuits to empty (no IC)** will clear this string up.

Parameters "Cmd" and "IC" can be edited in the parameter line. To edit parameters in the edit dialog box, click button.

Subcircuit is always loaded from the file when simulation starts. If subcircuit has been modified, it should be saved into the file before running simulation. An exception is if subcircuit and main schematic are opened in the same instance of NL5 application. In this case subcircuit will be taken directly from the NL5 memory, so that saving changes into the file is not required.

An example of using subcircuit:

Schematic **Subcircuit**



Working with PWL

PWL ("PieceWise Linear") model provides piecewise linear characteristic for different types of components. In fact, a component's parameter is defined as a "step-like" function (for instance, resistance as a function of voltage), and resulting piecewise linear component's characteristic is automatically calculated (volt-ampere characteristic). The table below shows "step-like" parameter and corresponding piecewise linear characteristic for components available in NL5:

"Step-like" parameter	PWL characteristic	Component
R(V)	I(V)	Resistor, diode, zener
R(V)	n/a	Voltage controlled resistor
R(I)	n/a	Current controlled resistor
C(V)	Q(V)	Capacitor
L(I)	H(I)	Inductor
K(V)	V(V)	Voltage controlled voltage source, operational amplifiers
K(I)	V(I)	Current controlled voltage source
K(V)	I(V)	Voltage controlled current source
K(I)	I(I)	Current controlled current source

PWL model for resistor ("R") is described below as an example; **PWL** models for other components are similar with just slight modifications.

A "step-like" parameter "pwl" is a string with comma-separated values. It can be can be symmetrical or non-symmetrical.

Symmetrical parameter is defined in the interval from zero to plus infinity; the negative part is symmetrical to positive one. The string format of symmetrical parameter is the following:

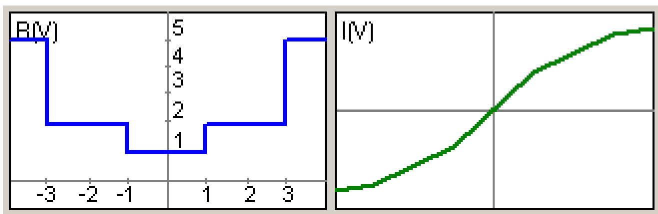
```
R0,V1,R1,V2,R2,…,VN,RN
```

where:

R0 is resistance while absolute value of voltage across the resistor is less than V1.
R1 is resistance while absolute value voltage across the resistor is between V1 and V2.
…
RN is resistance while absolute value voltage across the resistor is greater than VN.

The following R(V) and I(V) graphs represent parameter and PWL characteristic defined by the string:

```
1,1,2,3,5
```



Non-symmetrical parameter is defined in the whole range: from minus to plus infinity. The string format of non-symmetrical parameter is the following:

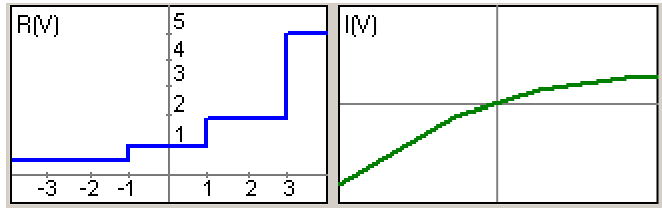
```
n,R0,V1,R1,V2,R2,…,VN,RN
```

where:

character "n" is an indicator of non-symmetrical parameter.
R0 is resistance while voltage across the resistor is less than V1.
R1 is resistance while voltage across the resistor is between V1 and V2.
…
RN is resistance while voltage across the resistor is greater than VN.

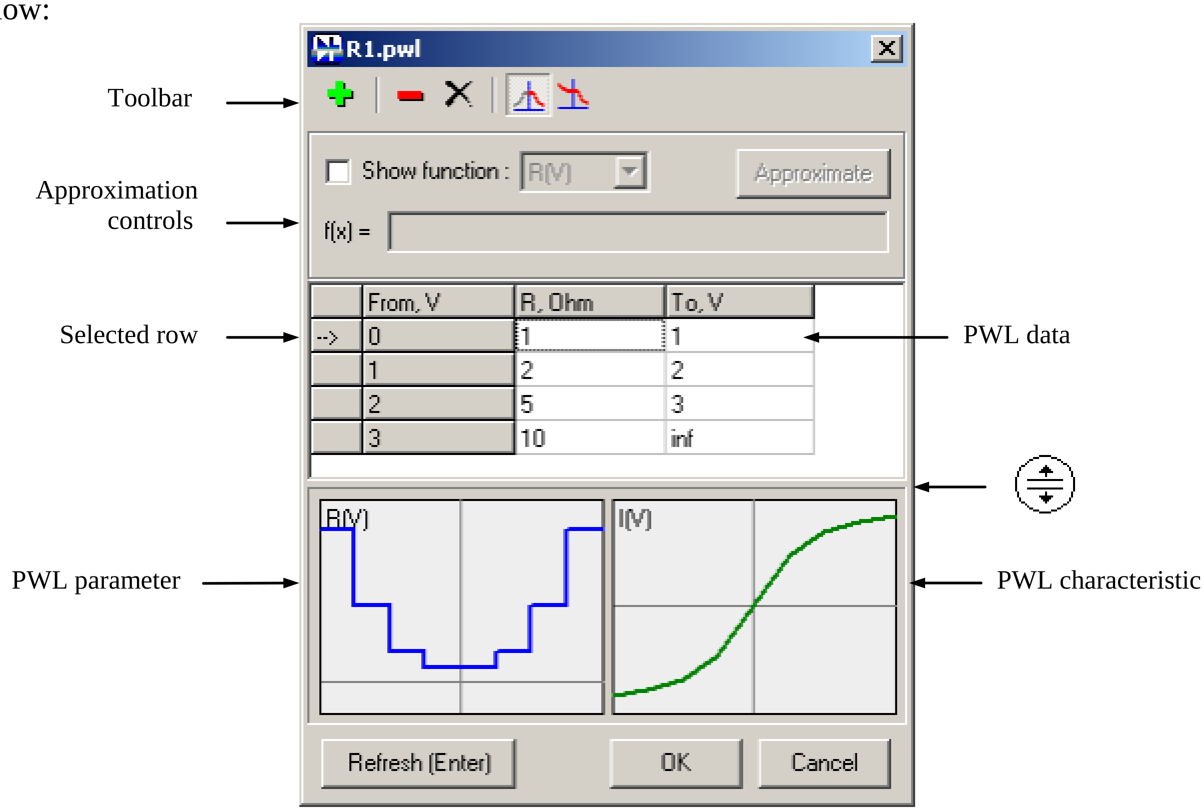
The following R(V) and I(V) graphs represent parameter and PWL characteristic defined by the string:

```
n,.5,-1,1,1,2,3,5
```



PWL characteristic always passes through the origin (0,0 point).

"pwl" parameter can be easily edited in the **PWL** dialog box. Click button right to the "pwl" parameter to open dialog box. Typical view of **PWL** dialog box and its main components are shown below:



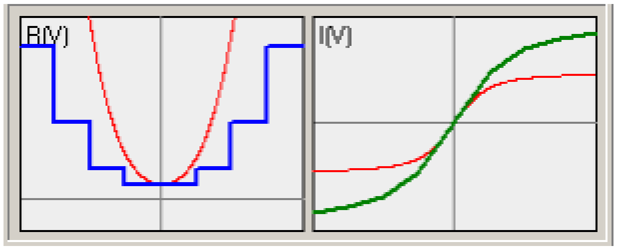
- Click **Toolbar** buttons to perform the following operations:
    - **Split** selected row.
    - **Remove** selected row.
    - **Clear** all data.
    - **Symmetrical** PWL parameter.
    - **Non-symmetrical** PWL parameter.

- **Arrow** in the first column indicates selected row. To select a row click in the PWL data area.
- Move cursor over "splitter" area , then press left mouse button and drag to resize panes.
- Press **Enter** or click **Refresh** button to update graphs.
- Select symmetrical or non-symmetrical type of parameter using toolbar buttons. Note that first "From" value is zero for symmetrical parameter, and "-inf" for non-symmetrical parameter.
- "To" value in the last row is always "inf" (infinity). Enter a number instead of "inf" and press **Enter** to add a new row. Rows will be automatically sorted out.

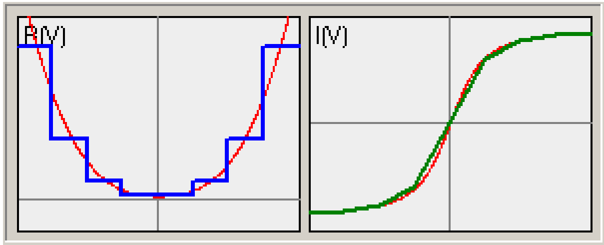
PWL parameter can be automatically approximated to fit an arbitrary function. To use approximation option:

- Select **Show function** checkbox.
- Enter "To" values in the last column of PWL data.
- Select PWL parameter or characteristic (if available) from drop-down list. For example, for PWL resistor select R(V) or I(V).
- Enter **f(x)** as a function of parameter **x**. For example, for I(V) characteristic enter function:

```
x/sqrt(1+x^2)
```



- Press **Enter** or click **Refresh** button to update graphs. If I(V) characteristic is selected, corresponding function for R(V) parameter will be automatically calculated; if R(V) parameter is selected, corresponding function for I(V) characteristic will be automatically calculated.



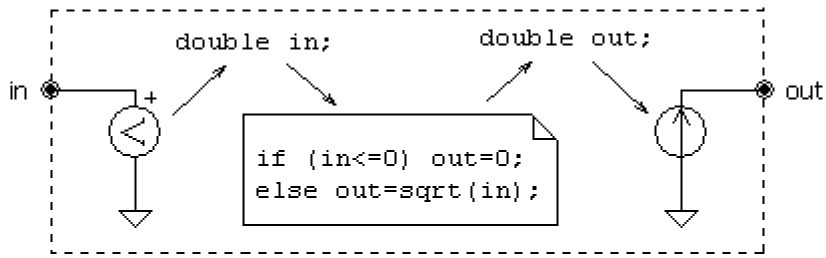
- Click **Approximate** button. PWL parameter will be calculated so that PWL characteristic fits function. Results will be displayed on the graph:

Please note that for better approximation with non-symmetrical parameter, it is recommended to include point "To" = 0 into PWL data.

Creating C-code

In **C** model of the **Code** component, the component function code can be written in simplified C language. The code will be interpreted by NL5 during transient simulation. Although C code interpretation is relatively slow, using **C** model allows very fast iterations of the code. When the code is finalized. it can be compiled and placed in the DLL for faster simulation (see *Creating DLL code* chapter).

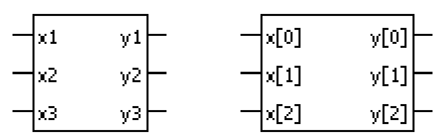
Execution. An example of C-code execution is shown on the following diagram:



A voltage on the input pin of the component ("in") is measured by a voltmeter and is assigned to the variable of the same name `in`. While C-code is executed, a new value of output variable `out` is evaluated. A voltage equal to `out` value is set to a grounded voltage source, connected to the output pin of the same name ("out").

Inputs and outputs. As seen on the diagram, input pins have infinite impedance, and output pins are grounded voltage sources with zero impedance. Variables corresponding to input and output pins are created automatically during initialization phase of the analysis at t=0.

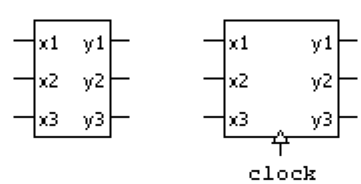
Input and output values are accessed through variables with the same names as pin names (for example `x1, x2, x3…`). Inputs and outputs can also be assigned to array elements (`x[0], x[1], x[2]…)`. In this case, an array (`x[])` of appropriate size should be declared in the initialization code.



Initialization code is executed once at the beginning of simulation at t=0. Initialization code is optional. Leave "Init" parameter of the model blank if initialization code is not used.

Initialization code is used to assign initial values to outputs (output variables), and to declare and initialize global variables and arrays. Global variables can be used for storing algorithm parameters that are calculated once, and then used in the **main** code. Global variables can also be used for storing values that are calculated on one simulation step, and then are used on the next simulation step. If inputs or outputs are assigned to array elements, the array should be declared in the initialization code.

Main code calculates output variables using current values of input variables. If ***clock*** pin does not exist, the code is executed on every calculation step. If ***clock*** pin exists, the code is executed only on rising edge of logical clock signal.



Variables declared in the main code are local, and exist only during execution of the main code. Along with global and local variables, the following **read-only** variables can be used in the main code:

> Component parameters (such as `R1, C2, V.period`, etc.),
> ***t*** - current time
> **V(*name*)** - voltage on the component ***name***
> **I(*name*)** - current through the component ***name***
> **P(*name*)** – power on the component ***name***

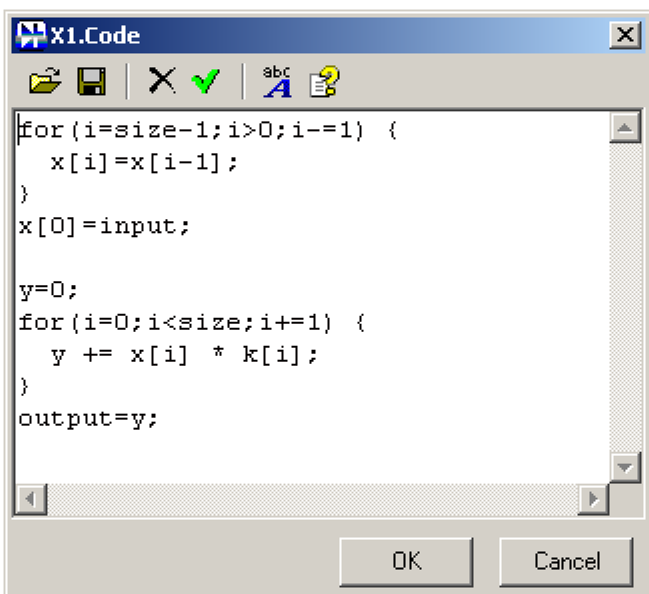
where ***name*** is the name of any component in the schematic. See **Attachments 2, 3, 4** for operators, functions, and C language reference and examples.

Initial conditions. Initial conditions ("IC") is a text string that may contain the code assigning initial values to output variables and global variables defined in the initialization code. For example:

```
y1=1.2;y2=0;y3=2.345;integral=-4.19;counter=100
```

where y1, y2, and y3 are output variables; integral and counter are global variables. "IC" string can be automatically modified by some commands. Command **Transient | Save IC** fills in the string by current variable values. Command **Schematic | Tools**, **Initial Condition** page, check box **Clear C-code IC,** clears this string up. If not empty, "IC" code will be executed after initialization code.

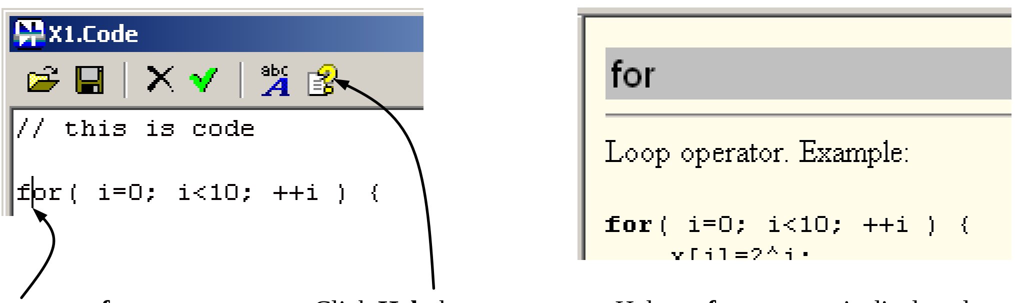
Editing C-code. To edit C-code click button on the parameter line. The following dialog box shows up:



Enter code in the edit area, then click **OK** to accept changes and close dialog box, or **Cancel** to ignore changes and close dialog box.

Click **Toolbar** buttons to perform the following operations:

- **Open code** from the text file.
- **Save code** to the text file.
- **Clear** code.
- **Check** code. A message box with results of the check will be shown.
- **Select font** of the text.
- **Help (F1)**. Click to open Help. To see Help topic on specific statement, operator, or function, place cursor on the word in the code and click the button, or press **F1**.

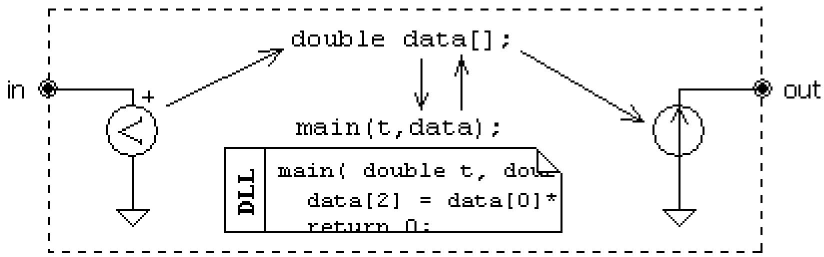


Place cursor on **for** operator — Click **Help** button or press **F1** — Help on **for** operator is displayed

Creating DLL code

In **DLL** model of the **Code** component, the component function code can be written in standard C language, compiled by C compiler, and placed into DLL (Dynamic-linked library). DLL functions will be called by NL5 during transient simulation. DLL code will be executed much faster than C-code in **C** model of the **Code** component, however changing the code requires recompiling the code and rebuilding the DLL.

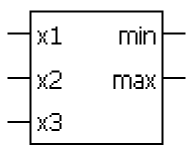
Execution. An example of the DLL code execution is shown on the following diagram:



A voltage on the input pin of the component ("in") is measured by a voltmeter and is assigned to the corresponding element of the array `data`. The pointer to the array is passed as a parameter to the DLL function `main`. The function is executed; a new value of output variable "out" is evaluated and assigned to the corresponding element of the array `data`. A voltage equal to that value is set to a grounded voltage source, connected to the output pin "out".

Inputs and outputs. As seen on the diagram, input pins have infinite impedance, and output pins are grounded voltage sources with zero impedance. Array **`double`** `data[]` is used to pass input/output values to/from DLL function. Array size is <*number of inputs*> + <*number of outputs*>. When calling DLL function, first <*number of inputs*> elements of the array are filled by input values, in the same order how pins are shown on the component symbol (left side, top-to-bottom). Output values calculated in the DLL are placed in the next <*number of outputs*> elements of the array, in the same order how pins are shown on the component symbol (right side, top-to-bottom). For example, if the component has three inputs and two outputs, the following code will assign minimum input value to the first output, and maximum input value to the second output:

```
data[3] = min( data[0], min(data[1],data[2]) );
data[4] = max( data[0], max(data[1],data[2]) );
```



Functions. DLL functions should be declared as follows:

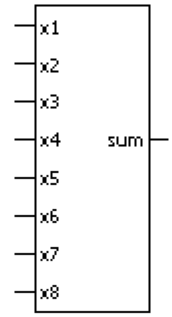
```
extern "C" __declspec (dllexport) int NAME(double t, double* data);
```

where:

`NAME` – function name, for instance `main`
`t` – current time
`data` – pointer to the array with input/output data

Function returns zero if no errors occur, or user-defined non-zero integer error code. The error code will be displayed in the error message window.

For example, function `sum` calculates sum of 8 inputs:

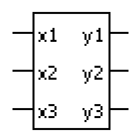


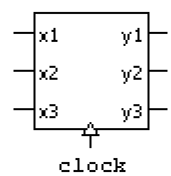

```
extern "C" __declspec (dllexport) int sum(double t, double* x)
{
    double y=0;
    for( int i=0; i<8; ++i ) {
        y += x[i];
    }
    x[8] = y;
    return 0;
}
```

Initialization function is executed once at the beginning of simulation at t=0. Initialization function is optional. Leave "Init" parameter of the model blank if initialization function is not used.

The function is used to assign initial values to outputs (output variables) by setting corresponding elements of the array `data[]`, and to declare and initialize global DLL variables and arrays. Global variables can be used for storing algorithm parameters that are calculated once, and then used in the **main** function. Global variables can also be used for storing values calculated on one simulation step, in order to use those values on the next simulation step. Please note that if several components are using the same DLL, only one copy of DLL is loaded into memory, so that the same global variables will be used for all components.

Main function calculates output variables using current values of input variables. If ***clock*** pin does not exist, the function is executed on every calculation step. If ***clock*** pin exists, the function is executed only on rising edge of logical clock signal.





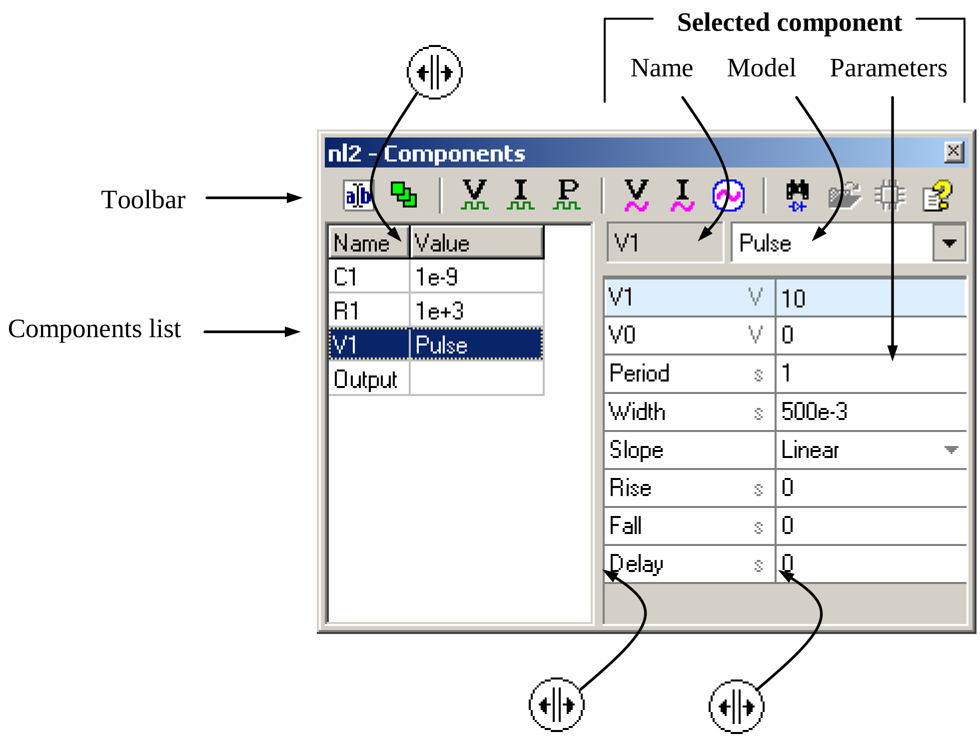
Creating DLL. The code can be compiled, and then the DLL created, using standard C/C++ development tools. Currently, it was tested with Borland C++ Builder and Microsoft Visual C++. Example Borland C++ Builder 6 projects are located in the `Examples/Components/X` folder of the NL5 complete package download zip file.

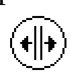
Components Window

To open/hide Components window:

- Select menu command **Window | Components**, or
- Press hot key **F3**, or
- **Double-click** on the component on the schematic.

Components window always shows components of active document (schematic). Typical view of Components window is shown below:

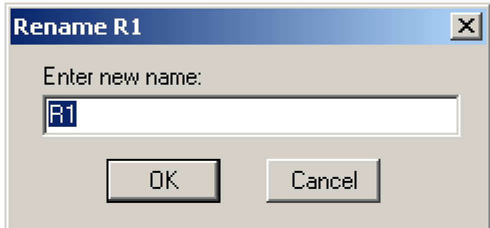


- **Toolbar** provides fast access to often used component-related commands.
- **Components list** shows all components and allows selecting component for editing.
- **Selected component** area shows **name, model,** and **parameters** of a selected component and provides selecting the model and editing parameters.
- Move mouse pointer over "splitter" areas  , then press left mouse button and drag to resize panes and columns.

Toolbar

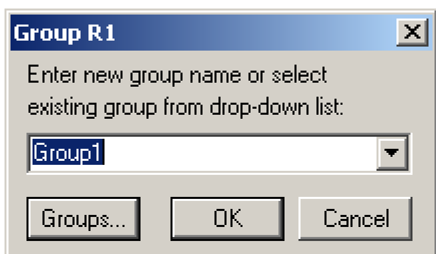
Click toolbar buttons to perform the following operations:

- **Rename a component.** Click to rename selected component. **Rename** dialog box will show up:



Enter new component name and click **OK.** If such a name already exists, an error message *"This name is used by another component"* will be displayed. A new name may consist of any characters and symbols, however it is recommended to use letters and numbers only, and have the name started with a letter. In this case, if the name is used in formula or function, there is no need to enclose it in quotes. The name is case-insensitive. If the component has been renamed, its name will be automatically modified in all appearances of the component name in trace names, formulas and functions.

- **Set a group.** Click to set a group for selected component. **Group** dialog box will show up:



Enter group name or select existing group from drop-down list and click **OK**. Click **Groups** button to open **Groups** dialog box for advanced groups management. To remove existing group ("ungroup" component), open **Group** dialog box, erase group name, and click **OK**. See **Groups** chapter for details.

- **Add transient voltage trace.** Click to add transient voltage trace for selected component.
- **Add transient current trace.** Click to add transient current trace for selected component
- **Add transient power trace.** Click to add transient power trace for selected component.
- **Add AC voltage trace.** Click to add AC voltage trace for selected component.
- **Add AC current trace.** Click to add AC current trace for selected component.
- **Set AC source.** Click to set selected component as an AC source for AC analysis.
- **Find component.** Click to show selected component on the schematic. The component will be selected (highlighted) and centered on the screen.

- **Open subcircuit schematic file.** Click to open subcircuit schematic file. This button is enabled only for components with **SubCir** model, and if subcircuit file name is specified.
- **Edit component**. Click to edit symbol and pins. The button is enabled only for components of customized types.
- **Help (F1)**. Select component and click to open Help topic on component model.

Components list

Components list shows all components and allows selecting component for editing.

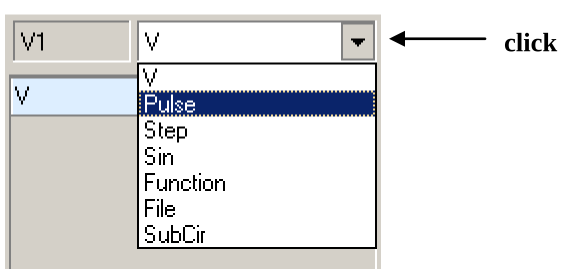
- Column **Name** shows name of the component.
- Column **Value** shows either first parameter of the component, or model name.
- Column **Group** is automatically displayed if at least one component is assigned to a group:

Name	Value	Group
R1	1e+3	Group_R
R2	1e+3	Group_R
R3	1e+3	
C1	1e-9	Group_C
L1	1e-6	
V1	10	

- Click on the list to select a component. The component selected in the list will also be selected (highlighted) on the schematic.
- Press **Enter** or **double-click** on the component to edit component parameters. In this case, if you finish editing parameters by pressing **Enter** or **Esc**, you will switch back to components list.
- Press **Tab** to switch between components list and component parameters.

Selecting Model

Select **model** of a component from a drop-down list by clicking button:



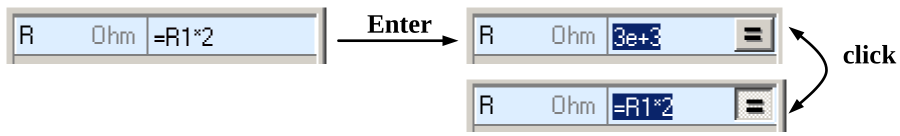
Editing Parameters

Edit component parameters in the parameters list. Click on the line to select the parameter and then use one of the following methods for different parameter types.

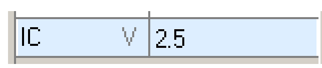
- **Number**.
  - Enter numerical value:

  - Enter expression and press **Enter**. The expression will be evaluated and replaced with numerical value:

  - Enter formula and press **Enter**. The formula will be evaluated and its current numerical value will be displayed. Click = button to view/edit formula:

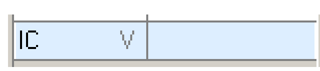


- **Initial Condition (IC**) (voltage, current, charge).
  - Enter numerical value:



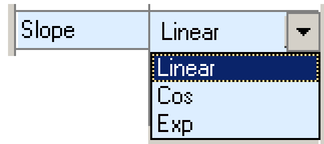
  - Enter expression and press **Enter**. The expression will be evaluated and replaced with numerical value:

  - Leave parameter blank (no IC):



- **Function.**
    - Enter function as an expression with proper variables:

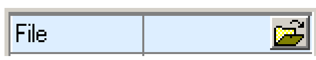
- **List.**
    - Click  button and select value from drop-down list:



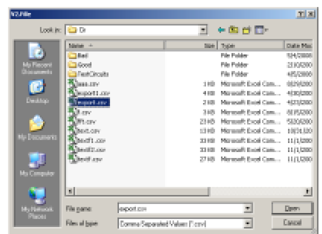
click

- **File name.**
    - Enter file name:

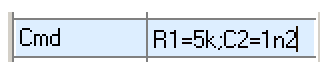
- Click  button to select file in the dialog box:



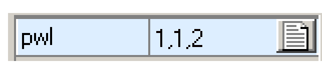
click



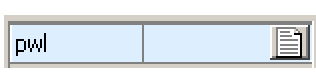
- **Text.**
    - Enter text:



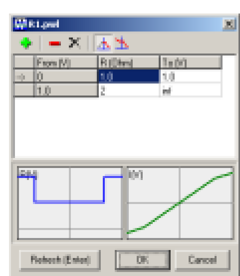
- **PWL (piece-wise linear).**
    - Enter **pwl** parameter manually:



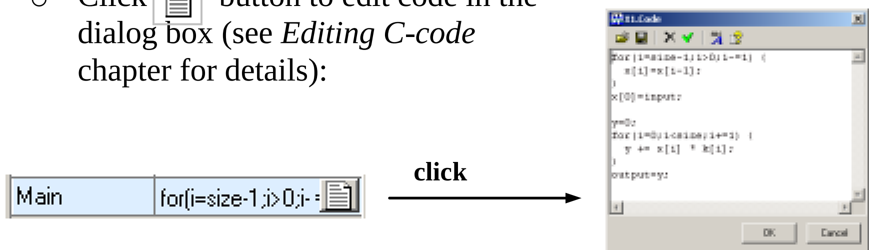
- Click  button to edit PWL in the dialog box (see *Working with PWL* chapter for details):



click



- **C-code.**
    - Click  button to edit code in the dialog box (see *Editing C-code* chapter for details):

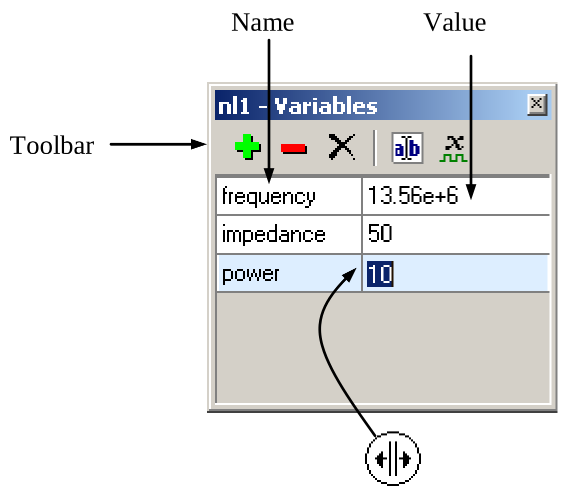


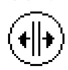
Variables Window

To open/hide Variables window:

- Select menu command **Window | Variables**, or
- Press hot key **F4**

Variables window opens automatically while opening schematic from a file, if schematic has defined variables. Variables window always shows active schematic variables. Typical view of Variables window is shown below:



- **Toolbar** provides access to variable-related commands.
- Variables are shown in the **Name/Value** columns.
- Move mouse pointer over "splitter" area  , then press left mouse button and drag to resize columns.

Toolbar

Click toolbar buttons to perform the following operations:

- **Add a variable**. Click to add a new variable. **Add variable** dialog box will show up:

Enter variable name and click **OK.** If a new name already exists, an error message *"This name is used by another component"* will be displayed.

- **Remove a variable**. Click to remove selected variable.
- **Delete all variables**. Click to delete all variables.
- **Rename a variable.** Click to rename selected variable. **Rename** dialog box will show up:

Enter new variable name and click **OK.** If such a name already exists, an error message *"This name is used by another component"* will be displayed. A new name may consist of any characters and symbols, however it is recommended to use letters and numbers only, and have the name started with a letter. In this case, if the name is used in formula or function, there is no need to enclose it in quotes. The name is case-insensitive. If the variable has been renamed, its name will be automatically modified in all appearances of the variable in trace names, formulas and functions.

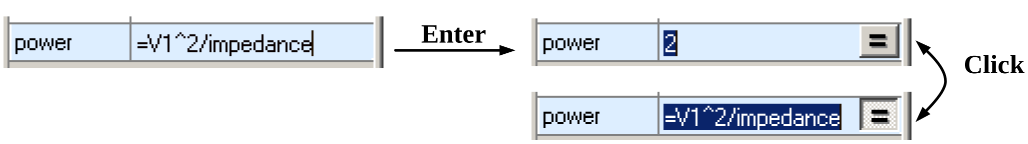
- **Add variable trace.** Click to add transient trace of selected variable.

Editing variable

- Enter numerical value:

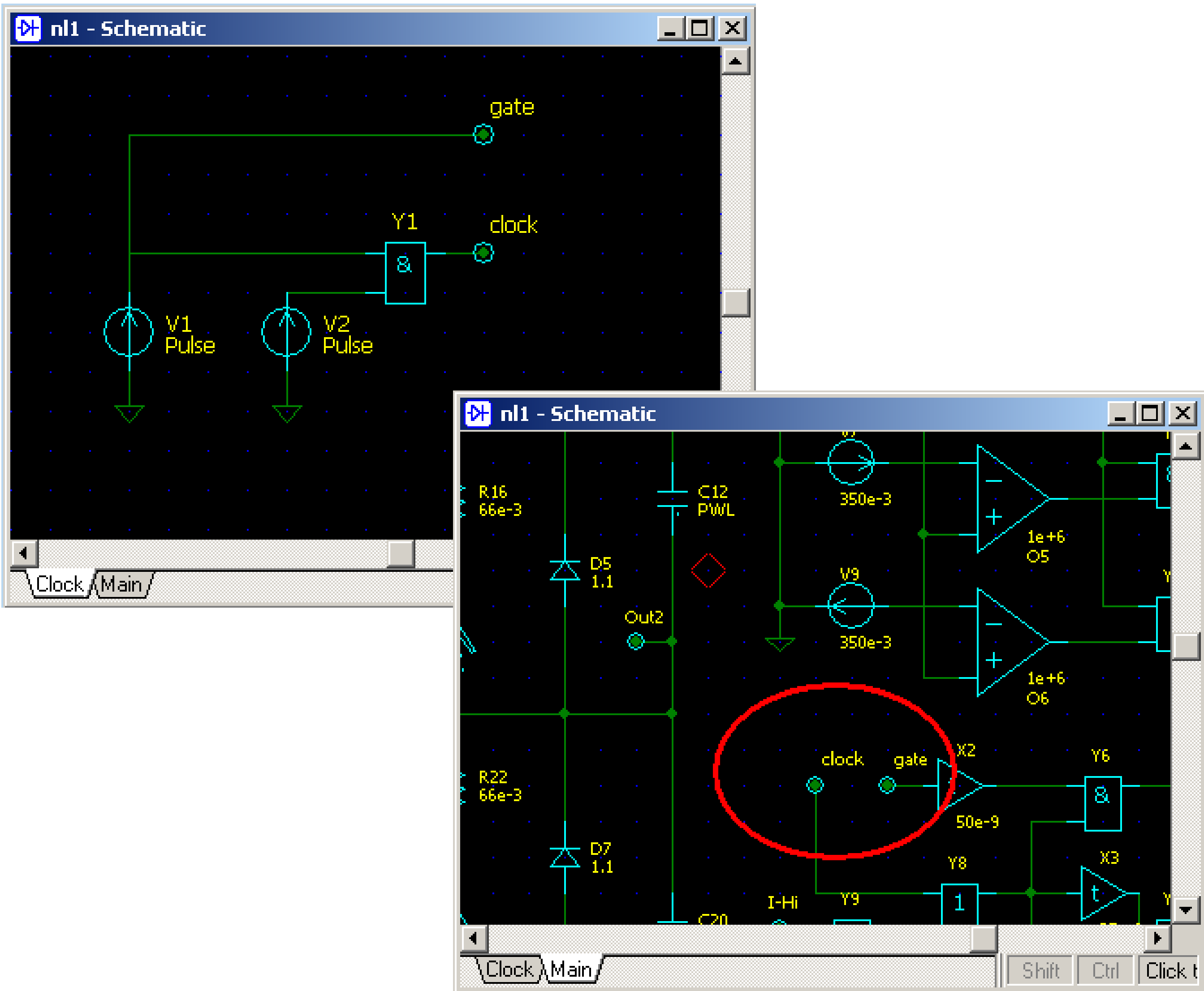
- Enter expression and press Enter. The expression will be evaluated and replaced with numerical value:

- Enter formula and press **Enter**. The formula will be evaluated and its current numerical value will be displayed. Click = button to view/edit formula:



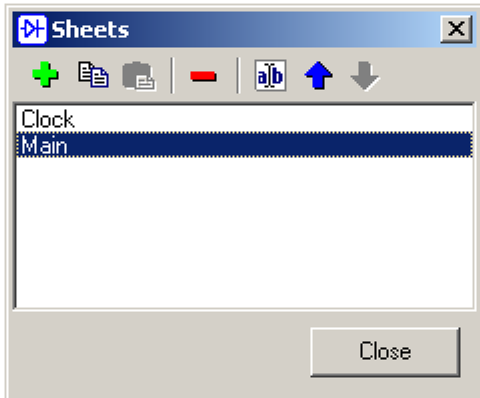
Sheets

Schematic may contain several sheets. Electrical connection between schematic sheets can be done through labels and functions. In the following example, "Clock" sheet consists of pulse generator, and "Main" sheet consists of main schematic. Labels "clock" and "gate" provide electrical connection between sheets:



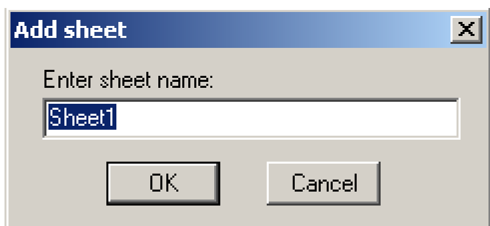
Existing sheets are shown at Sheet selection area of Schematic window. **Right-click** on the selection area to access relevant commands in the context menu, such as: **Add**, **Rename**, **Copy**, **Paste**, **Delete**. Also manage sheets in the **Sheets** dialog box**.**

To open **Sheets** dialog box: select **Schematic | Sheets** command, or right-click on the Sheet selection area and select **Sheets** command from context menu:



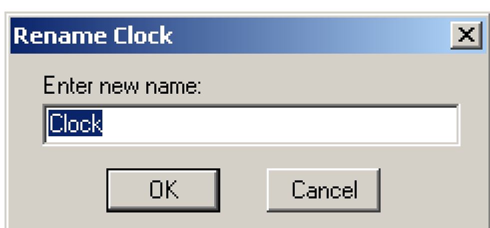
Select sheet in the list and click toolbar button to perform the following operations:

- **Add** new sheet. **Add sheet** dialog will show up:



Enter sheet name and click **OK.**

- **Copy** sheet to clipboard.
- **Paste** sheet from clipboard.
- **Remove** selected sheet.
- **Rename** selected sheet. **Rename** dialog will show up:

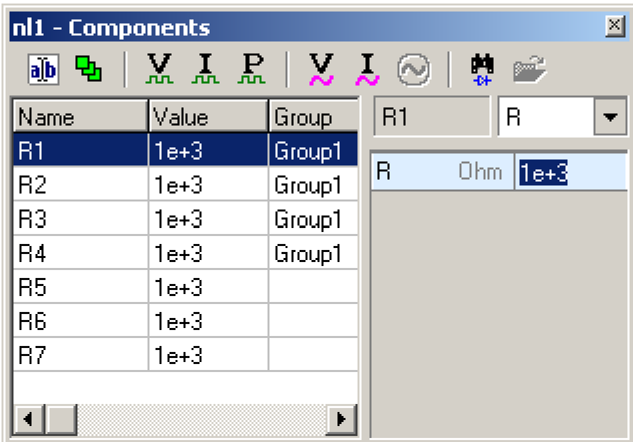


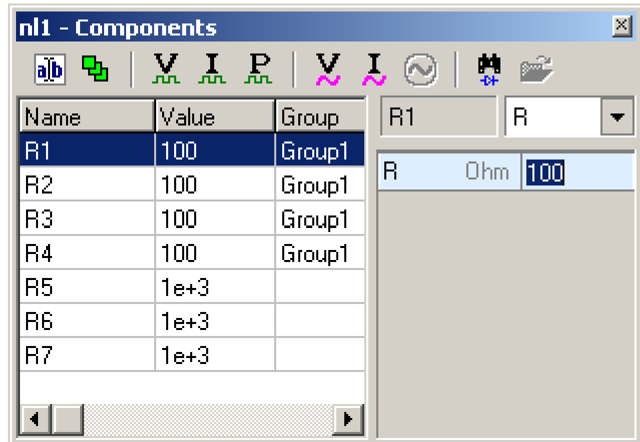
Enter new sheet name and click **OK**.

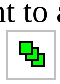
- **Move** selected sheet **up** (or **left** in the Sheet Selection area).
- **Move** selected sheet **down** (or **right** in the Sheet Selection area).

Groups

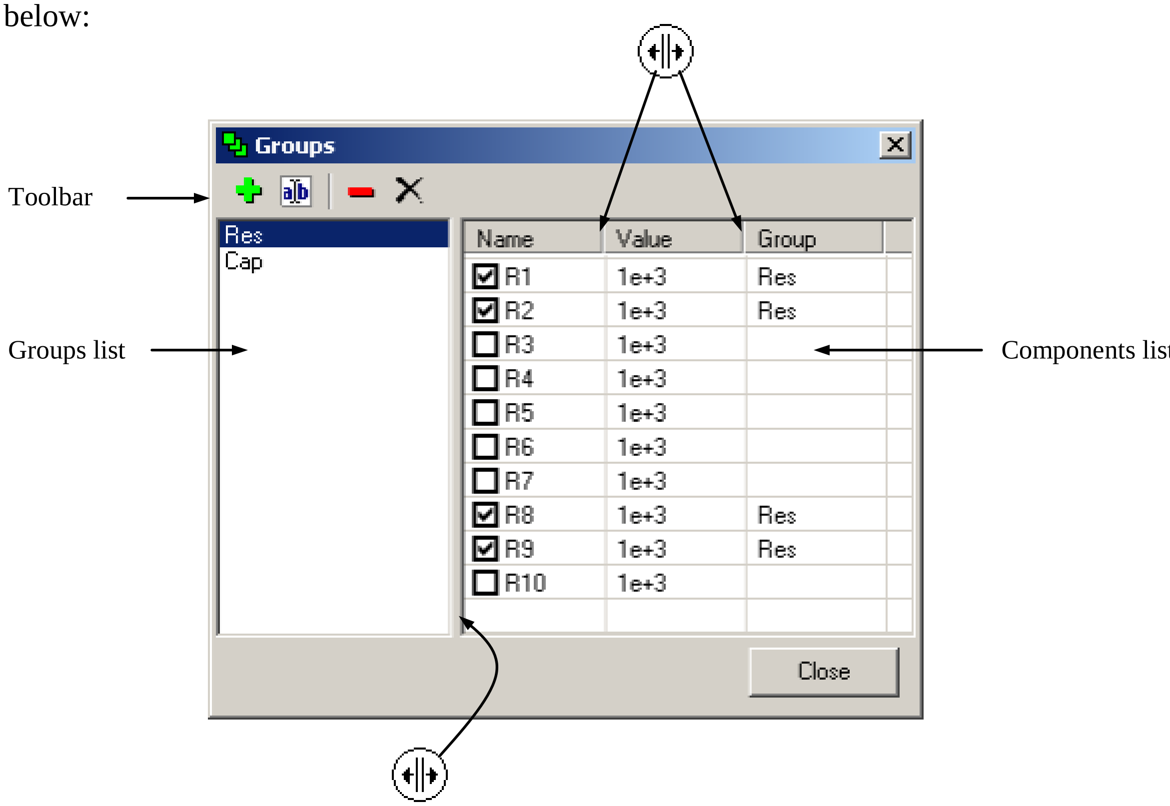
Group is a set of components that always have the same model and parameters. When model or parameter of any component in the group changes, all other components automatically change as well. For example: resistors R1…R4 belong to group **Group1**. Changing R1 value from 1e+3 to 100 automatically changes all resistors in the group.





To assign one component to a new or existing group, select component in the **Components Window** and click **Group** button  . To manage all groups, or assign a number of components to groups, use **Groups** dialog box.

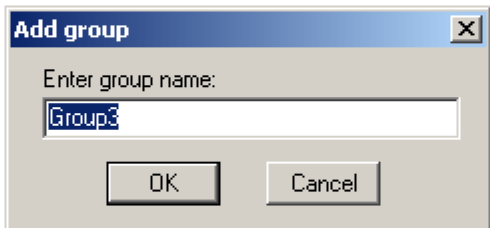
To open **Groups** dialog box select **Schematic | Groups** command. Typical view of the dialog box is shown below:



- Existing groups are shown in the **Groups list.** Click on the group name to display components.
- Components are shown in the **Components list**. The components shown either belong to selected group (have a check mark in the list, and are selected on the schematic), or are of the same type and can be assigned to the selected group.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize columns.

Click toolbar buttons to perform the following operations:

- **Add** new group. **Add group** dialog will show up:



Enter group name and click **OK.** A new group will be added to the Groups list, and **all** the components will be displayed in the Components list. Check a component to assign it to the group. When at least one component is assigned to the group, **only** components of the same type will be displayed in the list.

- **Rename** selected group. **Rename group** dialog will show up:

Enter new group name and click **OK.**

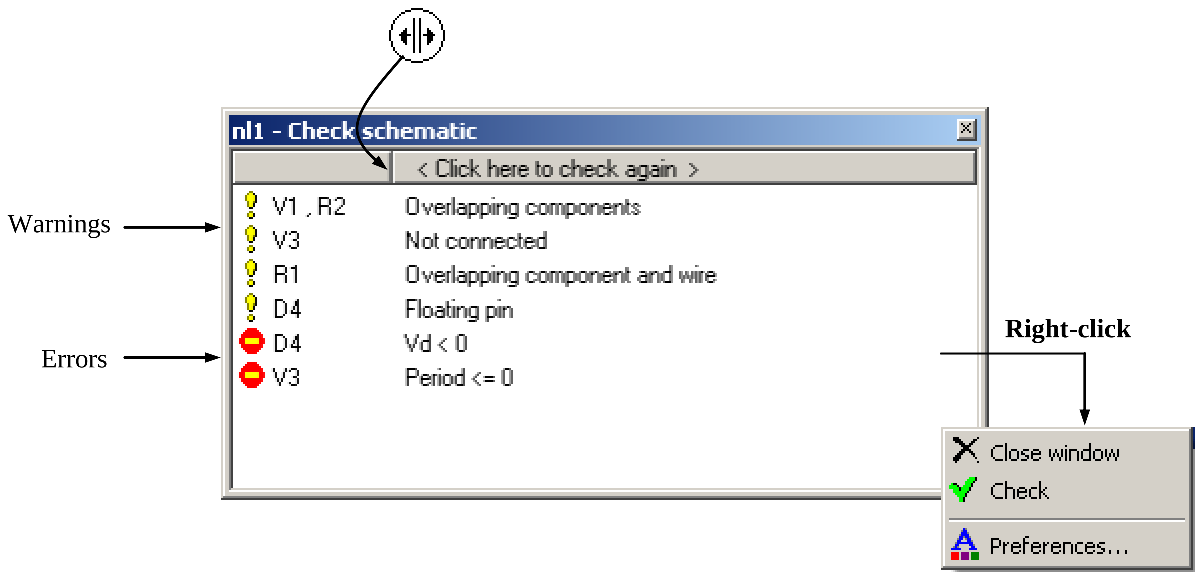
- **Remove** selected group. Components will not be removed!
- **Delete** all groups. Components will not be removed!

Check Schematic

Check Schematic command performs schematic check for potential problems, and component parameters check for errors. To check schematic:

- Select menu command **Schematic | Check**, or
- Click **Check schematic** toolbar button ✔ .
- Check Schematic is automatically performed at Transient and AC analysis start.

Typical view of Check Schematic window and its main components are shown below:



- Messages with **warning** icon (!) notify about potential problems in the schematic, and do not prevent from running Transient and AC analysis. Click on the message line to see problematic schematic elements: they will be selected and centered on the screen. The following potential problems of the schematic are currently detected:
  - **Floating pins.** One or more pins of the component are not connected to another component, wire, or ground.
  - **Non-connected components.** All pins of the component are not connected to another component, wire, or ground.
  - **Overlapping components and wires.** An image of schematic element (component, wire, ground) is overlapping with another schematic element. This may result in non-expected connection, or lack of connection.
  - **Possibly floating schematic.** Schematic does not have any ground or defined potential (voltage source or label) in it. This may cause convergence problems.

Go to **Warnings** page of **Preferences** to disable all warnings or selected warning types from being displayed. Disabled warnings will still be reported in the Transient/AC log, and can be seen by selecting **Transient | Log** or **AC | Log** command.

- Messages with **error** icon ( ) notify about errors in component parameters, such as value out of valid range, or error in formula. If errors exist, Transient and AC analysis will not be performed. Click on the message line to select component or variable with the error. Errors are also reported in the Transient/AC log, and can be seen by selecting **Transient | Log** or **AC | Log** command.
- **Right-click** on the window to see context menu with relevant commands.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize columns.

Schematic Tools

To open **Schematic Tools** dialog box select menu command **Schematic | Tools**, or click **Schematic tools** toolbar button . Select Tools page in the selection list.

Renumber

Renumber components on the schematic. Select **Order** and **Names** option, see example of renumbering in the **Example** window, click **Execute** button to proceed.

"Numerical" name is the name that starts with letter, and the rest of the name is a number. Otherwise the name is considered as "text". Default (automatic) component name is numerical. For example:

`R123, C2` - numerical
`V12V, Rout` - text

Initial Conditions

Set Initial Conditions (IC) of selected component types to specified value. Select component types and IC values, click **Execute** button to proceed. Click **Check all** and **Uncheck all** buttons to check/uncheck all components.

Clean Up

Clean up schematic. Select options, click **Execute** button to proceed. Click **Check all** and **Uncheck all** buttons to check/uncheck all options.

Formulas

Replace all formulas with values. Select option, click **Execute** button to proceed.

Parameters

Set selected parameters of all components to specified value. Select parameters, enter new parameter value, or select from drop-down list, click **Execute** button to proceed.

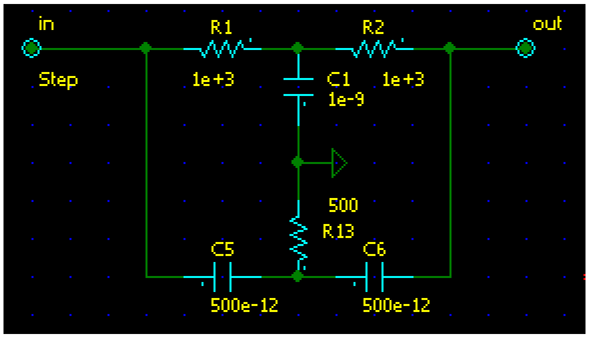
- **Set diodes.** Set **Vd** parameter of all diodes and zeners and/or **Vbe** parameter of all transistors.
- **Set period.** Set **Period** parameter of all **Sin** and **Pulse** voltage and current sources, and **Pulse** switches. **Width**, **Rise**, **Fall**, and **Delay** values will be adjusted proportional to period.

Transform

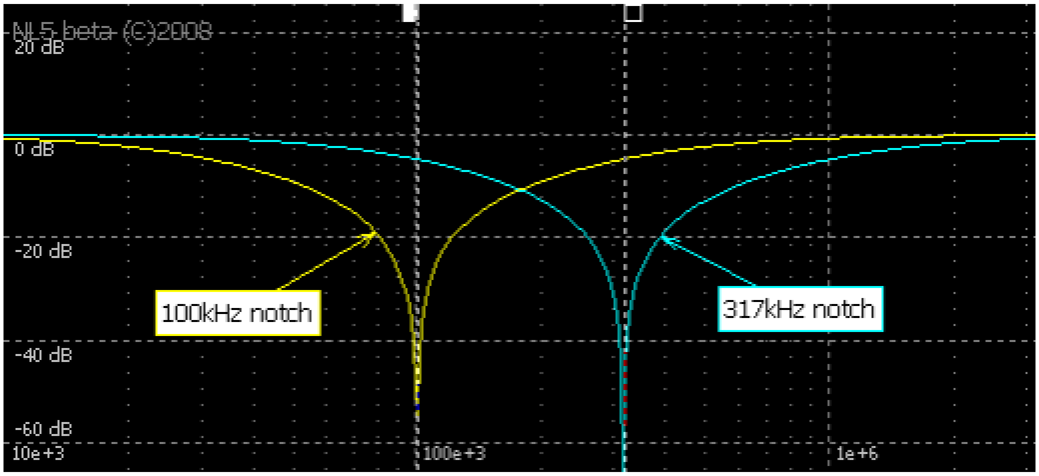
Frequency. Transform frequency response of the schematic by changing R, C, and L values. Enter f1 and f2 values, or place cursors at f1 and f2 frequencies on the AC graph (active cursor at f2). Select one of the following options, then click **Execute** button to proceed:

- **R = const**. C and L will change as follows: `C=C*f1/f2, L=L*f1/f2`
- **C = const**. R and L will change as follows: `R=R*f1/f2, L=L*(f1/f2)`^{2}
- **L = const**. R and C will change as follows: `R=R*f2/f1, C=C*(f1/f2)`^{2}

Example: notch frequency of the filter moved from 317 kHz to 100 kHz, keeping C = const.



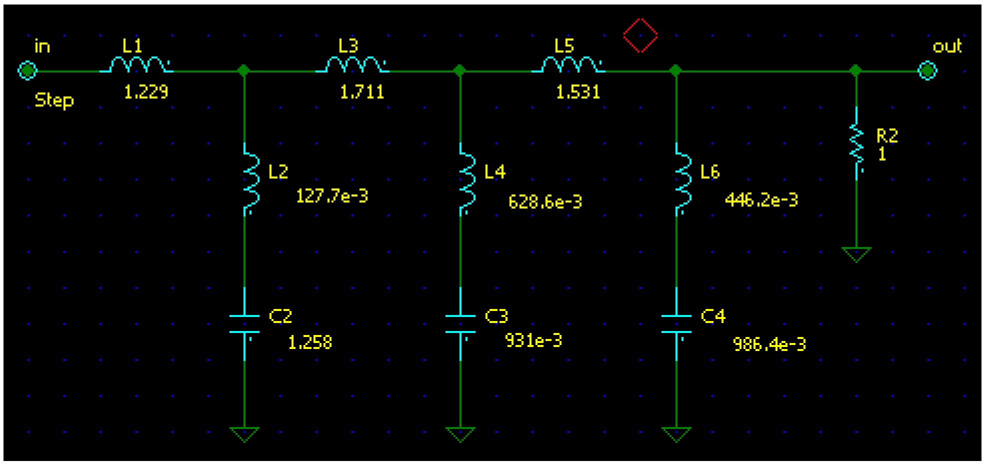
Calculate AC response. Set cursors at notch frequency and 100 kHz (active cursor), open **Schematic Tools** dialog box, **Transform** page, select **C = const**, click **Execute**. Calculate new AC response. Notch frequency has moved to 100 kHz.



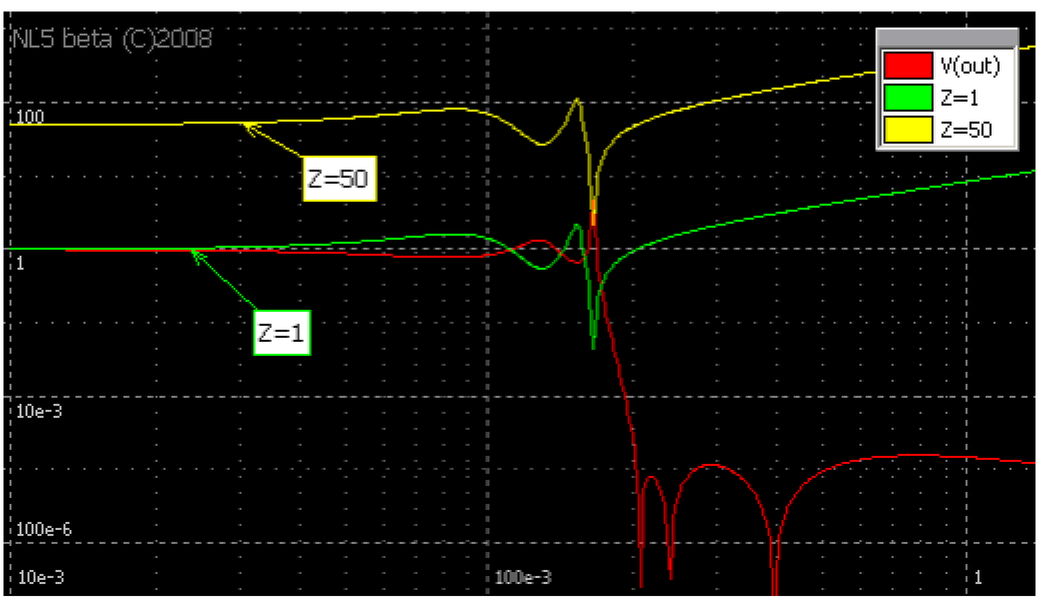
Impedance. Transform schematic impedance from r1 to r2 by changing R, C, and L values. Enter r1 and r2 values, then click **Execute** button to proceed. R, C, and L values will change as follows:

```
R=R*r2/r1
C=C*r1/r2
L=L*r2/r1
```

Example: change characteristic impedance of the filter from 1 to 50 Ohm.

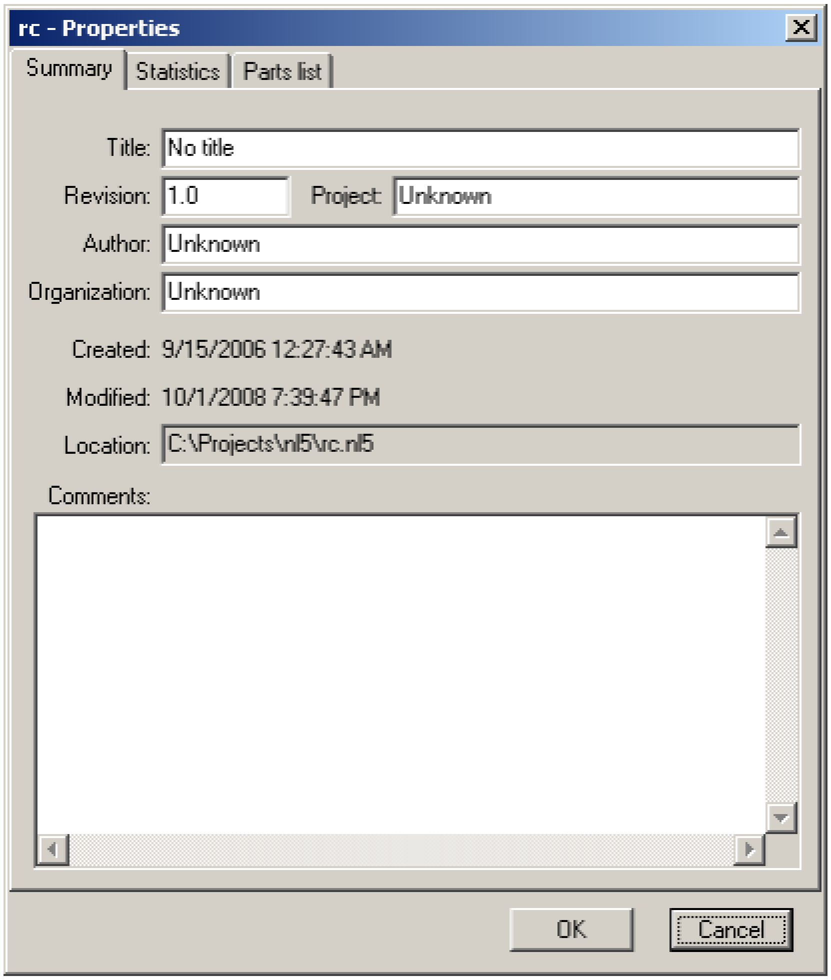


Calculate AC response and input impedance. Open **Schematic Tools** dialog box, **Transform** page, enter **r1** = 1 Ohm and **r2** = 50 Ohm, click **Execute.** Calculate new AC response and input impedance. AC response is the same, input impedance has changed as requested.



Properties

Select **File | Properties** command. **Properties** dialog box will show up:

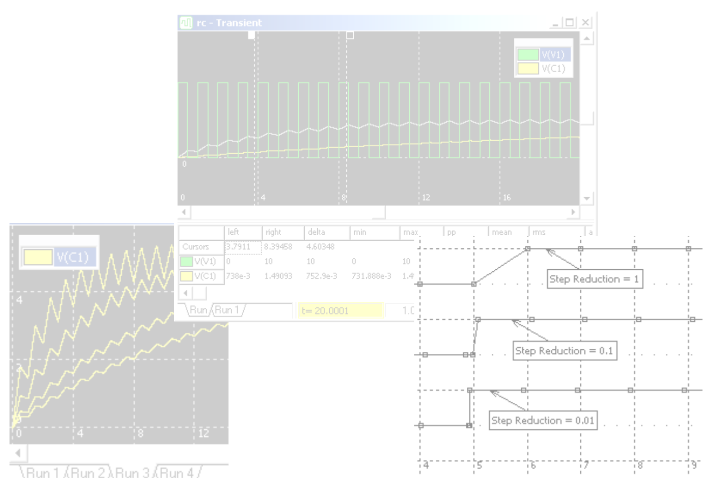


Summary page shows general document and file information. Fields **Author** and **Organization** of a new schematic are set to the values specified on the **Document** page of **Preferences** dialog box. Most of the fields can be edited.

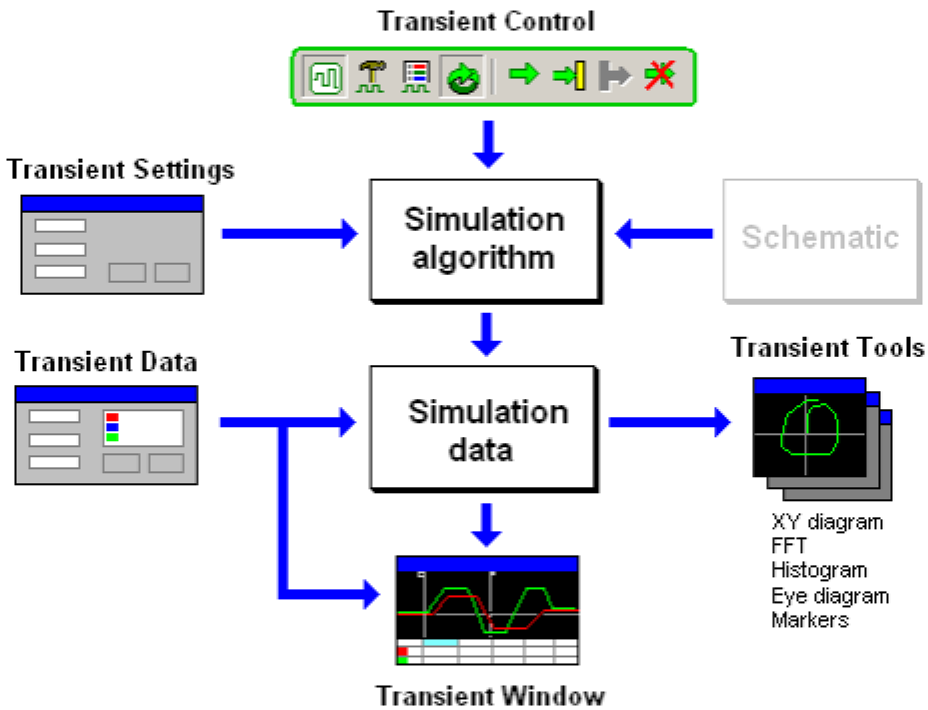
Statistics page shows schematic statistics information.

Parts list page shows components list in short or detailed format. Click **Copy to clipboard** button to copy parts list to the clipboard.

IV. Transient Analysis



The following simplified diagram explains transient simulation process:



Simulation algorithm is configured by **Transient Settings** dialog box, and controlled by **Transient Control** commands (Main Menu and Toolbar). The results of **Schematic** simulation are stored into **Simulation data**, and simultaneously displayed as a graph in the **Transient Window**. **Transient Data** window is used to configure what simulation data are to be stored, and how the data are displayed. In addition, the data can be used by **Transient Tools**, which offer a variety of transient data analysis and data presentation.

Simulation

Simulation algorithm

NL5 is a piecewise linear (PWL) simulator. All the components in the NL5 are either linear, or piecewise linear: consisting of a number of linear segments. For instance, a diode is either open or closed, so that its PWL representation consists of just two segments. As long as all of the components are staying within their current linear segment, the schematic is described by the same system of linear differential equations. The system is modified only at the moments when at least one component changes its linear segment. When this happens, the current linear range simulation ends, and another one starts. Typical NL5 simulation consists of DC operating point calculation (at t=0), followed by one or more linear range simulations. The performance of the algorithm can be optimized by a number of parameters, located in the **Transient Settings** and **Advanced Settings** dialog boxes.

DC operating point. Simulation always starts at t=0. First, Direct Current (DC) operating point is calculated. The calculation is performed considering Initial Condition (IC) of the components. For instance, capacitor is replaced by voltage source if IC voltage is specified, or ignored (open circuit) if IC is not specified (blank). Inductor is replaced by current source if IC current is specified, or by a short circuit if IC is not specified. Diode is considered an open circuit if IC state is "Off", and short circuit if IC state is "On".

If schematic has more than one steady state, it can be set to a specific state by defining proper ICs. Another way to do that is using label (model **Label**), and specifying **VIC** parameter of the label. If **VIC** is not blank, the temporary voltage source **VIC** is connected to the label through resistor **R** only during DC operating point calculation. When calculation is done, voltage source is removed.

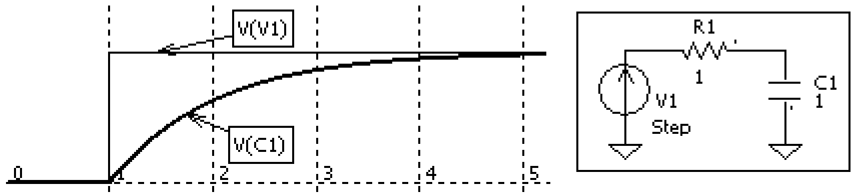
The result of DC operating point calculation is known voltages, currents, and states of all components. When DC operating point is found, a first linear range simulation starts.

Linear range simulation. In the linear range, schematic is described by the system of linear differential equations, which is solved by Trapezoidal integration method. The method provides sufficient accuracy with reasonable robustness and calculation speed. During linear range simulation, the algorithm is performing "switching point detection": checking conditions on all components that may change their state (diodes, switches, logical components), linear segment (**PWL** models), or change the amplitude or slope (**Pulse** and **Step** models). If any change occurred, the current linear range ends, and another one starts.

Calculation step. Unlike many analog simulators, NL5 does not perform automatic step control. Selecting calculation step is user's responsibility. This gives user full control on simulation, although requires certain experience and understanding of the process. The rule of thumb is keeping calculation step below smallest time constant in the schematic, otherwise the integration method may get unstable and produce "numerical" oscillations. NL5 detects such oscillations and displays warning message: in this case it would be useful to investigate the problem and either reduce calculation step or ignore oscillations as nonsignificant.

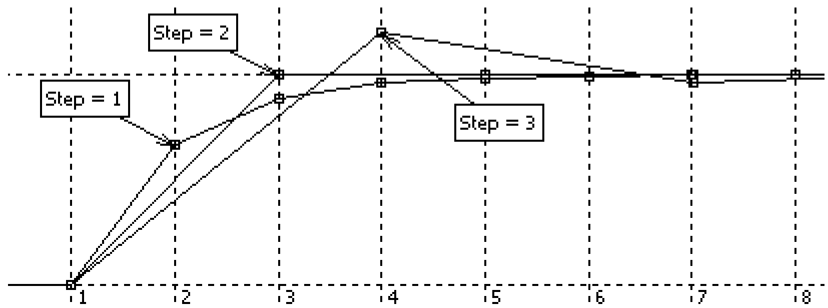
However, having calculation step to satisfy "smallest time constant" condition is not necessarily required. Sometimes even high enough calculation step provides good stability, while simulation speed can be significantly increased. To find out an optimal calculation step, run simulation several times with different step and compare simulation results. As a rule, reducing calculation step below some level does not have any visible effect on results. Selecting calculation step close to this level would give the best simulation performance.

The following example shows how calculation step affects simulation of a simple schematic. The time constant of the RC chain is 1s, so that calculation step is supposed to be < 1s.



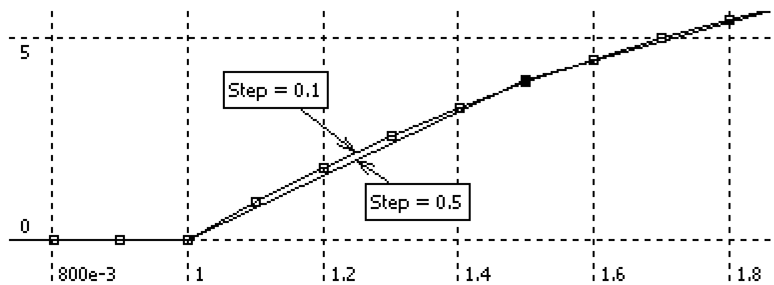
Ideal response

When 1s, 2s, and 3s step is selected, the transient waveform is wrong. The overshoot and further oscillations exist at 3s step. However, if exact waveform is not of interest, and if it does not affect functionality of the rest of schematic, these steps could be used. The "numerical oscillation" warning can be turned off by **Do not detect oscillations** checkbox.



Simulation with large steps

Calculation steps below 1s produce very accurate waveforms. For instance, difference between traces with 0.5s and 0.1s steps can be noticed only at the very beginning of the transient, and it is extremely small.

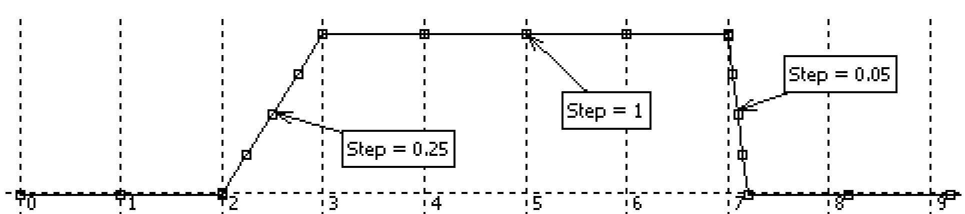


Simulation with small steps

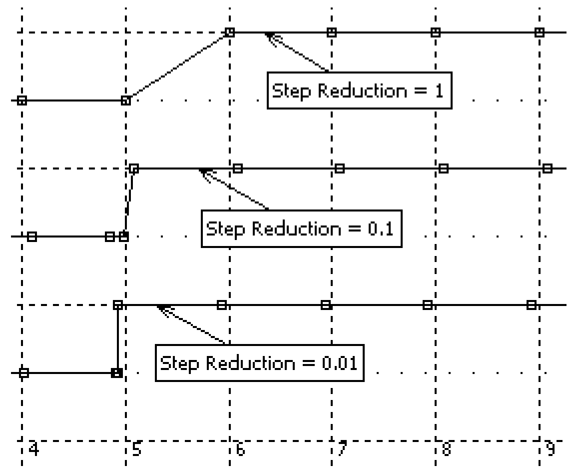
Automatic step reduction. Although calculation step is specified by user, NL5 still can automatically reduce the step to satisfy the following conditions:

- Period of sine source contains at least 16 steps.
- Pulse or switch "On"/"Off" state contains at least 4 steps.
- Non-zero rising or falling edge contains at least 4 steps.
- Interval between two points in models, which perform interpolation of the signal between those points, contains at least 4 steps.
- Delay time of transmission line and "delay" component contains at least 2 steps.

The following example shows how the calculation step is reduced during the pulse edges:



Automatic step reduction may also be used to provide better time resolution of switching point detection. If time constants of the circuit are large, and high calculation step can be used for linear range simulation, reducing the step only at switching points may significantly improve simulation performance. The **Step Reduction** parameter specifies how much step resolution during switching point detection is better than specified calculation step.

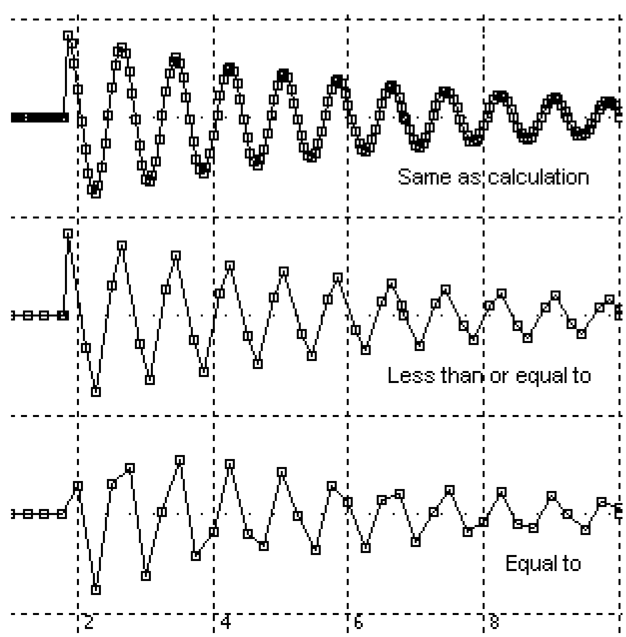


For instance, **Step Reduction** = 0.1 means that switching points will be detected with time resolution approximately ten times better than calculation step. The following graph shows waveforms obtained with calculation step = 1s, and step reduction 1, 0,1, and 0.01.

Using automatic step reduction does not affect calculation speed much. Number of additional calculation steps is approximately equal to -$\log_2$(**Step Reduction**). For step reduction = 0.01, there will be only 6 additional steps.

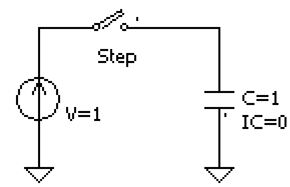
However, if constant calculation step is preferred, any change of calculation step can be disabled. For example, **Function** model of some components has output signal always delayed by one calculation step. If step is constant, this can be considered as a known constant delay, which can be properly taken care of. If calculation step is changing, the resulting variable delay might be really hard to deal with, especially in the systems with closed loop. Select **Do not reduce calculation step** check box to disable automatic step reduction.

Data sampling step. For accurate simulation, the calculation step may be required to be very small. However, storing all the simulation data in the memory is not needed, if the signal of interest is smooth and changing relatively slow. **Data sampling step** options allow performing simulation with as small step as needed, but storing only part of the data, thus saving significant amount of memory. The following options are available:

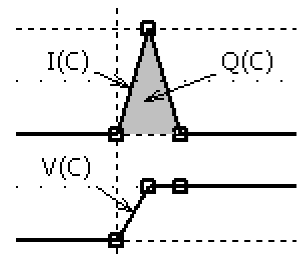


- **Same as calculation (save all data).** All the calculated data are saved into memory. This option provides most accurate data display and analysis with the highest memory consumption.
- **Less than or equal to.** If selected, a maximum sampling step should be entered as well. The data are saved with specified sampling step. In addition, it saves all the "critical" data points, such as extremes (max and min), sharp edges, switching points, etc. This provides sufficient memory saving with yet reliable displayed data.
- **Equal to.** If selected, a sampling step should be entered as well. Data will be saved with the constant sampling step only, which may give significant memory saving. However, some important details of the transient may be lost, as well as risk of aliasing exists for fast changing signals.

Handling infinite voltage and current pulses. Unlike many Spice-based tools, NL5 is capable to simulate schematics with **true ideal components**. An example of such a component is an ideal switch, which has zero impedance when closed, infinite impedance when open, and instantaneous switching from one state to another. When an ideal switch is used for charging or discharging capacitors, an infinitely short current pulse with infinite amplitude may occur. Although amplitude of the pulse is infinite, the area (integral over time) is limited and is equal to the total charge coming to or out of the capacitor at the moment of switching. Similar situation may occur when current through the inductor is discontinued, which results in an infinite voltage pulse across the inductor. Integral of the voltage over time corresponds to a magnetic flux in the inductor.

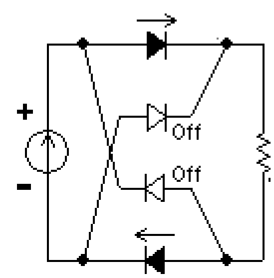


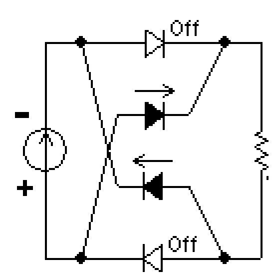
Such an infinitely short pulse with infinite amplitude and limited area is usually referred to as a Dirac pulse, or delta-function. Since showing true delta-function on the transient graph would be problematic, the following approach is implemented in NL5. The current or voltage delta-function is shown as a triangle pulse with the duration of each slope equal to minimal calculation step used at that moment, and the area satisfies charge or magnetic flux conservation law. If calculation step is reasonably small, the displayed pulse will be short and will have high enough amplitude to be visually considered as a delta-function. At the same time, the integral of the pulse will give true value of the charge (for current pulse) or magnetic flux (voltage pulse). Changing calculation step will change duration and amplitude of the displayed pulse, however the integral of the pulse will stay the same.



Convergence. In Spice-based simulators convergence problem may occur any time: during DC analysis and transient analysis. Since NL5 is piecewise linear simulator, most of the time it is dealing with linear systems, which do never experience convergence problem. The only time NL5 simulation may have some difficulties, is the moment when one or more components change their state or linear segment.

For systems with ideal piecewise linear components, a typical situation is when several components have to change states exactly at the same moment, otherwise the system won't converge. For example, in a standard four-diodes bridge rectifier, diodes are always switching by pairs, or even all diodes at a time. With ideal diodes having zero resistance when closed, and infinite resistance when opened, a simple algorithm may have some trouble resolving switching process. Possible solution would be adding non-zero resistors in series and/or large resistors in parallel to the diodes. However, this may produce very small time constants, which results in very small calculation step, so that all the benefits of using ideal components vanish.





Since traditional iterative methods do not work reliable for such systems, NL5 uses robust proprietary algorithm. So far, the algorithm works perfect for all schematics tested, however nobody could prevent users from designing something special, which may have convergence difficulties.

Another problem, common for any software that uses floating-point arithmetic, is loss of accuracy due to rounding errors. Those errors may also affect convergence at switching points, as well as linear range simulation.

If simulation slows down at switching points, stops with "*No solution*" error message, or if simulation produces obviously wrong results, the following options and parameters may help:

- **Change states one at a time**. Selecting this checkbox may improve convergence at switching points.
- **Machine precision**. (AKA "machine epsilon"). This parameter specifies the minimum relative difference between two floating point numbers, which can be reliably recognized. This value affects not only convergence at switching points, but overall simulation results, and can be changed in really wide range (1e-6…1e-15).

Simulation data

Traces. During simulation NL5 is storing data into memory. The data to be stored is selected by user as **traces** in the **Transient Data** window. Several types of transient traces are available: **V** (voltage), **I** (current), **P** (power), **Variable**, and **Function**.

When simulation starts, all traces are automatically cleared, and then start storing new simulation data. A new data is displayed in the **Run** tab of **Transient Window.** The last data can be moved into **storage** with special tab in the Transient Window assigned to it. Storage data is not automatically cleared, and can be used for comparing results of different simulation runs.

If special option **Store last Run** is selected, then when new simulation starts, current simulation data (**Run**) will be moved into storage under the name **Last**. If run with the name **Last** already exists, it will be overwritten. Thus, previous simulation data will always be in the storage under the name **Last** and can easily be compared with the current simulation data.

Traces can be copied to clipboard, saved into "nlt" data file, or exported into text file in "csv" format. In turn, the data can be pasted from clipboard, loaded from "nlt" data file, or imported from text file as a new trace. Such a trace is always displayed in the **Transient Window**, regardless what data tab is selected. It is not cleared when new run starts, and can be used as a reference trace for simulation. It also can be renamed to arbitrary text.

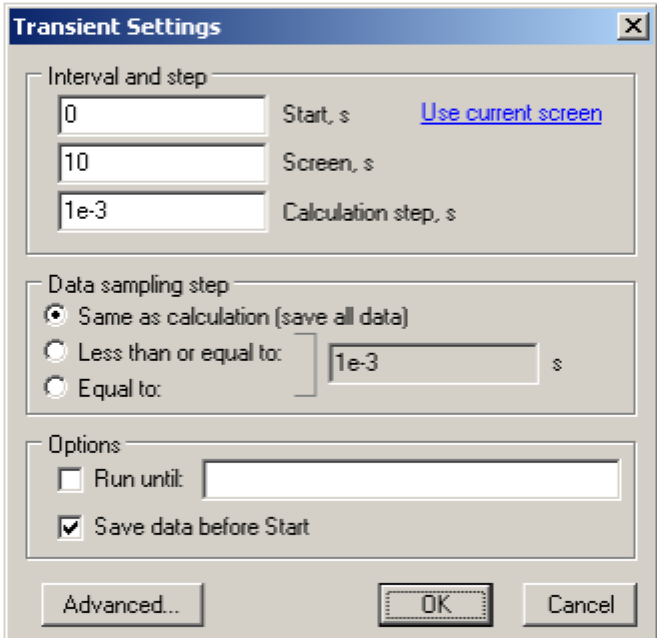
Memory. Simulation data is stored in the operating memory. The memory is allocated as needed by relatively small blocks. If available operating memory is not enough for storing continuously increasing amount of simulation data, the operating system starts saving data to the disk, which may slow down simulation and display significantly. To prevent from this, the following mechanism is used: when amount of memory required for the trace exceeds the maximum value specified on **Transient** page of **Preferences** dialog box, the block of the memory currently storing the very beginning of the trace will be released and allocated again for the new data. Thus, the trace will be truncated at the beginning in order to keep the latest data. At the first time this happens for one or more traces, the warning message will be displayed in the status bar of **Transient Window**.

When trace memory has been truncated, the trace may not be immediately updated on the graph: the graph will be showing "non-existing" data until the trace is redrawn.

The total amount of memory currently used for simulation algorithm and all traces is always displayed in the **Memory used** field of **Transient Window** status bar, so that user can take proper action if needed.

Transient Settings

Click **Transient settings** toolbar button , or select **Transient | Settings** command. **Transient Settings** dialog box will show up:



Interval and step. When simulation starts, the transient window time range is automatically set to specified interval (**Start** and **Screen**).

- **Start, s**. Left edge of the transient window.
- **Screen, s**. Transient window size.
- **Calculation step, s.** Maximum calculation step. Actual step can be reduced by the algorithm as needed.
- **Use current screen**. Click to use current transient screen settings as a new simulation interval. **Start** and **Screen** parameters will be set according to what is currently displayed on the transient graph.

Data sampling step. Specify data sampling (saving) step equal or different than calculation step. This option **does not** affect calculation, it only reduces amount of stored data.

- **Same as calculation (save all data).** All the calculated data are saved into memory.
- **Less than or equal to.** If selected, a maximum data sampling step should be entered as well. When possible, the data are saved with specified data sampling step. In addition, all the "critical" data points, such as extremes (max and min), sharp edges, switching points, are saved.
- **Equal to.** If selected, a data sampling step should be entered as well. Data will be saved with a constant step.

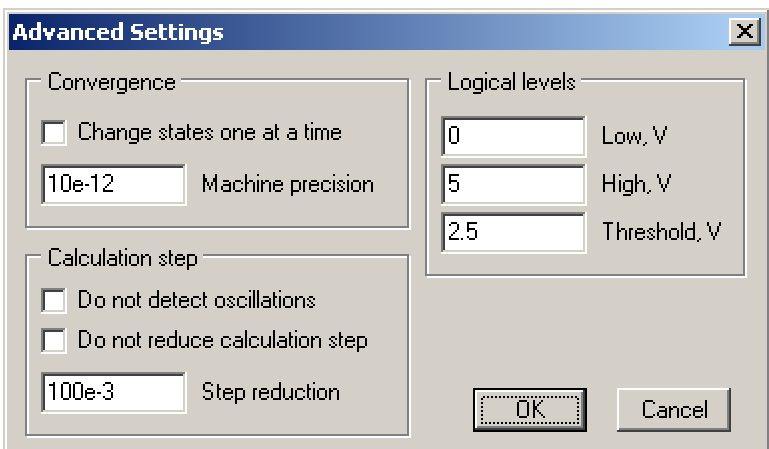
Options.

- **Run until.** If selected and expression is entered in the window, the expression will be evaluated at each step of transient simulation, and simulation will be paused immediately as soon as result of the expression is positive value. Then simulation can be continued. Expression may use variable **t** – current simulation time, voltage, current, and power at any component in the form: **V(*name*), I(*name*)**, and **P(*name*)**, where ***name*** is the name of the component (V, I, or P traces should be allowed for the component). The expression will not be evaluated while **t < Start**. Examples of **Run until** expressions:

```
V(C1)>5.0
(I(R2)>1m)&&(t>10)
(P(Rload)>3.3)||(t>100)
```

- **Save data before Start**. If selected, all the simulation data prior to **Start** are stored into memory and available for display. Otherwise, the data prior to **Start** are lost, providing some memory saving.

Advanced. Click to open **Advanced Settings** dialog box:



Convergence. Parameters that may affect convergence of DC operating point calculation and switching points calculation.

- **Change states one at a time.** Specifies switching iteration mode.
- **Machine precision** ("machine epsilon"). The minimum relative difference between two floating point numbers, which can be reliably recognized.

Calculation step. Some options related to calculation step.

- **Do not detect oscillations.** Do not display warning messages if "numerical" oscillations detected.
- **Do not reduce calculation step.** Always use specified calculation step only.
- **Step reduction.** Specifies how much step resolution during switching point detection is better than calculation step.

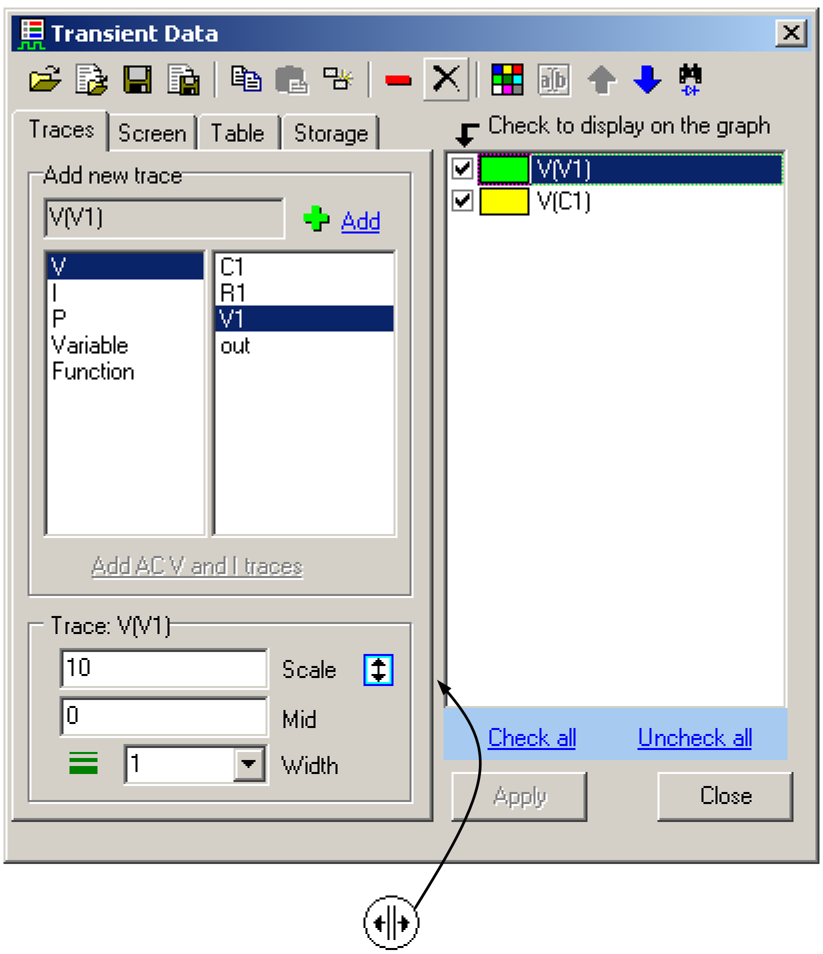
Logical levels. These settings apply to logical components, and some models with logical type of input.

- **Low, V.** Low logical level. Must be < **High**.
- **High, V.** High logical level. Must be > **Low**.
- **Threshold, V.** Logical threshold: the voltage below threshold is considered **Low**, above threshold is considered **High**. Threshold must be between **Low** and **High**.

Transient Data

Click **Transient data** toolbar button , or select **Transient | Data** command. **Transient Data** window will show up. The window always shows data of active document (schematic). Switching to another document automatically updates the data in the window. The window consists of Toolbar, Trace list, and 4 pages used for the following operations:

- **Traces:** add traces, set up individual trace scales and width.
- **Screen:** set up graph scales, gridlines, and other screen options.
- **Table:** configure data table**.**
- **Storage:** manage storage data.



Move cursor over "splitter" area , then press left mouse button and drag to resize panes.

Trace list shows all currently available traces. Checkbox indicates the following trace property depending on selected page:

- **Trace** and **Screen** page – trace is shown on the graph.
- **Table** page – trace is shown in the table.
- **Storage** page – storage is allowed for trace.

One or more traces can be selected in the list using mouse, **Ctrl**, and **Shift** keys. Click **Check all** to check all al traces, **Uncheck all** to uncheck all traces. Most of toolbar commands apply to selected traces only. Please note: **selected** trace is highlighted in the list, and trace selection state is not related to trace checkbox state. On the screenshot above, both traces are "checked", and only V(V1) is selected.

Double-click on the trace to change the color of the trace.

This chapter describes toolbar commands and **Traces** page only. Other pages are described in the **Transient Window** chapter (**Graph**, **Data table**, and **Storage**).

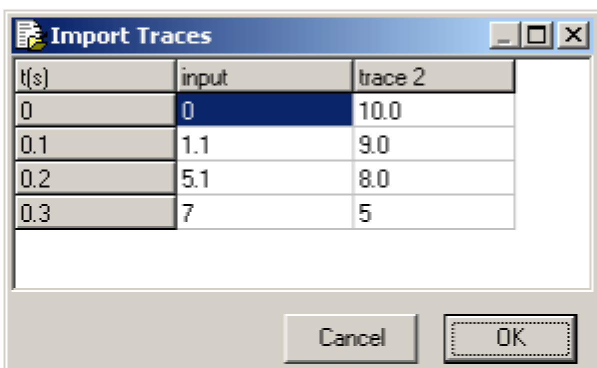
Toolbar

Toolbar button commands apply to all or selected traces. Some of those commands are also accessible through context menus in the **Transient Window**.

- **Open file**. Load traces from "nlt" data file.
- **Import traces** from text or "csv" file. Format of the file is similar to the export format. The first column consists of time (in seconds), other columns consist trace data. The first line is a header line: it may have any text in the first column, and trace names in other columns. If trace name consists of symbols other than numbers and letters, it should be enclosed in quotes. The data and names can be comma, space, or Tab separated. For example:

```
t(s),input,"trace 2"
0,0,10.0
0.1,1.1,9.0
0.2,5.1,8.0
0.3,7.5,7.0
```

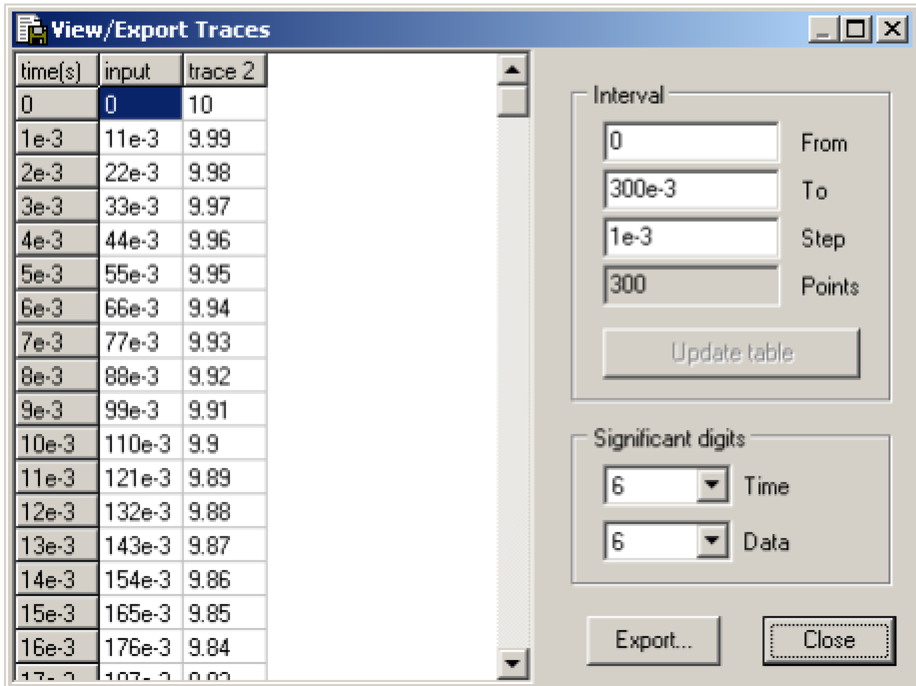
  When file is loaded, its contents will be displayed in the **Import Traces** dialog box, for verifying:



  Click **OK** to confirm import. New traces will be created and shown on the graph.

- **Save selected traces** into "nlt" data file. Only traces selected (highlighted) in the Trace list will be saved into the file.

- **View/Export selected traces.** Only traces selected (highlighted) in the Trace list will be viewed and exported into text or "csv" file. **View/Export** dialog box will show up:



Selected traces are shown as a text in the table. Initially, traces are shown in the time interval between cursors, or, if cursors are disabled, in the full screen. Change **From** and **To** values and press **Enter**, or click **Update table** button to change interval. Traces are shown with fixed time step specified in the **Step** value. Initial step is automatically set so that number of points is close to the value specified by **Approximate number of points** parameter on **Transient** page of **Preferences** dialog box (but not exceeds **Max number of points**, defined on the same page). Number of **significant digits** for time and data columns can be specified.

Click **Export** to save the table into the text file as comma-separated values .

- **Copy selected traces** to the clipboard. Only traces selected (highlighted) in the Trace list will be copied in the clipboard.
- **Paste traces** from the clipboard. Traces from the clipboard will be added to Trace list.
- **Duplicate selected traces.** This operation is equivalent to **Copy**/**Paste** operations. Only traces selected (highlighted) in the Trace list will be duplicated.
- **Remove selected traces.** Only traces selected (highlighted) in the Trace list will be removed.
- **Delete all traces.**
- **Select color** of selected trace. Only one trace should be selected. **Double-click** on the trace in the Trace list performs the same operation.

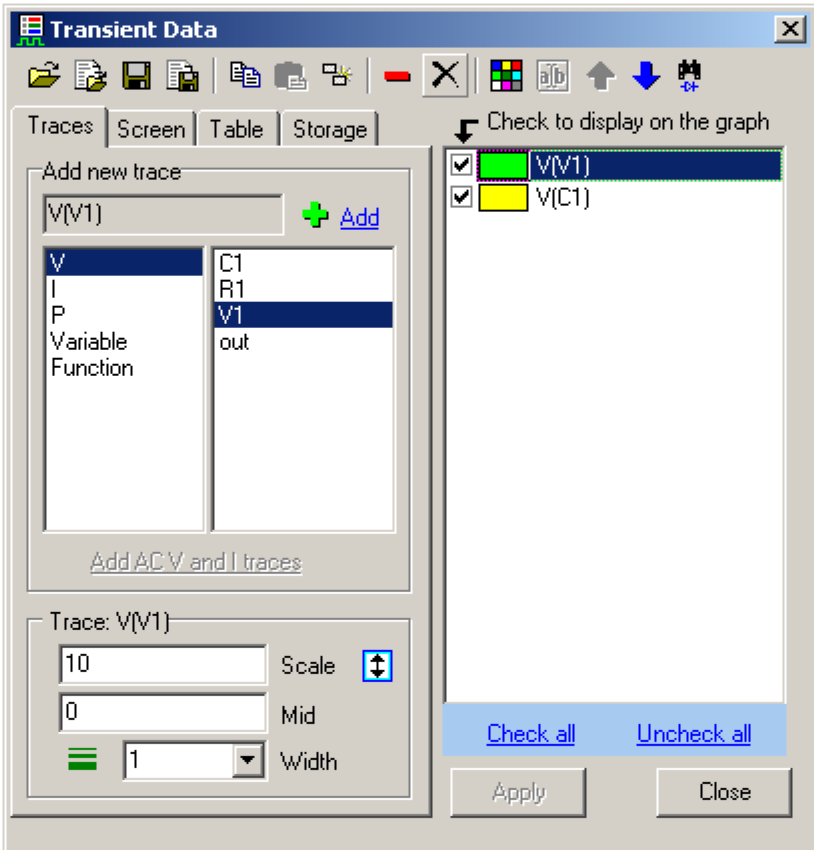
- **Rename trace.** Only one trace should be selected. Only traces loaded from data file, imported from text file, or pasted from clipboard can be renamed. Renaming the trace of **Function** type changes the function. **Rename** dialog box will show up:

Enter new trace name and click **OK**.

- **Move selected traces up.** This operation changes the order of traces in the list, on the graph, and in the data table.
- **Move selected traces down.** This operation changes the order of traces in the list, on the graph, and in the data table.
- **Find component.** If selected trace is V, I, or P of the component, click to show the component on the schematic. The component will be selected (highlighted) and centered on the screen.

Traces

Traces page of **Transient Data** window is used to add and remove traces, and set up individual trace scales and width.



Add new trace. Select trace type in the left list:

- **V** – voltage.
- **I** – current.
- **P** – power.
- **Variable** – schematic variable defined in the **Variables** window.
- **Function** – arbitrary function.

If **V**, **I**, or **P** trace is selected, the right list will show components available for this trace type: the model of the component should support selected type. Select component and click **Add** button , or double-click on the component name to add new trace to the trace list. The name of the trace consists of the letter followed by the name of component in parentheses:

```
V(R1), I(C2), P(L3)
```

If **Variable** trace is selected, the right list will show all variables available in the schematic. Select variable and click **Add** button , or double-click on the variable name to add new trace to the trace list. The name of the trace is the same as variable name.

If **Function** trace is selected, enter function in the edit window and click **Add** button to add new trace to the trace list. The function may consists of arithmetic operators and functions, component parameters, current transient time t, and V, I, and P on the component:

```
sin(t*1000)*(1+cos(t*10))
V(in)/I(A1)
V(X1.V1)
sq(V(r1))/r1
```

The name of the trace is the function itself: so that renaming the trace will change the function.

V, **I**, and **P** traces can also be added from Schematic context menu, and by Components Window toolbar buttons.

The following individual trace parameters can be set:

- **Scale.** Trace value for half of the screen.
- **Mid.** Trace value in the middle of the screen.
- **Width.** Width of the trace line in pixels.
- **Fit the Screen.** Automatically set **Scale** and **Mid** values so that the trace fits current screen.

Select one or more traces in the Trace list, change parameters and press **Enter**, or click **Apply** button.
If selected traces have different values for some parameter, the corresponding field will be left blank. Leave the field blank to keep individual values unchanged, or enter a new value to apply it to all selected traces.

Performing simulation

Use Menu commands, Toolbar buttons, or hotkeys to perform transient simulation.

- **Start transient** (**Transient | Start**, or **F6).** When transient starts, the **Transient Window** opens up, and the time range of the screen is set to the values specified in the **Transient Settings** dialog box: **Start** is left edge of the screen, **Screen** is size of the screen. Although simulation always starts at t=0, the results will be displayed on the graph only from **Start** time. Depending on **Save data before Start** checkbox state, the simulation data prior to **Start** can be ignored, or saved into the memory to be available for display later.

Transient results are simultaneously displayed in the **Transient Window**. Current simulation time is shown in the **Simulation progress** field with green background, if transient is currently running, or yellow background, if it is paused. The amount of memory used for simulation and traces is shown in the **Memory used** field.

- **Pause transient** (**Transient | Pause**, or **Space**). Transient can be paused and then continued at any time. When transient reaches the end of the screen, it pauses automatically, unless **Continuous transient mode** Toolbar button is selected, or **Run until** option is used.

- **Continue transient** (**Transient | Continue**, or **F7**, or **Space**).

Almost all operations in NL5 can be performed when transient is running. You can change graph scales, colors, add and remove traces, change simulation step, component parameters, even edit schematic. The changes will be immediately accepted and affect transient simulation. If required for operation, the transient will be automatically paused, and then restarted immediately upon completion of operation. For some critical operations, however, the transient may stay paused, or even be stopped.

- **Stop transient** (**Transient | Stop**). When transient stopped, it can't be continued and should be started again from the beginning.

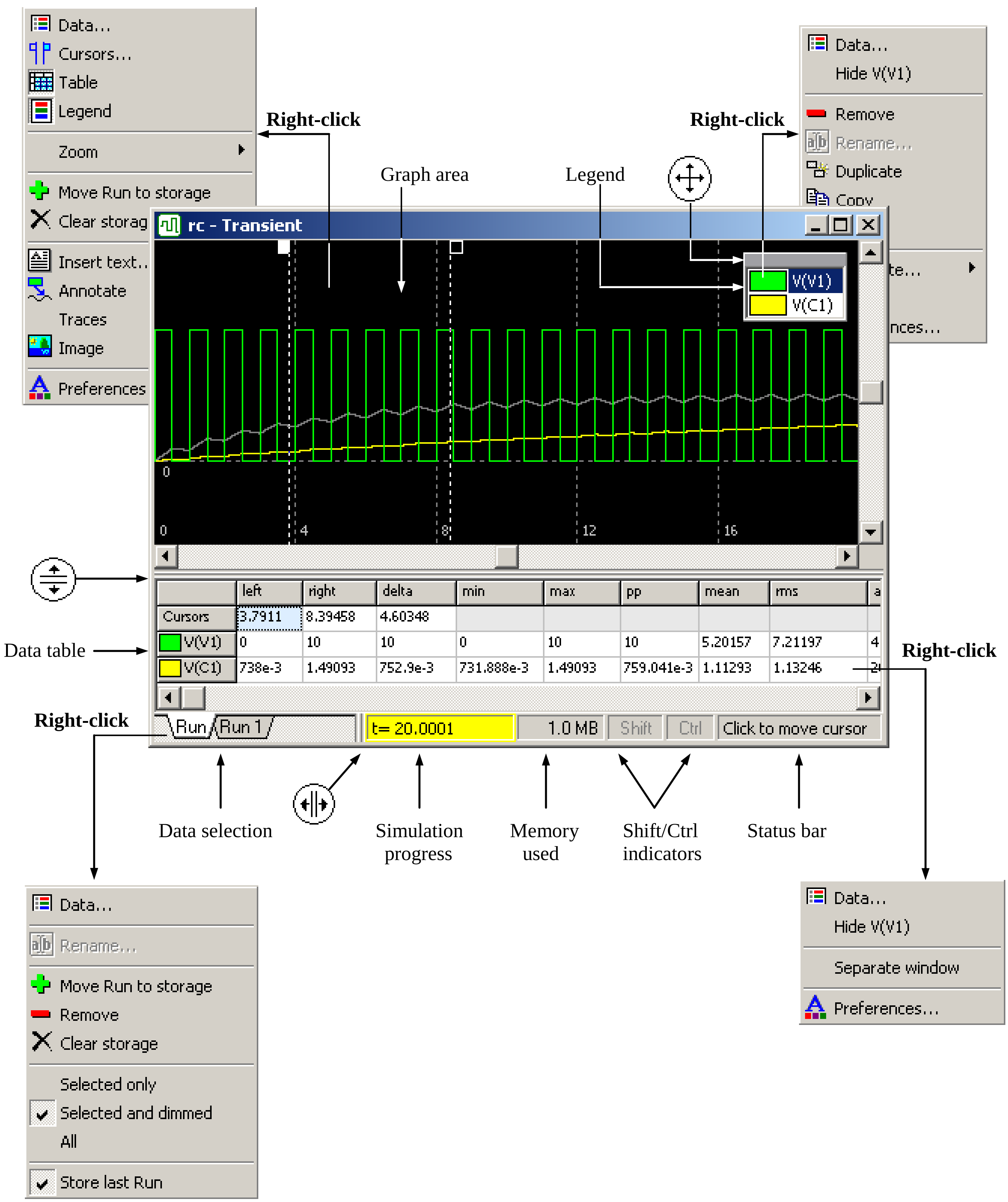
- **Transient Log** (**Transient | Log**). Log information shown in the dialog box may be useful for troubleshooting. The last log is saved into schematic file. When submitting schematic file to Customer Service for help, please save schematic after simulation, in order to have last log included into the file. Click **Copy to clipboard** button to save log text into clipboard.

- **Save IC (Transient | Save IC**) saves current states of all components into their IC (Initial Conditions), if IC parameter exists for a component. This function can be used to store components states when periodic steady-state point is found, so that next simulation can be started already at steady state, without performing long simulation of settling process again. Please note that **Save IC** command does not save phase of periodic sources, so that for accurate results the moment when command is performed should be properly selected.

- **Sweep** (**Transient | Sweep**) allows running series of transient runs while changing component parameter or variable in specified range, and storing transient data in the storage. Sweep is performed using script commands, and is configured on **Sweep** page of **Tools** window. See **Tools**, **Sweep** chapter for details.

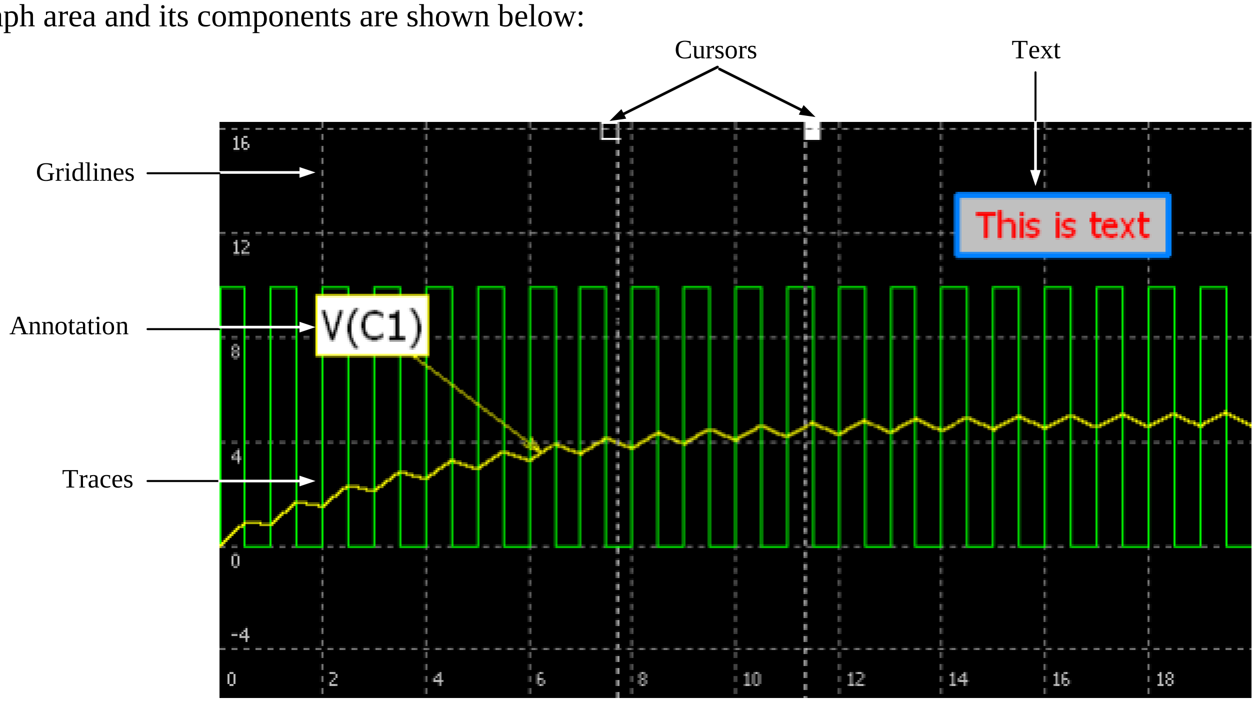
Transient window

Typical view of transient window and its main components are shown below:



- **Graph** area contains traces with annotations, cursors, and text.
- **Legend** window contains list of traces shown on the graph. Click on the gray header bar of the legend window and drag to move legend window.
- **Data table** contains cursors/screen information and calculated traces data.
- **Data selection** area contains last simulation and storage data tabs. Click on a tab to select **Run** or storage data.
- **Simulation progress** shows current simulation time and status (running/paused).
- **Memory used** indicator shows amount of memory used for simulation and traces.
- **Shift/Ctrl indicators** are highlighted when **Shift** and/or **Ctrl** key are depressed.
- **Status bar** shows hint related to current position of mouse pointer and **Shift/Ctrl** state.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize Storage selection area.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize data table.
- **Right-click** on the graph, legend, Data table, or Data selection area to see context menu with relevant commands.
- Common properties of **Transient Window**, such as colors, fonts, and some options, can be customized on **Graphs, Table, Annotation,** and **Text** pages of **Preferences** dialog box. Properties specific to the document (schematic) can also be set up in the **Transient Data** window.

Graph area and its components are shown below:



Graph

Graph navigation can be performed by commands available in the transient context menus, transient toolbar buttons, shortcuts, keyboard keys, and mouse. Very often, the same operation can be performed by different ways. For instance, zooming graph in/out can be done using keyboard keys only, mouse only, or both. It is user's choice to select the most effective and convenient one.
There are 3 graph operation modes:

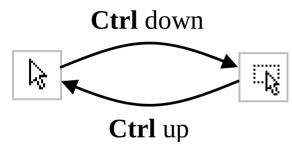
- ***Cursors*****.** Moving cursors.

- ***Zoom***. Zooming graph using mouse.

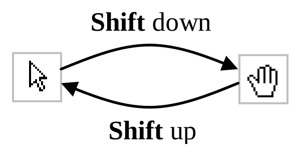
- ***Scrolling***. Scrolling the graph.

The mode can be selected by clicking button on the transient toolbar. Also, there are quick ways to switch temporary from ***Cursors*** mode to ***Zoom*** and ***Scrolling*** modes:

- Press and hold **Ctrl**, click and drag mouse to zoom the graph.
  Release **Ctrl** to return to ***Cursors*** mode:



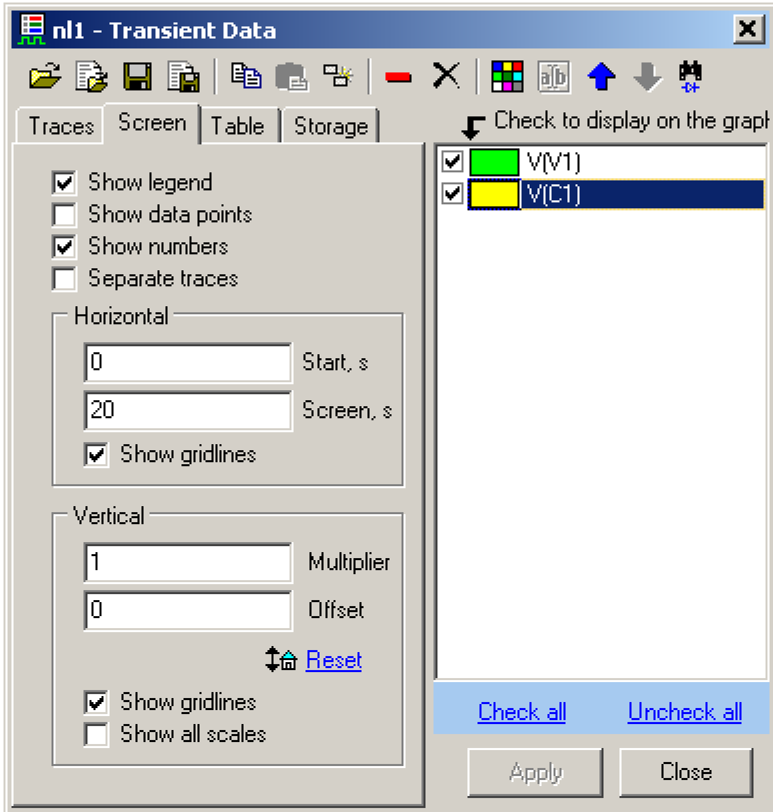
- Press and hold **Shift**, click and drag mouse to scroll the graph.
  Release **Shift** to return to ***Cursors*** mode:

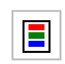
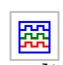


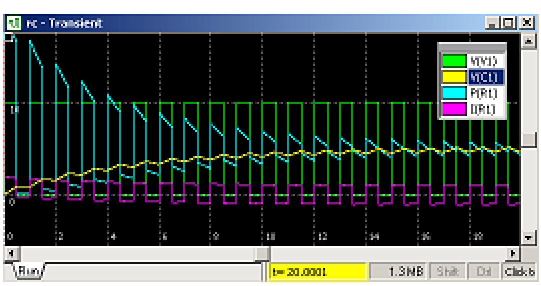
Traces are shown on the graph with their individual scales, width, and colors, defined on **Traces** page of **Transient Data** window. When graph zooming is performed, it does not change scales of individual traces. Instead, it changes screen **Multiplier** and **Offset** parameters, which are applied to all traces. Scale of selected trace is shown on the graph. If trace selection changes, scale numbers and gridlines position may change too.

Gridlines spacing is selected automatically so that last significant digit step is 1, 2, or 5, and distance between gridlines is approximately equal to the value specified on **Graphs** page of **Preferences** dialog box as **Gridlines interval** in pixels.

Scales, gridlines, and some other options of the graph can be changed on **Screen** page of **Transient Data** window:



- **Show legend**. Select to show **Legend** window. Also use **Legend** button  on the Toolbar or context menu command.
- **Show data points.** Select to mark calculated data points of all traces as small squares. This option may be useful for troubleshooting and calculation step selection.
- **Show numbers.** Select to show scale numbers.
- **Separate traces.** Also use **Separate traces** toolbar button  or press **Tab** in the **Transient Window**. Traces will be separated vertically, which helps to distinguish similar traces. Horizontal gridlines are used to divide traces, and vertical scales are not shown. When zooming graph by mouse, only horizontal zoom will work for separated traces.



Normal mode

Separated traces

Horizontal. Set up horizontal scale and gridlines.

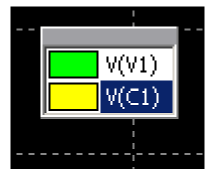
- **Start**. Time at the left edge of the screen.
- **Screen.** Time per screen.
- **Show gridlines.** Select to show gridlines.

Vertical. Set up vertical scale and gridlines.

- **Multiplier**. Screen scale multiplier: applied to all traces.
- **Offset.** Screen offset: applied to all traces.
- **Reset.** Reset vertical multiplier to 1 and offset to 0.
- **Show gridlines.** Select to show gridlines.
- **Show all scales.** Show scales for all traces in trace colors.

Legend

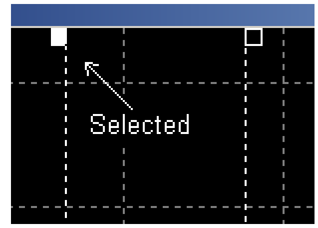
Legend window contains list of traces shown on the graph.



- To show/hide legend click **Legend** button on the Toolbar or in the context menu, or use **Show legend** checkbox in the **Screen** page of **Transient Data** window.
- **Click** on the trace to select the trace. Selected trace will be shown on top of all traces.
- **Double-click** on the trace to select trace and to open **Transient Data** window.
- **Right-click** to select trace and open context menu. The menu will contain some common commands, and commands related to selected trace.
- **Click** on the gray header bar of the legend and drag to move the window.
- Legend font size and window width limit can be selected on the **Legend** page of **Preferences** dialog box.

.

Cursors

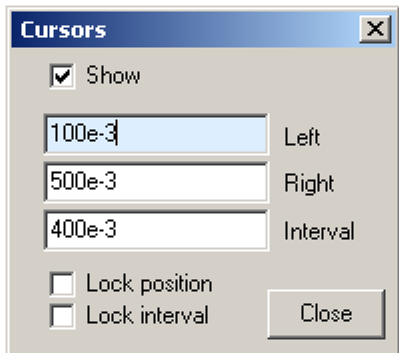


Cursors are used mostly for selecting time interval on the graph for Data table calculations. Selected (active) cursor is shown with solid colored square on the top. To show/hide (enable/disable) cursors click **Show/hide cursors** Toolbar button .

Select ***Cursors*** mode ( ) to move cursors on the graph.

- **Double-click** on the graph to set both cursors to the same point. This will also enable cursors, if they were disabled.
- **Click** on the graph to move nearest cursor to this point.
- **Click and drag** to select and move cursor.

To place cursors to specific positions and for other options **right-click** on the graph and select **Cursors** command from context menu. **Cursors** dialog box will show up:



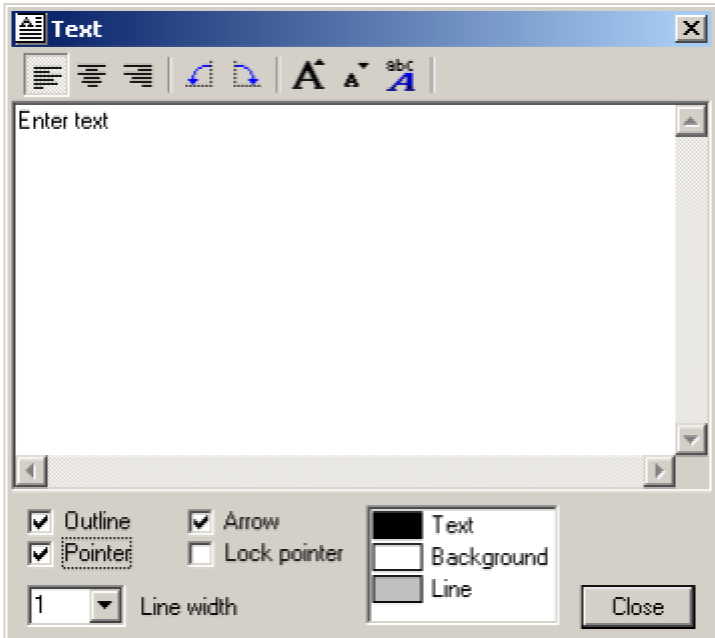
- **Show**. Select checkbox to show (enable) cursors
- **Left, Right, Interval**. Enter new cursors position or interval; press **Enter** to apply or **Esc** to cancel. If interval changed, an active cursor will move.
- **Lock position**. Lock cursors at current position, so that cursors can not be moved.
- **Lock interval**. Keep current interval between cursors. If one cursor is being moved, another one will automatically follow it to maintain specified interval.

The following toolbar buttons can be used to move cursors:

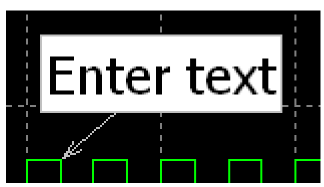
- **Right maximum.** Move selected cursor to the nearest right maximum of selected trace.
- **Left maximum.** Move selected cursor to the nearest left maximum of selected trace.
- **Right minimum.** Move selected cursor to the nearest right minimum of selected trace.
- **Left minimum.** Move selected cursor to the nearest left minimum of selected trace.
- **Maximums.** Move one cursor to the nearest right maximum, and another cursor to the nearest left maximum of selected trace.
- **Minimums.** Move one cursor to the nearest right minimum, and another cursor to the nearest left minimum of selected trace.

Text.

To add text on the graph **right-click** on the graph and select **Insert Text** command from the context menu. **Text** dialog box will show up:



Enter text in the text box. The text will be simultaneously shown on the graph:



The text can be formatted using toolbar buttons and controls:

Alignment. Set alignment of multi-line text.

- **Align left.**
- **Center.**
- **Align right**.

Orientation. Change orientation of the text.

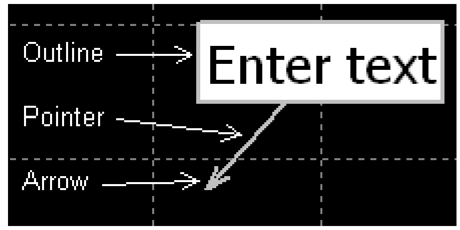
- **Rotate left.**
- **Rotate right.**

Font. Change size of the font or select specific font type and options.

- **Larger font.**

- **Smaller font.**
- **Select font.**

Outline and pointer options



- **Outline.** Draw outline rectangle.
- **Pointer.** Draw pointer line from the text to specified point.
- **Arrow.** Draw pointer line with arrow.
- **Lock pointer.** Lock the end of the pointer: the end of the pointer will not move even when text is being moved.

- **Line width.** Specify line width of the outline and pointer.
- **Color.** Double-click on the item in the list to change the color.

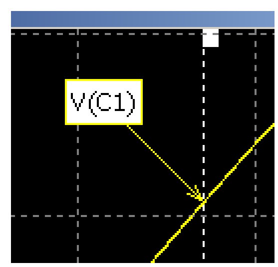
If graph is zoomed or scrolled, the text stays at the same place, anchored to left-top corner of the graph window. To move the text, click on the text and drag. If pointer is locked, only text will move. To move the pointer only, click on the end of the pointer and drag.

To edit the text, **double-click** on the text, or **right-click** on the text and select **Edit text** command from context menu. The same **Text** dialog box will show up.

To delete the text, **right-click** on the text and select **Delete text** command from context menu.

Annotation

Annotation is a text with a pointer, which always points to the same data point of a trace, even when graph is zoomed or scrolled. Annotation belongs to a trace, so if trace is deleted, all trace annotations are deleted as well. Annotation is also deleted if trace data is cleared. For instance, if annotation is added to simulation (**Run**) trace, and a new simulation is started, the annotation will disappear, since the trace data is cleared at simulation start.



To add annotation, set active cursor to the time point where annotation is needed, right-click on the graph, select **Annotate**, then select **Selected trace** or **All traces** command. The same buttons are available in the Transient toolbar. Annotation(s) will be added only if trace data exists at cursor's time. If cursors are disabled, annotation will be added approximately at 1/3 of a screen.

Annotation font, colors, number of significant digits, and some other properties can be specified on **Annotation** page of **Properties**. To change annotation text and annotation-specific properties, **double-click** on the annotation, or **right-click** on the annotation and select **Edit annotation** command from context menu. **Annotate** dialog box will show up.

Enter text in the text box. The text will be simultaneously shown on the annotation. The following options and formatting are available:

- **Name.** Display trace name in the text.
- **Time.** Display time of the annotation in the text.
- **Value.** Display trace value (amplitude) in the text.

Alignment. Set alignment of multi-line text.

- **Align left.**
- **Center.**
- **Align right**.

Orientation. Change orientation of the text.

- **Rotate left.**
- **Rotate right.**

- **Apply to all annotations.** Select to apply current settings to all annotations on the graph.

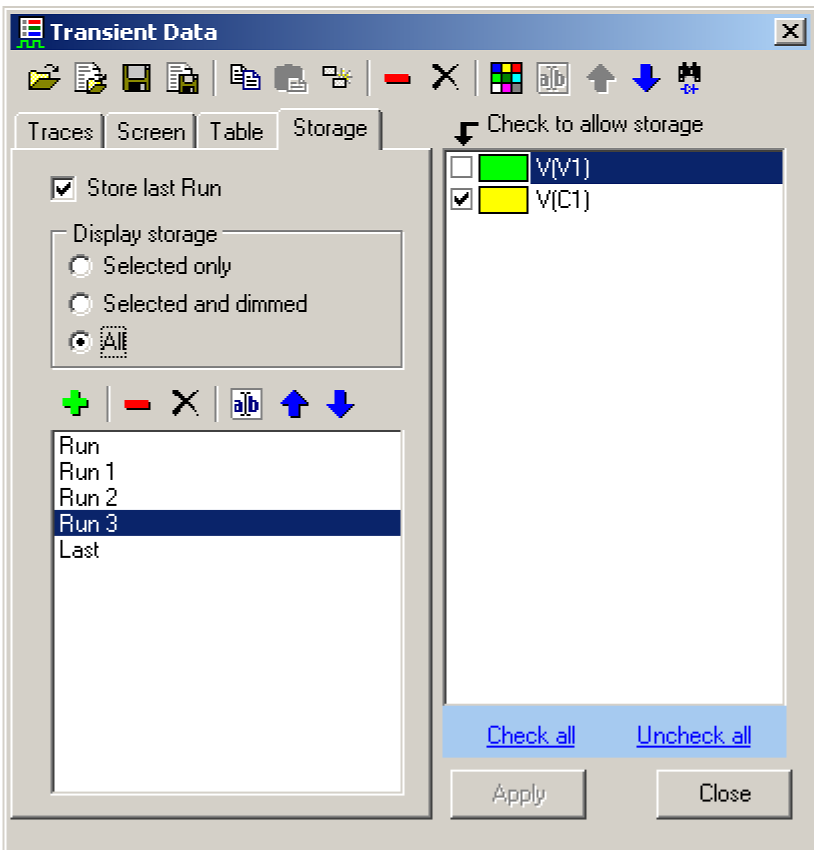
To move annotation text keeping pointer at the same trace point, click on the annotation text and drag. To move the pointer, click on the end of the pointer and drag. The pointer will change time, yet following trace amplitude. Annotation text will move with the pointer.

To delete annotation, **right-click** on the annotation and select **Delete annotation** command from context menu. To delete all annotations, **right-click** on the graph, select **Annotate**, then select **Delete all** command .

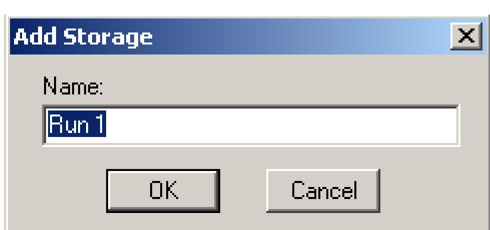
Storage

The results of the last simulation run are always shown in the **Run** tab of the **Transient Window**. Last run data can be moved into storage under the name **Last**, so that it can be compared with other simulation runs. Each storage data has a tab on the **Data selection** area assigned to it. Storage data can be selected by clicking on the tab. Storage data belongs to the **trace**, so that if trace is deleted, storage data for this trace will be deleted as well.

To access storage-related commands **right-click** on the graph or Data selection area, then select command from context menu. List of available storage data, storage-related commands, and storage display selection can be found on **Storage** page of **Transient Data** window:

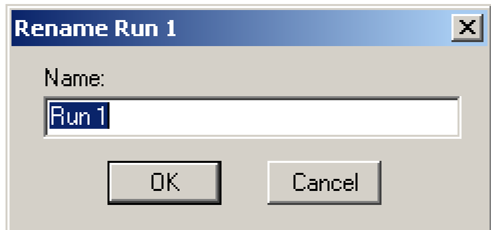


- **Move Run to storage.** Move current simulation data into storage. **Add Storage** dialog box will show up:



Enter new storage name or leave suggested default name and click **OK**. A new tab with storage name will be created in the **Data selection** area of **Transient Window**

- **Remove** selected storage. Last **Run** data can be removed as well.
- **Clear storage.** Delete all storage data.
- **Rename** selected run. **Rename** dialog box will show up:



Enter new storage name and click **OK**. Names "Run" and "Last" are reserved and not allowed.

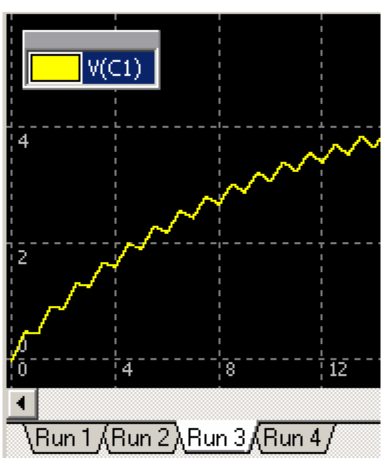
- **Move selected up.**
- **Move selected down.**

- **Store last Run.** Select this option to compare new simulation with previous one. When new simulation starts, current simulation data will be moved into storage under the name **Last**. If run with the name **Last** already exists, it will be overwritten. Thus, previous simulation data will always be in the storage under the name **Last**.

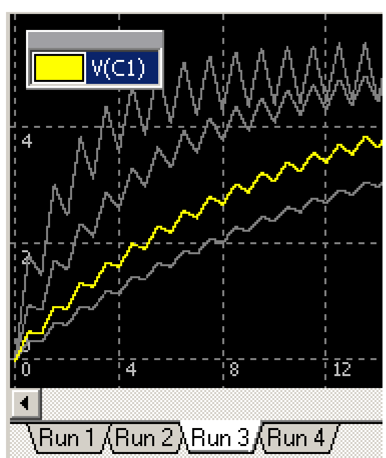
Display storage

- **Selected only.** Only selected data is shown on the graph.
- **Selected and dimmed.** Selected data is displayed with normal trace colors, other data is displayed with dimmed color, specified at **Graphs** page of **Preferences**.
- **All.** All data is displayed with normal trace colors.

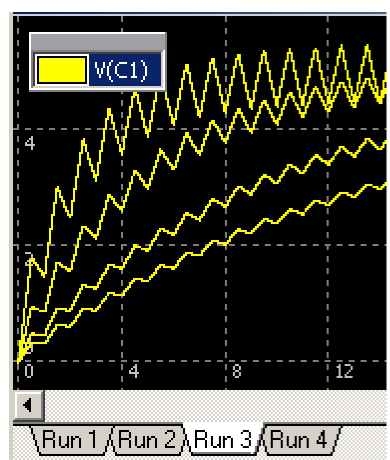
Example:



Selected only



Selected and dimmed

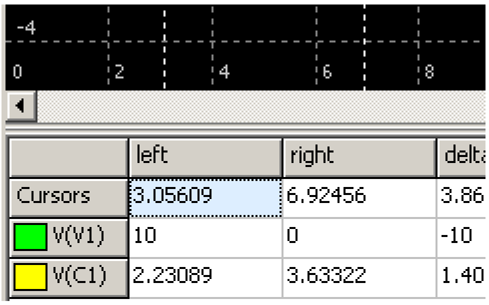


All

When **Storage** page is selected, checkboxes in the trace list specify traces with storage allowed.

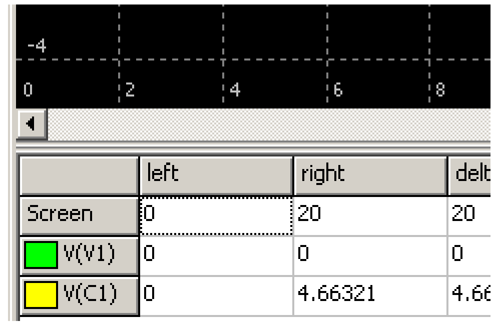
Data table

The Data table shows cursors position, trace values, and some characteristics of the traces calculated between cursors, such as: mean, max, min, rms values, and more. If cursors are disabled, the table shows the data at the left and right edges of the screen, and values calculated between left and right edges of the screen:



	left	right	delt
Cursors	3.05609	6.92456	3.86
V(V1)	10	0	-10
V(C1)	2.23089	3.63322	1.40

Cursors enabled, active cursor is highlighted



	left	right	delt
Screen	0	20	20
V(V1)	0	0	0
V(C1)	0	4.66321	4.66

Cursors disabled, screen size = 20

- To show/hide Data table click **Table** toolbar button , or **right-click** on the graph and select **Table** command from context menu.
- **Click** on the trace row to select the trace. Selected trace will be shown on top of all traces.
- **Double-click** on the table to open **Transient Data** window.
- **Right-click** to open context menu. The menu will contain some common commands, and commands related to selected trace.
- Colors, fonts, and number of significant digits used in the table can be customized on **Table** page of **Preferences** dialog box.

The table can be displayed on the bottom of the **Transient Window**, or as a separate window: **right-click** on the table and select **Separate window** command:

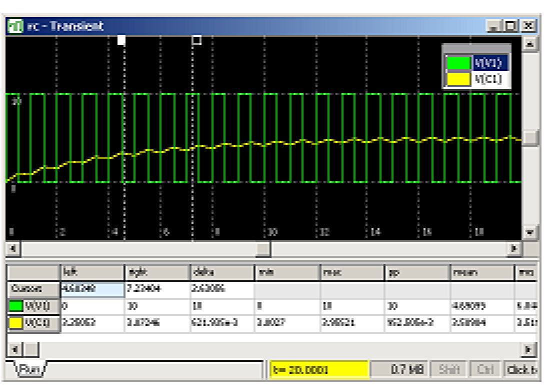


Table in the Transient Window

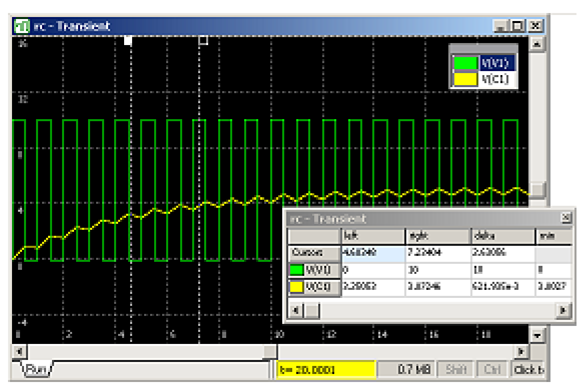
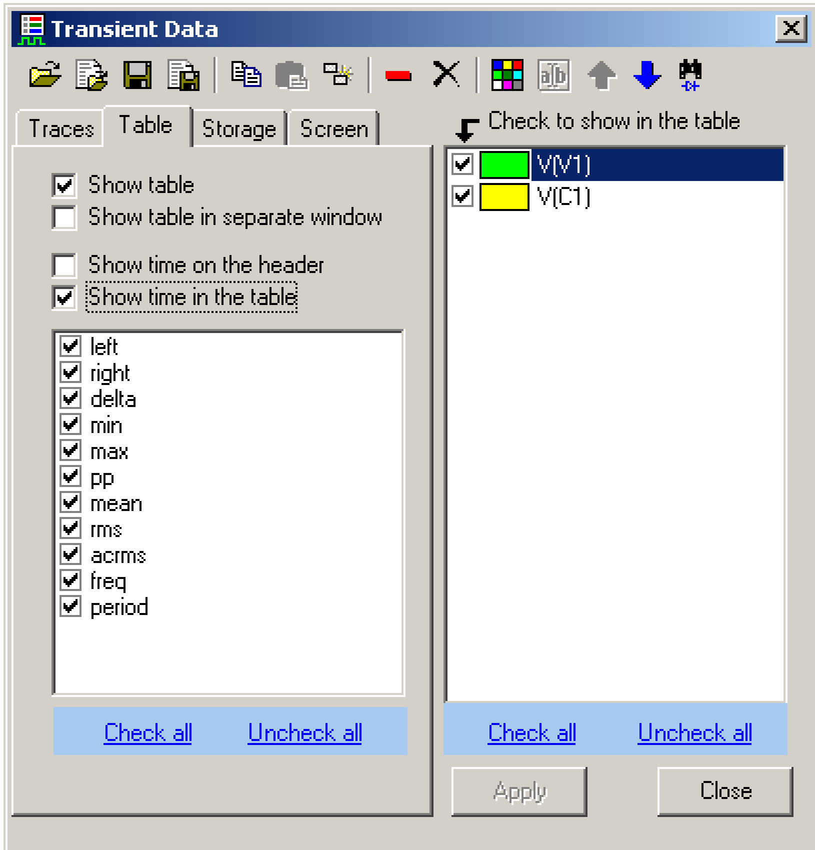


Table in separate window

The values shown in the table, as well as other table options, can be selected on **Table** page of **Transient Data** window:



- **Show table.** Select to show table.
- **Show table in separate window.** If selected, the table will be displayed as a separate window.
- **Show time on the header.** If selected, cursors position will be shown in the header line, in the **left**, **right**, and **delta** columns.

Cursors	3.67505	6.46035	2.7853
V(V1)	0	10	10
V(C1)	2.79145	3.90708	1.11563

- **Show time in the table.** If selected, cursors positions will be shown in separate row.

	left	right	delta
Cursors	3.67505	6.46035	2.7853
V(V1)	0	10	10
V(C1)	2.79145	3.90708	1.11563

- **Table values.** Select values to display in the table:
    - **left** – trace value at left cursor.
    - **right** – trace value at right cursor.
    - **delta – right** minus **left.**
    - **min –** trace minimum between cursors.

- **max** – trace maximum between cursors.
- **pp** – trace peak-to-peak value between cursors.
- **mean** – trace averaged value between cursors.
- **rms** – trace RMS (root-mean-square) value between cursors.
- **acrms** – trace AC RMS value between cursors**:** RMS calculated on the trace with subtracted mean value.
- **freq** – calculated frequency of the signal between cursors. Frequency is calculated based on number and intervals between points where the trace is crossing its **mean** level.
- **period – 1 / freq**

When **Table** page is selected, checkboxes in the trace list specify traces shown in the table.

Scrolling and Zooming

To scroll graph use any of the following methods:

- Move mouse pointer to the left or right edge of the graph. Mouse pointer will take "big arrow" shape. Click or hold left mouse button to scroll graph.
- ***Cursors*** mode : hold **Shift** key, then click and drag graph.
- ***Scrolling*** mode : click and drag graph.
- Hold **Ctrl** key and rotate **mouse wheel** to scroll horizontally.
- Hold **Shift** key and rotate **mouse wheel** to scroll vertically.
- Press **Right** and **Left** keys.
- Press **End** to center beginning of the traces (set to the middle of the screen).
- Press **Ctrl-End** to center end of the traces.
- Press **Shift-End** to center middle of the traces.
- ***Zoom*** mode : **double-click** on the graph to center this point.

To zoom graph use any of the following methods:

- Rotate **mouse wheel** to zoom horizontally.
- Hold **Ctrl** and **Shift** key and rotate **mouse wheel** to zoom vertically.
- Click toolbar buttons, or use keyboard shortcuts, or **right-click** on the graph, select **Zoom**, then select command:

Horizontal Zoom-in (Ctrl-PgUp).

Horizontal Zoom-out (Ctrl-PgDn).

Fit the screen horizontal (Ctrl-Home).

Fit cursors to screen.

Vertical Zoom-in (PgUp).

Vertical Zoom-out (PgDn).

Fit the screen vertical (Home).

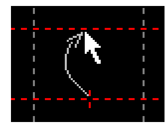
- **Fit the screen (Shift-Home).**
- **Reset vertical scale** (set Multiplier=1, Offset=0).
- **Zoom-in (Shift-PgUp).**
- **Zoom-out (Shift-PgDn).**

To zoom selected area

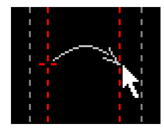
- ***Zoom*** mode : click and drag to select area.
- ***Cursors*** mode : hold **Ctrl** key, then click and drag to select area.

Selection area depends on how mouse pointer is moving relative to starting point.

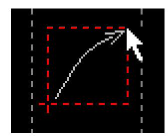
- If mouse pointer is moving only up or down, two horizontal lines will be shown. When left button released, selected area will be zoomed-in vertically.



- If mouse pointer is moving only left or right, two vertical lines will be shown. When left button released, selected area will be zoomed-in horizontally.



- If mouse pointer is moving diagonally, rectangle will be shown. When left button released, selected rectangle area will be zoomed-in to fit the screen.



To Undo and Redo scrolling and zooming click toolbar buttons:

- **Undo** scrolling or zooming.
- **Redo** scrolling or zooming.

Transient commands

The following commands, buttons, and shortcuts are available in the Main menu, Main Toolbar, transient toolbar, and transient context menus.

- **Open/Show transient window (F5)**.
- **Transient Settings.** Open Transient Settings dialog box.
- **Transient Data.** Show Transient Data window.
- **Continuous transient mode.**
- **Start (F6).** Start transient.
- **Pause (Space).** Pause transient.
- **Continue (F7, Space).** Continue transient.
- **Stop.** Stop transient
- **Log.** Show transient log.
- **Sweep**.
- **Save IC**.
- **Preferences.** Open **Preferences** dialog box.

Toolbar and some context menus:

- **Cursors** mode.
- **Zoom** mode.
- **Scrolling** mode.
- **Horizontal Zoom-in (Ctrl-PgUp).**
- **Horizontal Zoom-out (Ctrl-PgDn).**
- **Fit the screen horizontal (Ctrl-Home).**
- **Fit cursors to screen.**
- **Vertical Zoom-in (PgUp).**
- **Vertical Zoom-out (PgDn).**
- **Fit the screen vertical (Home).**
- **Fit the screen (Shift-Home).**
- **Reset vertical scale** (set Multiplier=1, Offset=0).
- **Undo scale (Backspace).** Undo scale.
- **Redo scale.**
- Show/hide **Cursors**.
- Show/hide **Data Table**.
- Show/hide **Legend**.
- **Separate traces.**

- **Right maximum.** Move selected cursor to the nearest right maximum of selected trace.
- **Left maximum.** Move selected cursor to the nearest left maximum of selected trace.
- **Right minimum.** Move selected cursor to the nearest right minimum of selected trace.
- **Left minimum.** Move selected cursor to the nearest left minimum of selected trace.
- **Maximums.** Move one cursor to the nearest right maximum, and another cursor to the nearest left maximum of selected trace.
- **Minimums.** Move one cursor to the nearest right minimum, and another cursor to the nearest left minimum of selected trace.

Graph commands (context menu):

- Open **Cursors** dialog box.
- **Traces ▶** (Commands apply to all traces displayed on the graph)
  - **Open.** Load traces from "nlt" data file
  - **Import** traces from text or "csv" file.
  - **Save** traces into "nlt" data file.
  - **View/Export.** View traces as a text and save into text or "csv" file.
  - **Copy** traces to clipboard.
  - **Paste** traces from clipboard.
- **Image ▶**
  - **Copy to clipboard.** Copy image of transient window to clipboard.
  - BMP **Save as BMP.** Save image of transient window to file in BMP format.
  - JPG **Save as JPG.** Save image of transient window to file in JPG format.

Storage commands:

- **Move Run to storage**.
- **Remove** selected storage.
- **Clear storage.**
- **Rename** selected storage.
- **Store last Run.** Move current Run into storage "Last" when new simulation starts.

- **Selected only**
- **Selected and dimmed**
- **All**

} Storage display mode

Annotation commands:

- **Annotate selected trace.**

- **Annotate all traces.**
- **Edit annotation.**
- **Delete annotation.**
- **Delete all.**

Text commands:

- **Insert text** on the graph.
- **Edit text.**
- **Delete text.**

Data table commands:

- **Hide *trace name*:** do not show trace in the Data table.
- **Separate window.** Show Data table in the Transient Window or as a separate window.

Legend commands:

- **Hide *trace name*:** do not show trace on the graph.
- **Remove** selected trace.
- **Rename** selected trace.
- **Duplicate** selected trace.
- **Copy** selected trace to clipboard.
- **Paste** traces from clipboard.
- **Find component:** V, I, and P traces only. Click to show the component on the schematic.

Keyboard keys and shortcuts

The following keyboard keys and shortcuts can also be used:

- **Space.** Pause or Continue transient.
- **Tab.** Separate traces.
- **Left, Right.** Scroll graph.
- **Up, Down.** Select trace.
- **End.** Center beginning of the traces (set to the middle of the screen).
- **Ctrl-End.** Center end of the traces.
- **Shift-End.** Center middle of the traces.
- **Shift-PgUp.** Zoom-in.

- **Shift-PgDn.** Zoom-out.

Mouse operation

The following mouse operation can be used.

- **Right-click.** Open context menu.
- **Mouse-wheel.** Horizontal zoom-in/zoom-out.
- **Ctrl-mouse wheel**. Scroll horizontally.
- **Shift-mouse wheel**. Scroll vertically.
- **Ctrl-Shift-mouse-wheel.** Vertical zoom-in/zoom-out.

Cursors mode :

- **Click (left button).** If cursors visible, move nearest cursor.
- **Click and drag.**
  - On annotation: move annotation text or pointer.
  - On text: move text or pointer.
  - Otherwise: move cursor.
- **Double-click.**
  - On annotation: edit annotation.
  - On text: edit text.
  - Otherwise: show cursors, move both cursors.

Zoom mode :

- **Click and drag.** Select and zoom.
- **Double-click.** Center screen.

Scrolling mode :

- **Click and drag.** Scroll graph.
- **Double-click.** Center screen.

Transient Tools

Transient Tools offer different ways of presenting simulation results. To open Tool go to **Transient | Tools**, and then select the line with required Tool.

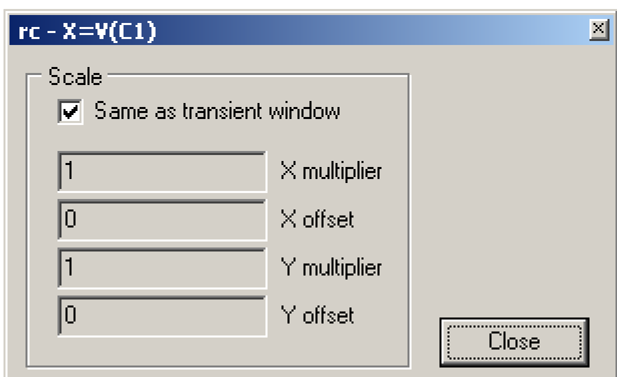
The following Tools are currently available:

- XY diagram
- Histogram
- FFT
- Eye diagram
- Markers
- Power

XY diagram

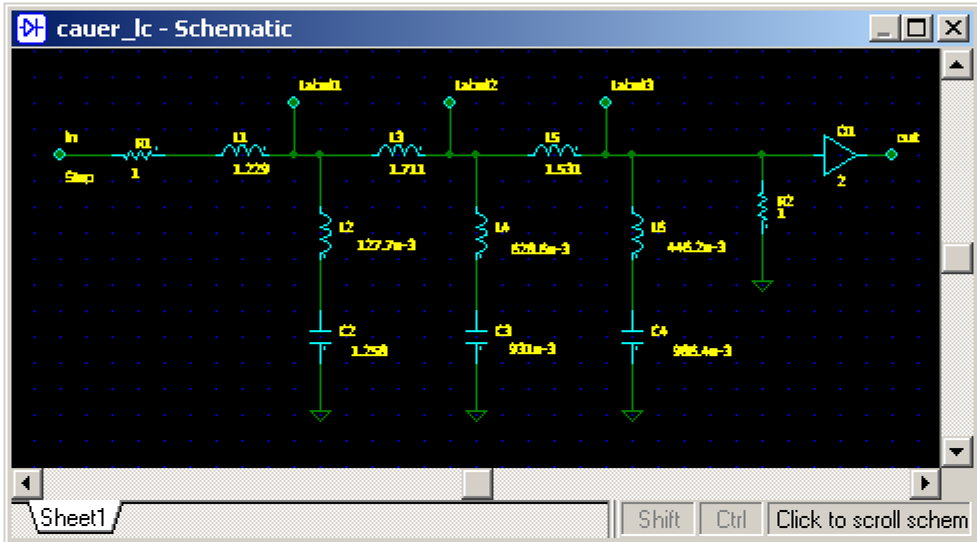
XY diagram shows all traces as a function of selected trace. Selected trace name (X axis) is shown in the header line. The diagram shows traces between cursors only (or on the screen, if cursors are disabled).

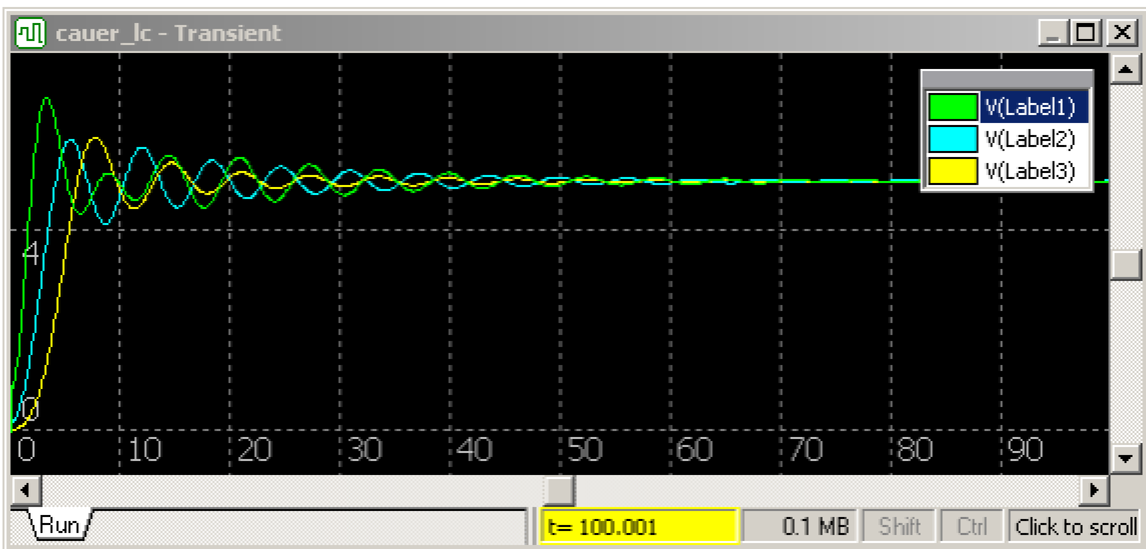
- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:

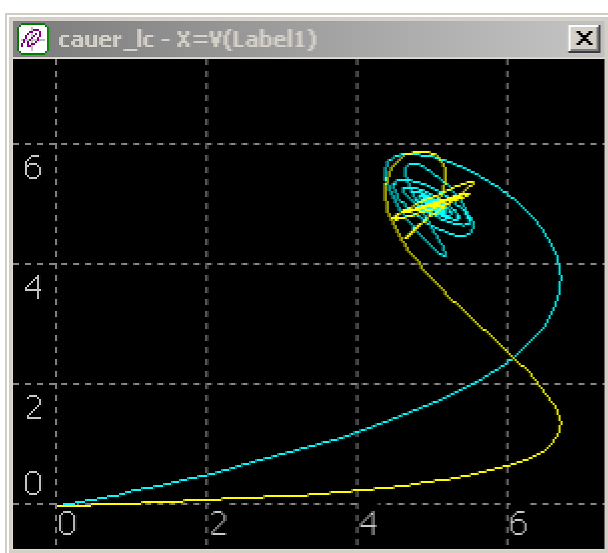


By default, diagram X and Y axes scales are the same as transient screen vertical scale. Uncheck **Same as transient window** checkbox and enter individual X and Y axes multipliers and offsets.

Example: schematic, transient, and XY diagram.



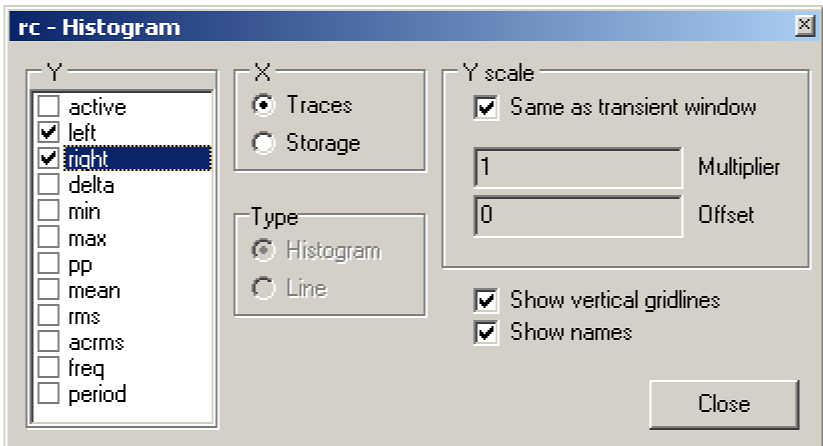




Histogram

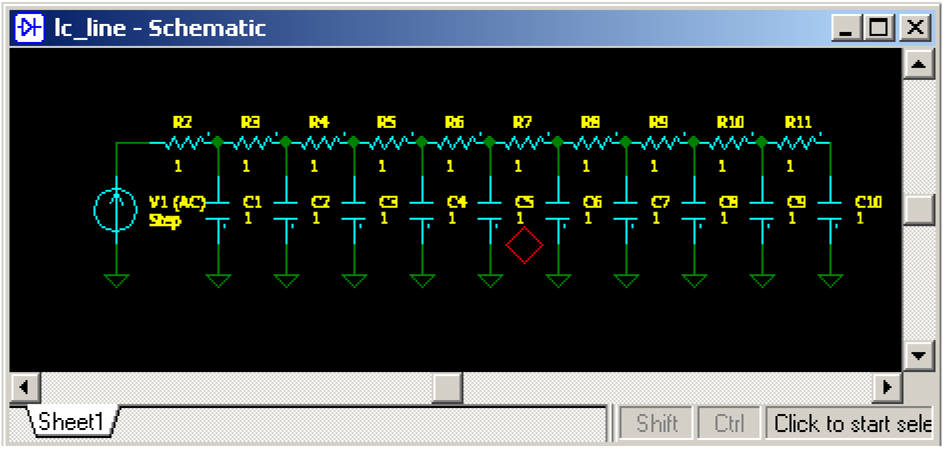
Histogram presents trace values and some characteristics of the traces calculated between cursors (or on the screen, if cursors are disabled), in a graphical format. Histogram can also show "cross-section" of traces or storage data.

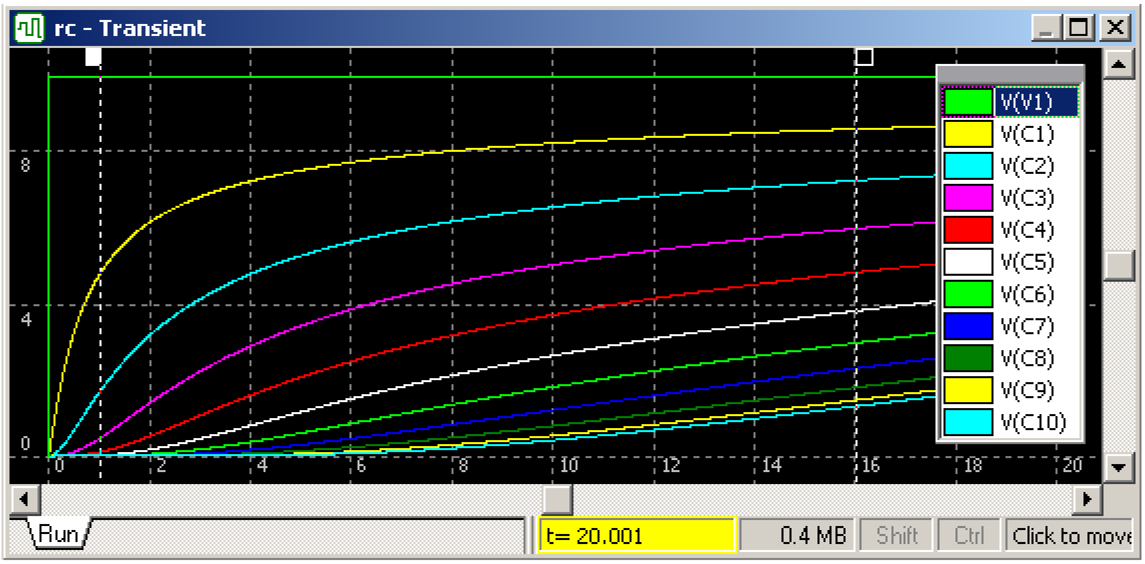
- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:

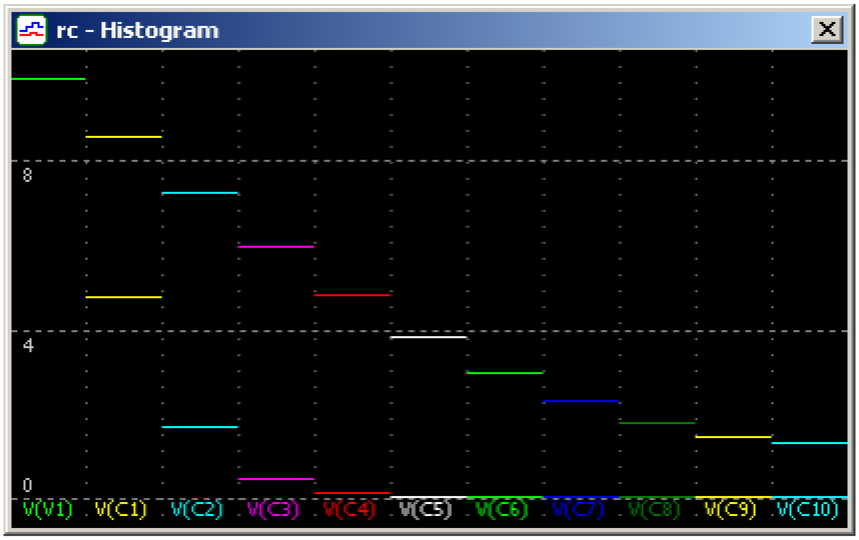


- **Y** values. Select variables to be displayed on Y axis. "**Active"** is currently selected cursor (left or right). Other values are similar to what is displayed in the transient Data table.
- **X**. Select histogram mode: what is shown on X axis.
  - **Traces**. Show "cross-section" of all traces currently displayed on the graph.
  - **Storage**. Show "cross-section" of the storage for all traces currently displayed on the graph.
- **Type**. Select histogram type for **Storage** mode:
  - **Histogram.**
  - **Line**.
- **Y scale**. By default, histogram Y scale is the same as transient screen vertical scale. Uncheck **Same as transient window** checkbox to enter Y scale **Multiplier** and **Offset**
  - **Show vertical gridlines.** Check to show vertical gridlines dividing histogram data.
  - **Show names.** Check to show trace or storage names on the X axis.

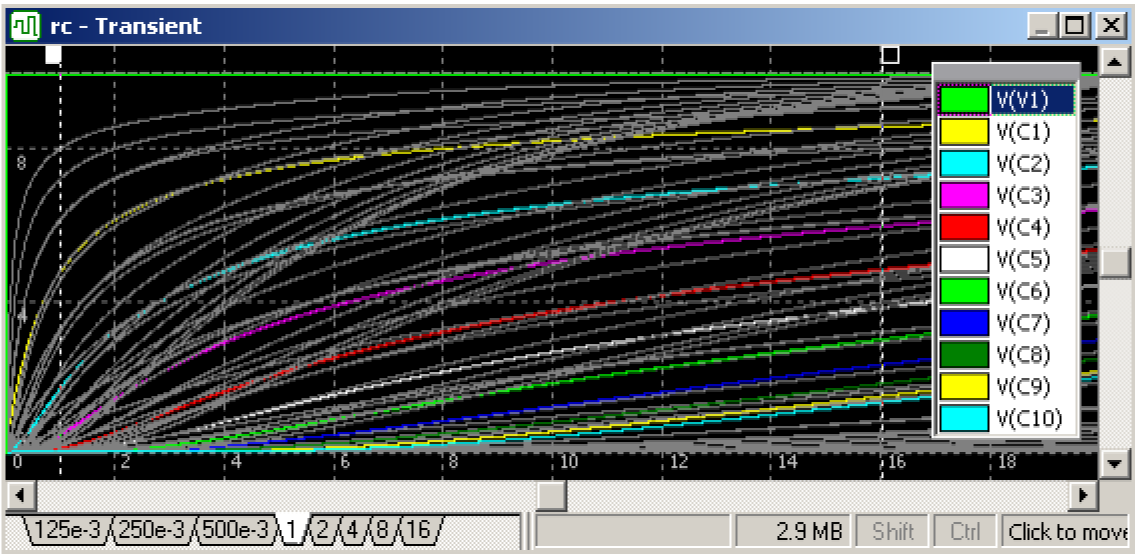
Traces mode, or "traces cross-section", can be used to display "spatial" distribution of the signal in the schematic. The following example shows modeling of heat conduction through the rod using electrical analogy. RC chain models 1-dimensional rod with temperature source (V1) applied to one end. Traces show temperature at certain distance from the end. When temperature changes as a step, temperature front propagates through the rod. Histogram shows temperature distribution along the rod at t=1 (left cursor), and t=16 (right cursor).

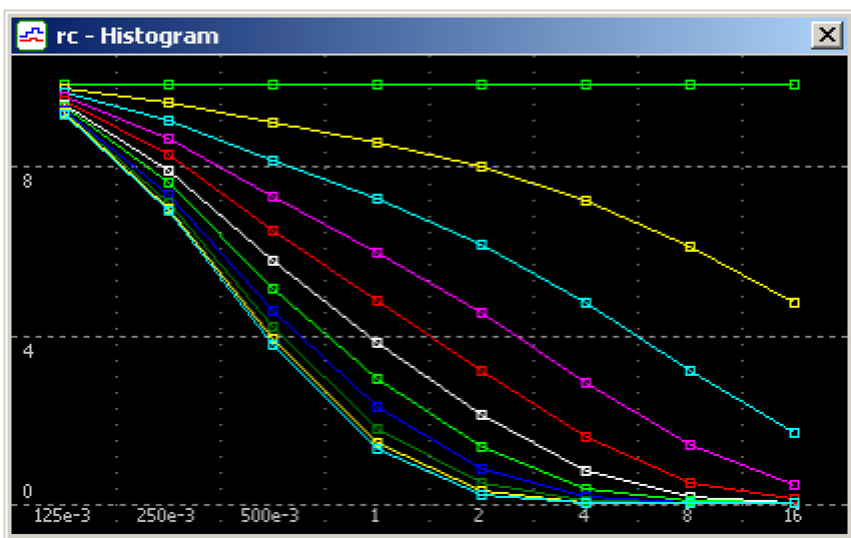

lc_line - Schematic
R2 R3 R4 R5 R6 R7 R8 R9 R10 R11
V1 (AC) Step
C1 C2 C3 C4 C5 C6 C7 C8 C9 C10
Sheet1
Shift
Ctrl
Click to start sele


rc - Transient
V(V1)
V(C1)
V(C2)
V(C3)
V(C4)
V(C5)
V(C6)
V(C7)
V(C8)
V(C9)
V(C10)
0 4 8
0 2 4 6 8 10 12 14 16 18 20
Run
t= 20.001
0.4 MB
Shift
Ctrl
Click to move


rc - Histogram
0 4 8
V(V1) V(C1) V(C2) V(C3) V(C4) V(C5) V(C6) V(C7) V(C8) V(C9) V(C10)

Storage mode, or "storage cross-section", can be used to display how trace values at specific time depend on schematic parameters. The following example shows modeling of previous schematic, with resistance (reciprocal of heat conductance) changing from 0.125 to 16, with X2 step using transient sweep. Each run is saved into storage. X axis of histogram is storage data (i.e. resistance). The lines of different colors show temperature at certain distance from the end at t=16 (active cursor), as a function of resistance.

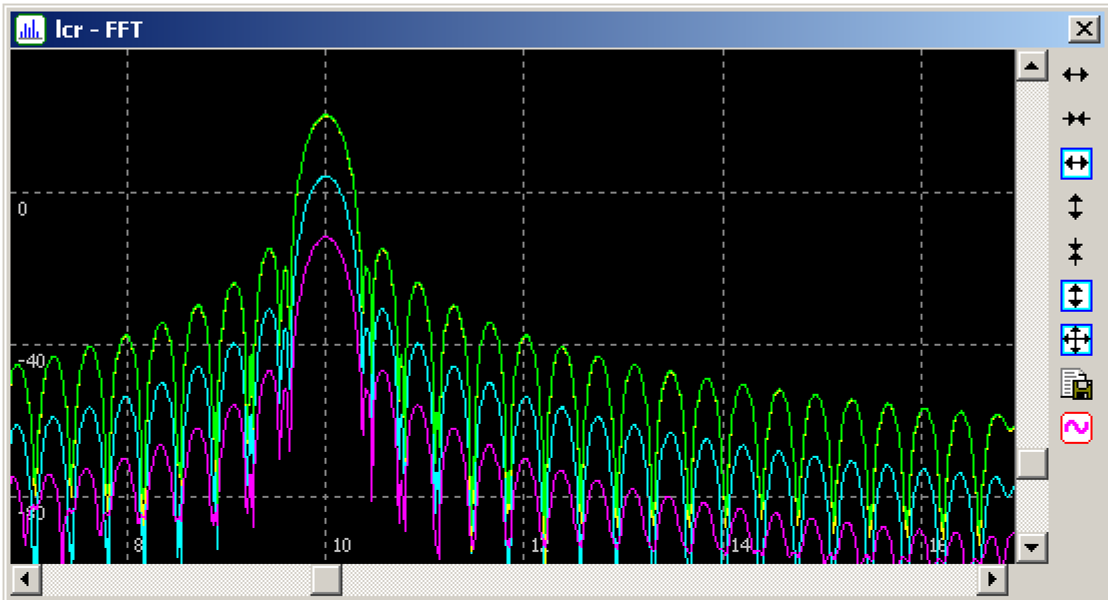




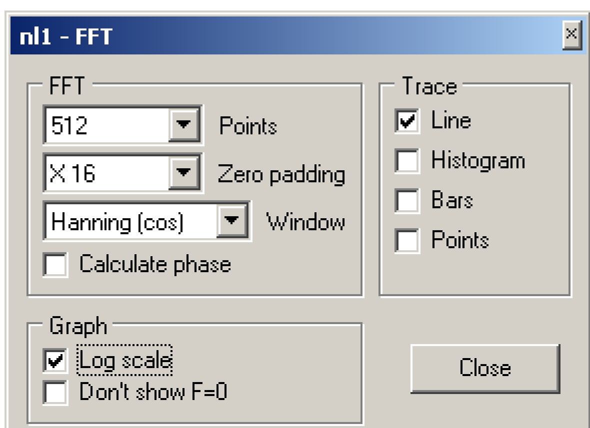
One should notice that "cross-section" of Storage histogram at R=1 is the same as upper line of Trace histogram shown in the previous example.

FFT

FFT (Fast Fourier Transform) is calculated between cursors only (or on the screen, if cursors are disabled), for all traces displayed on the graph. FFT graph does not show phase.



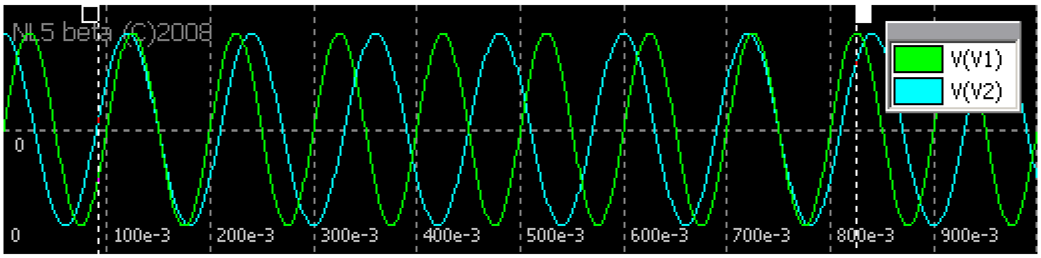
- **Right-click** on the window to access relevant commands.
- **Mouse wheel** with **Ctrl and Shift** modifiers can be used for scrolling and zooming, exactly as in Transient Window.
- **Double-click** on the window to open Configuration dialog box:



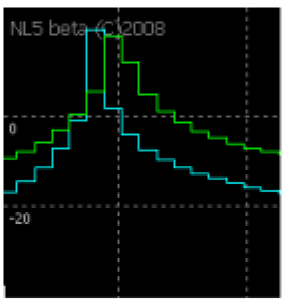
- ○ **FFT.**
  - **Points.** Number of FFT points, **256**…**1048576** ($2^8 \ldots 2^{20}$).
  - **Zero padding.** Adds zeroes to transient samples. Can be **None** to **X16**. Zero padding is a standard technique to improve spectral resolution.

- **Window.** Windowing is a standard technique to reduce leakage effects and improve spectral resolution. 10 windows (including rectangle) are currently available.
- **Calculate phase.** Although phase does not make much sense for FFT and is not displayed on the FFT graph, it can be calculated and displayed in AC window (see **Show in AC window**).

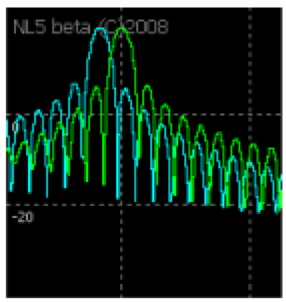
The following example shows Zero padding and Window effect on FFT result:



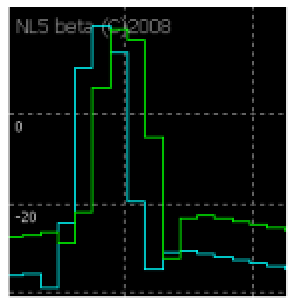
Transient



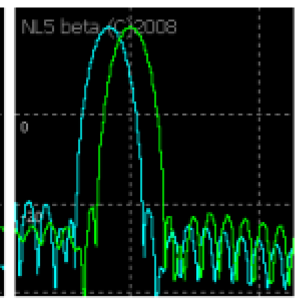
No zero padding
No window



X16 zero padding
No window

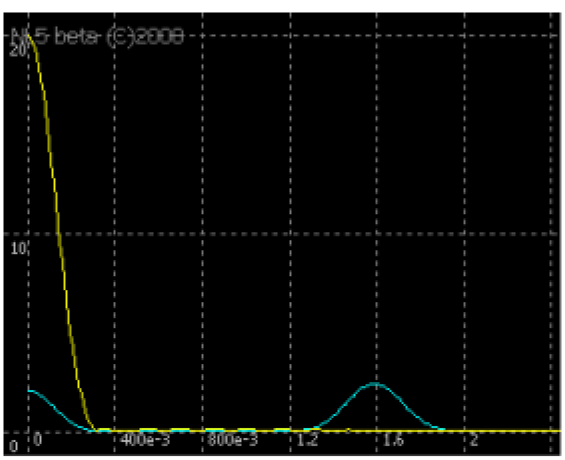


No zero padding
Hamming window

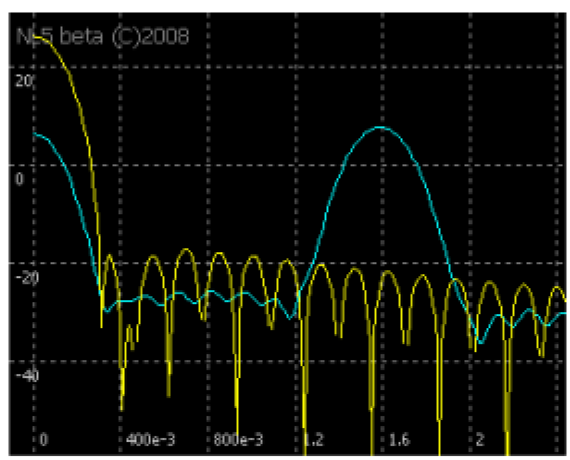


X16 zero padding
Hamming window

- **Graph.**
  - **Log scale.** If selected, amplitude is shown in dB.



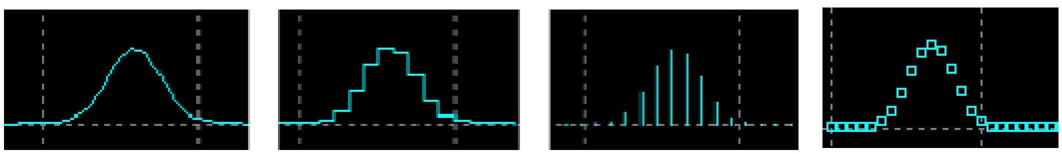
Linear scale



Log scale

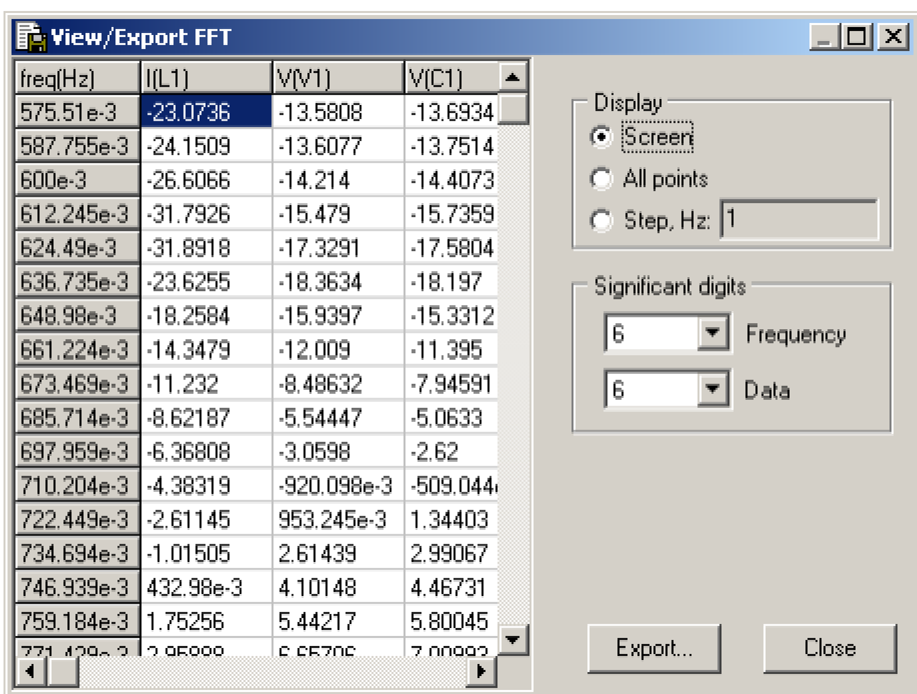
- **Don't show F=0.** If selected, zero frequency point is not shown on the graph. Select this option if DC component of the signal is not of interest.

- **Trace.** Several trace options can be selected at a time.
  - **Line.** Show straight lines between points.
  - **Histogram.** Show histogram-like traces.
  - **Bars.** Show vertical lines from zero to the trace point. Not available for Log scale.
  - **Points.** Show data points as squares.



Line Histogram Bars Points

- **Horizontal Zoom-in.**
- **Horizontal Zoom-out.**
- **Fit the screen horizontal.**
- **Vertical Zoom-in.**
- **Vertical Zoom-out.**
- **Fit the screen vertical.**
- **Fit the screen.**
- **View/Export.** View FFT data in the text table, export to text or "csv" file. **View/Export** dialog box will show up:



Text table shows amplitude of all FFT traces. Select **Display** mode to show:

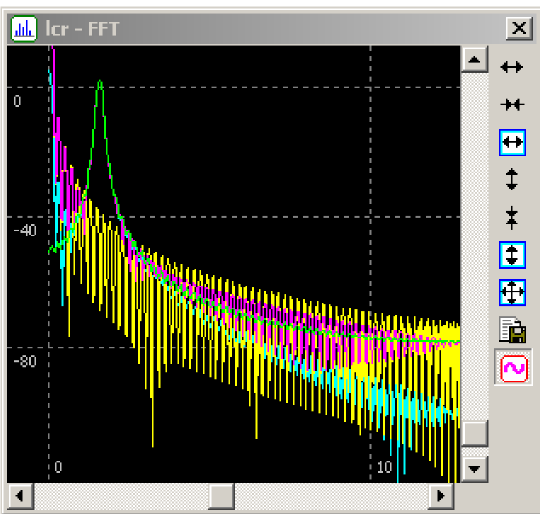
- **Screen**. Show points in the frequency range visible on the screen only.
- **All points**. Show all calculated FFT points.
- **Step, Hz**. Show points with specified frequency step. This mode can be used to see only harmonics of specified frequency.

Specify **Significant digits** for **Frequency** and **Data** columns of the table.

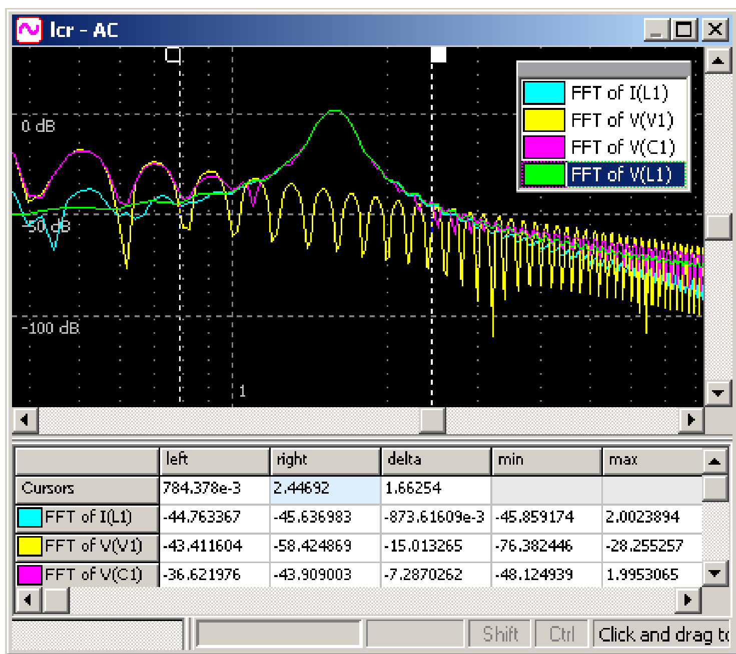
Click **Export** to export the table into text or "csv" file.

- **Show in AC window.** If selected, FFT traces will be shown in the AC window as well as in the FFT window. The name of the traces will be "FFT of ***trace name***". If option is unselected, FFT traces will be deleted in AC window. If FFT window is closed with option selected, the traces will not be deleted. Working with AC traces in AC window allows convenient zooming and scrolling, using of cursors and Data table, and it also shows phase. FFT traces can be duplicated in AC window to be used as a reference for future FFT analysis. Please note that FFT of storage is not shown in AC window.

FFT window

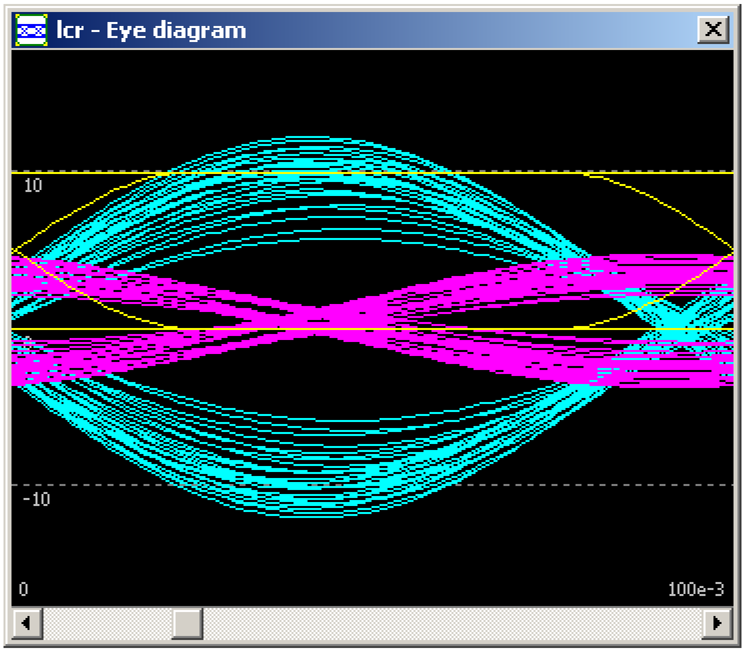


AC window

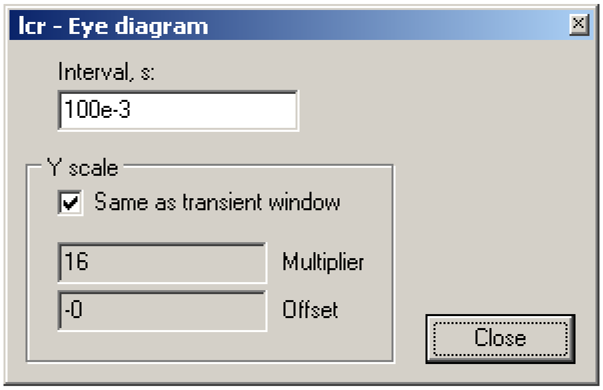


Eye diagram

Eye diagram is used to analyze periodical signals jitter and distortion. Although it is mostly applicable to telecommunication and digital processing, it might be useful for analog electronics as well. The eye diagram window is similar to oscilloscope, continuously running with specified trigger interval. The diagram shows traces between cursors only (or on the screen, if cursors are disabled).



- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:

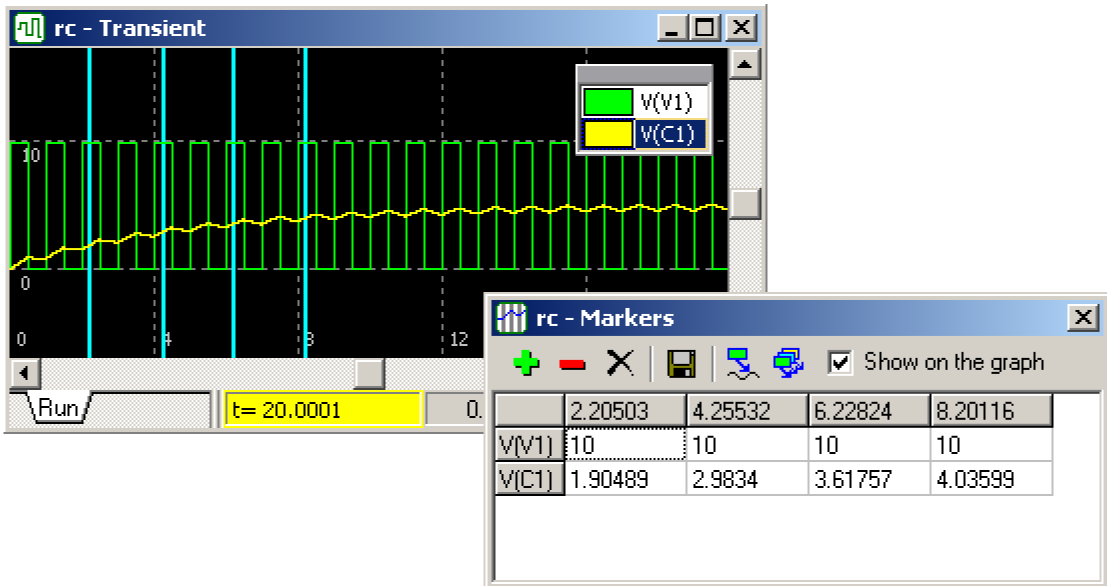


  - **Interval, s**. Eye diagram width (sampling period).
  - By default, diagram Y scale is the same as transient screen vertical scale. Uncheck **Same as transient window** checkbox to enter Y scale **Multiplier** and **Offset.**

- Use horizontal **scroll-bar** to change sampling phase.

Markers

Markers tool provides a convenient way to monitor traces amplitudes at specified points. Unlike cursors, markers always stay at specified position. Number of markers is not limited. Below you can see 4 markers shown on the transient graph, with traces amplitude displayed in the **Markers table**.

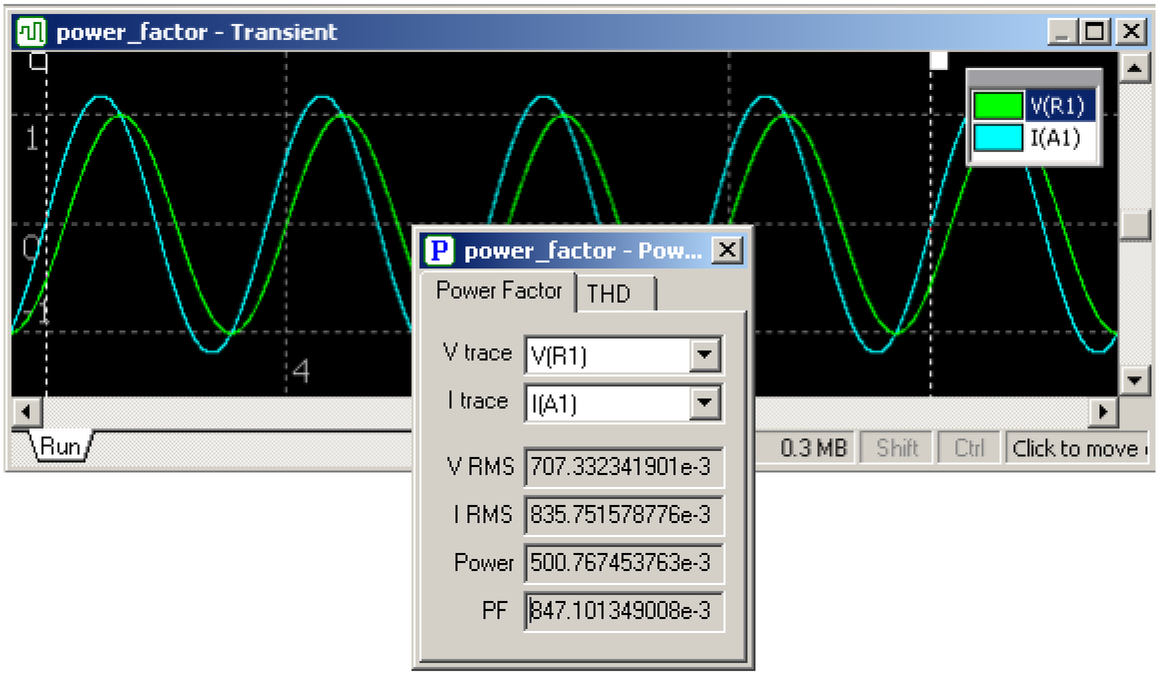


- **Markers table** consists of the traces displayed on the graph.
- **Add** new marker. Marker time is a time of selected cursor.
- **Remove** marker from the table. Select (click) any cell in the table, which belongs to the marker's column, then click the button. On the example above, the first marker (t=2.20503) will be removed.
- **Delete** all markers.
- **Export** markers table in the text or "csv" file.
- **Annotate** selected trace at markers positions on the transient graph.
- **Annotate all traces** at markers positions on the transient graph.
- **Show on the graph**. Select to show markers on the transient graph. Markers width and color can be changed on **Graphs** page of **Preferences** dialog box.

Power

Power tool calculates Power Factor and THD (Total Harmonic Distortion).

Select **Power Factor** page to calculate voltage and current RMS, power, and Power Factor.

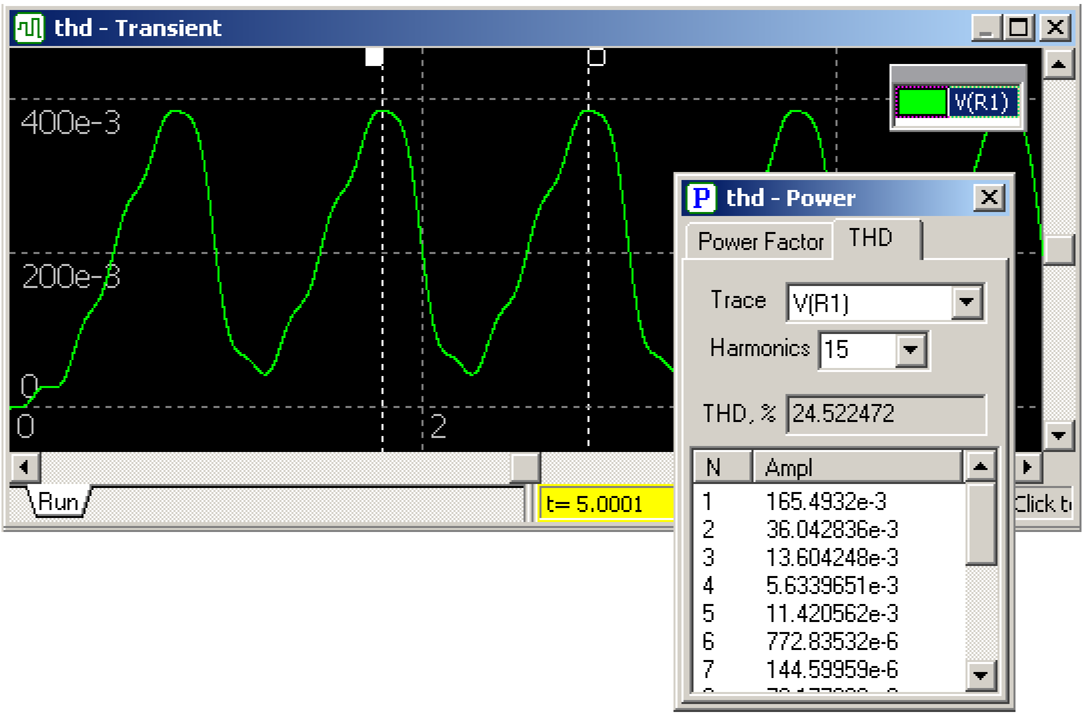


- **V trace.** Select voltage trace from the drop-down list.
- **I trace.** Select current trace from the drop-down list.

Power factor PF is calculated as: $PF = \frac{Power}{VRMS \times IRMS}$

All parameters are calculated between cursors only (or on the screen, if cursors are disabled), so make sure selected interval consists of integer number of signal cycles.

Select **THD** page to calculate Total Harmonic Distortion.

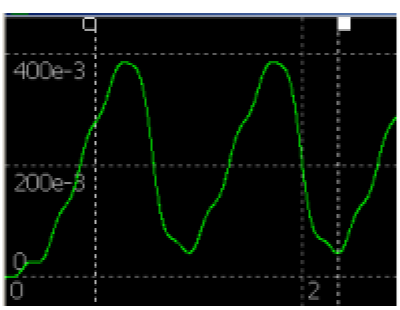


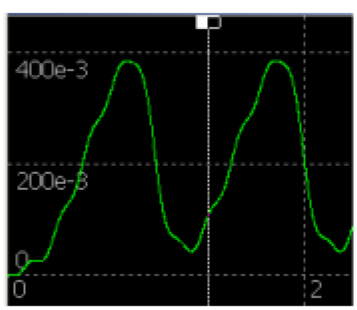
- **Trace.** Select trace from the drop-down list.
- **Harmonics.** Enter or select from drop-down list number of harmonics to calculate (max=40).

THD is calculated as: $THD = \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + \ldots}}{V_1}$, where V_n is amplitude of n^{th} harmonic.

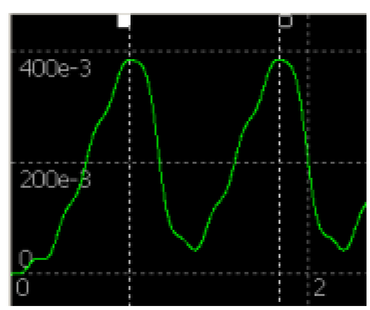
THD is calculated between cursors only (or on the screen, if cursors are disabled), so make sure selected interval consists of exactly 1 period of the signal fundamental (1^{st} harmonic). There are several convenient ways to make such a selection easy. For example:

- **Double-click** on the transient graph to set both cursors at one point, between signal maximums. Click **Maximums** button  , one cursor will be moved to the nearest right maximum, and another cursor to the nearest left maximum. This method may not work right if the signal has local maximums.



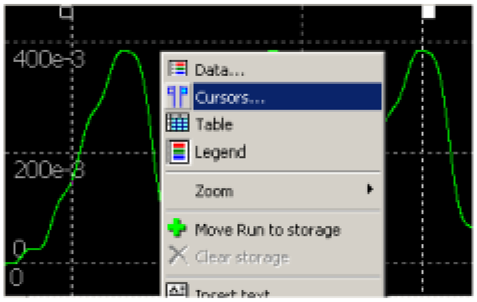


Double-click

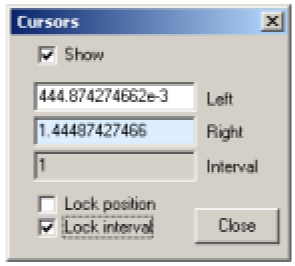


Click **Maximums** button

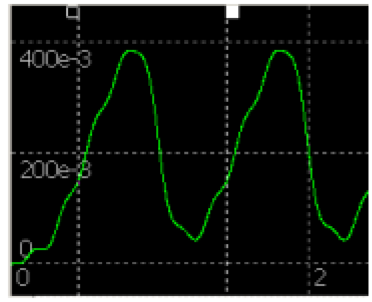
- If period of fundamental is known, **Right-click** on the transient graph and select **Cursors** command. In the **Cursors** window, enter period value in the **Interval** window and select **Lock interval** checkbox. Now interval between cursors will always be equal to the period of fundamental.



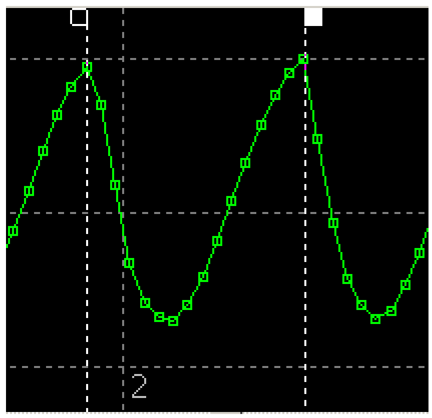
Right-click, select **Cursors**

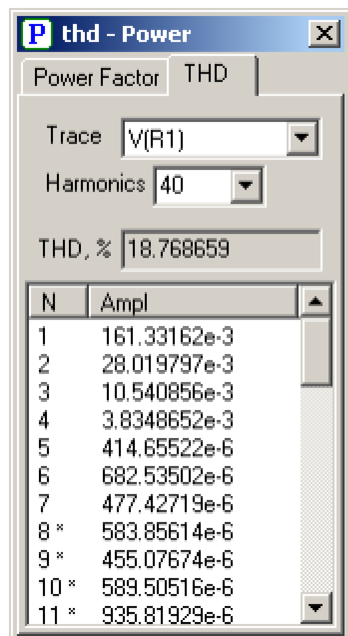


Enter **Interval,**
select **Lock interval**

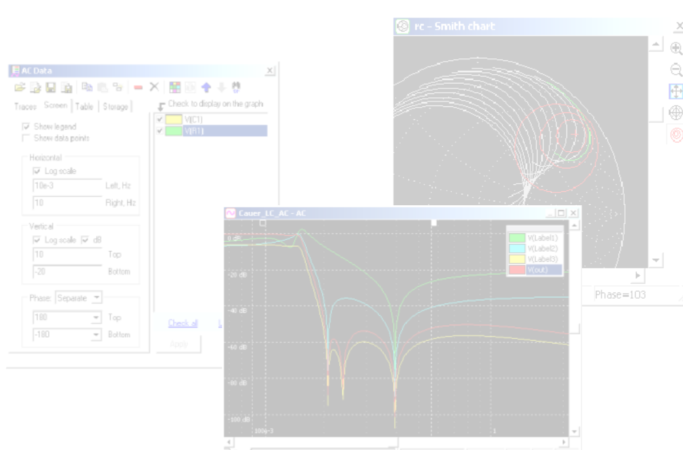


If calculation step is big enough, some higher harmonics may not satisfy Nyquist criteria: harmonic frequency exceeds half of "discretization" frequency. Such harmonics will be marked with an asterisk (*) symbol in the harmonics list.

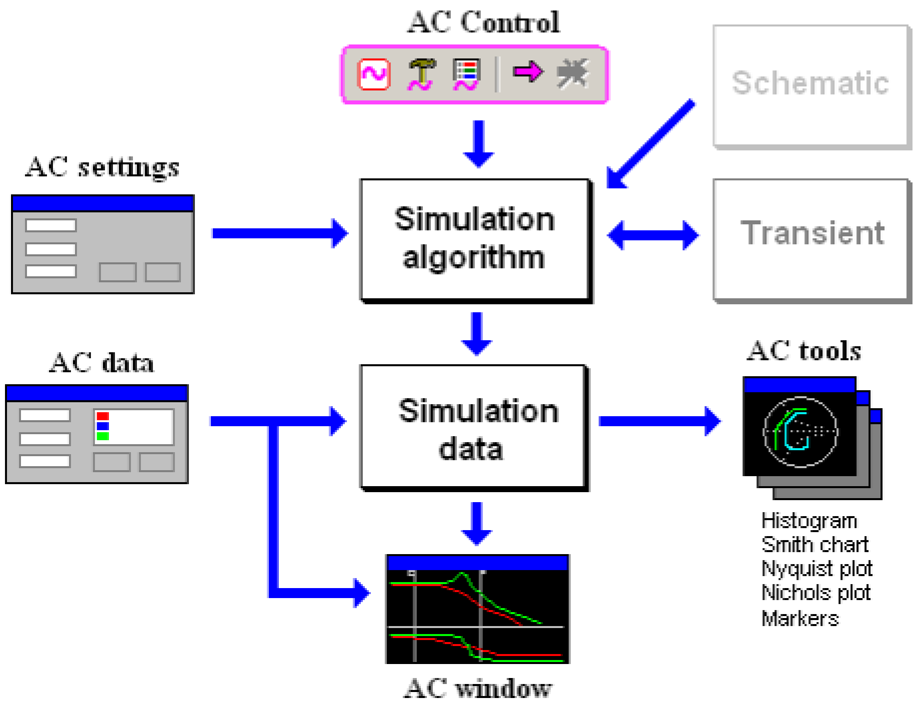




V. AC Analysis



The following simplified diagram explains AC simulation process:



Simulation algorithm is configured by **AC Settings** dialog box, and controlled by **AC Control** commands (Main Menu and Toolbar). The results of **Schematic** simulation are stored into **Simulation data**, and simultaneously displayed as a graph in the **AC Window**. **AC Data** window is used to configure what simulation data are to be stored, and how the data are displayed. In addition, the data can be used by **AC Tools**, which offer a variety of AC data analysis and data presentation. **Transient** is used for **Sweep AC source** simulation method.

Simulation

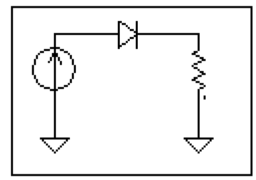
There are two methods of AC analysis in NL5: **Linearize schematic** and **Sweep AC source**.

Linearized schematic method

Linearize schematic is a standard low-signal AC analysis. First, all non-linear components are replaced with linear equivalents at their operating point. Second, a signal of specified frequency with unit amplitude and zero phase is applied to the input node, and signals at other nodes are found by solving a system of linear equations. The process is repeated for specified number of frequencies.

In order to linearize schematic, state of all components should be known. It can be done manually by setting Initial Conditions (IC) for all non-linear components, diodes, and controlled switches, or by automatic calculation of DC operating point (**Calculate DC operating point** check box in the **AC Settings** dialog box). DC operating point is calculated exactly as in transient analysis.

The method always works for linear circuits. The method can be also used for circuits with non-linear components, only if those components can be properly linearized at operating point: infinitely small amplitude of input AC signal should not change state of the components. For instance, the following circuit can't be correctly analyzed by this method, as the diode will change its state every time input AC signal changes polarity.



The method can't be used for switching-type circuits, since all the switches will be set to either open or closed state, and will not be switching as required.

Sweep AC source method

Sweep AC source method allows calculating AC response for any type of circuit. A real sine-wave signal of specified amplitude and frequency is applied to the input node; the transient analysis is automatically performed; finally, the harmonic of specified frequency is extracted form signals of interest by Discrete Fourier Transform (DFT). AC response is calculated by comparison of the harmonic amplitude and phase of the specified signal to the input signal. This process repeats for specified frequency range. A number of parameters required for the method can be set up in the **AC Settings** dialog box.

Transient simulation is automatically started and controlled by the method. You don't need to define Transient traces: all the required traces will be automatically created and removed when AC analysis is complete. The only transient-related parameter to be set up is **Transient calculation step**. Set it to the value you would use for a normal transient analysis of the schematic without input AC source. When AC analysis is running, transient simulation step may be automatically decreased as needed, in order not to exceed 1/16 of the AC source period.

AC source amplitude (voltage or current) can be a constant, or a function of AC source frequency **f**. Frequency-dependent amplitude may be helpful to provide reliable operation of the circuit at different AC source frequency. For instance, if switching-mode schematic has a gain decreasing with frequency, it might be helpful to have AC source amplitude increased with frequency, in order to increase signal-to-noise ratio.

When AC source is given a specific frequency, transient simulation is performed for time interval, equal to some number of periods of that frequency. The more periods of AC source are used for calculations, the better accuracy of AC response can be achieved. However, this may result in extremely long simulations time, especially at low frequencies. To set up a desired balance between accuracy and simulation time, **Error** parameter can be used. In fact, this is more qualitative description of the expected simulation performance, rather than actual simulation error. The following values of **Error** parameter can be used:

Error = 0 – at each AC frequency, transient simulation will be performed for 128 periods of the frequency. This is the most accurate option, however it could be a very long simulation time.

0 < Error < 100 – number of periods at each frequency will be automatically selected between 2 and 128 based on schematic response. Expected accuracy is very high for numbers 1 and below, good for 10 to 50, and fair for 50 and above. Respectively, higher accuracy requires longer simulation time.

Error = 100 - at each AC frequency, transient simulation will be performed only for 2 periods of the frequency. This is the fastest option, however the accuracy could be extremely poor.

Recommended simulation strategy is the following. First, evaluate schematic at value 100, just to see if the method works in general and schematic behavior is somewhat close to expected. Then use value 10 to perform schematic analysis with reasonable accuracy and simulation speed. If better accuracy is needed, or simulation result does not look correct (which may happen for some specific schematic types), use value 1.

To perform AC analysis of switching circuit, a "periodic operating point" should be found first. If **Find periodic operating point** option is selected, NL5 will automatically run transient with zero amplitude of AC source until periodic steady-state condition is satisfied, and only after that AC analysis will start. If **Find periodic operating point** option is unselected, it is highly recommended to find periodic steady state condition manually, by running transient with zero amplitude of AC source. When steady state is reached, save current state of the schematic into Initial Conditions (select **Transient | Save IC** in the Main menu). If schematic is not in its periodic steady state, the correct results of AC analysis are not guarantied.

When **Sweep frequency from high to low** option is selected, simulation will start from higher frequencies, where it usually takes much less simulation time per one frequency. Thus, simulation results will appear on the graph faster, and user can make a decision to stop simulation without waiting for lower frequencies results, if needed.

Show transient data option makes transient simulation data visible during AC simulation, and keeps the data available when AC simulation is complete. Transient data might be useful for finding optimal AC analysis settings and for troubleshooting.

Simulation data

Traces. During simulation NL5 is storing data into memory. The data to be stored is selected by user as **traces** in the **AC Data** window. Several types of transient traces are available: **V** (voltage), **I** (current), **Z** (impedance), **Gamma**, **VSWR**, and **Function**.

When simulation starts, all traces are automatically cleared, and then start storing new simulation data. A new data is displayed in the **Run** tab of **AC Window.** The last data can be moved into **storage** with special tab in the AC Window assigned to it. Storage data is not automatically cleared, and can be used for comparing results of different simulation runs.

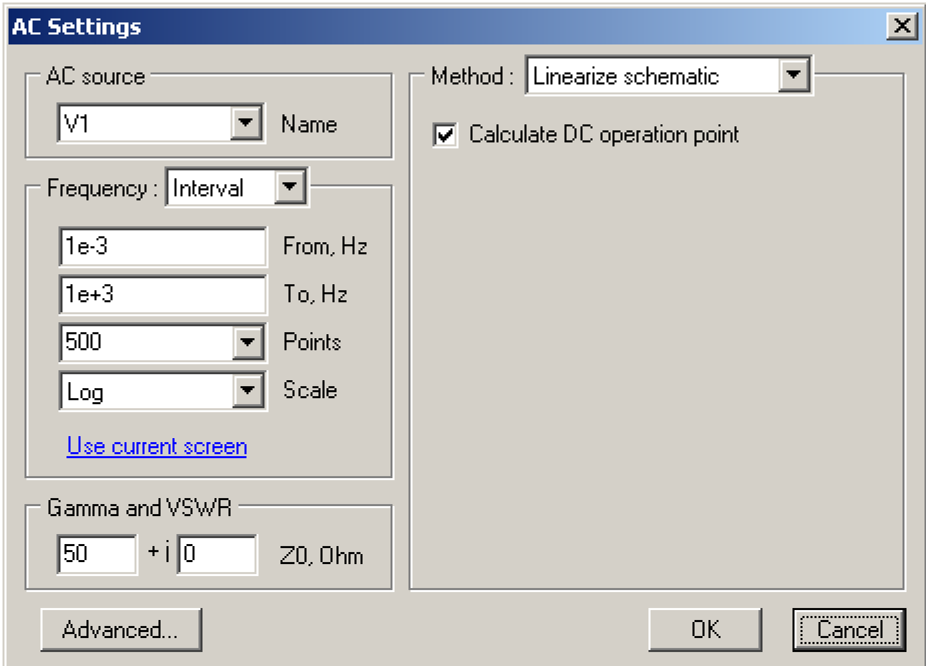
If special option **Store last Run** is selected, then when new simulation starts, current simulation data (**Run**) will be moved into storage under the name **Last**. If run with the name **Last** already exists, it will be overwritten. Thus, previous simulation data will always be in the storage under the name **Last** and can easily be compared with the current simulation data.

Traces can be copied to clipboard, saved into "nlf" data file, or exported into text file in "csv" format. In turn, the data can be pasted from clipboard, loaded from "nlf" data file, or imported from text file as a new trace. Such a trace is always displayed in the **AC Window**, regardless what data tab is selected. It is not cleared when new run starts, and can be used as a reference trace for simulation. It also can be renamed to arbitrary text.

Simulation data is stored in the operating memory.

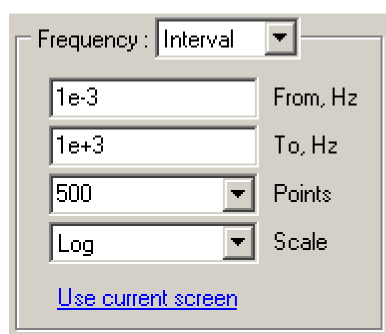
AC Settings

Click **AC settings** toolbar button  , or select **AC | Settings** command. **AC Settings** dialog box will show up:



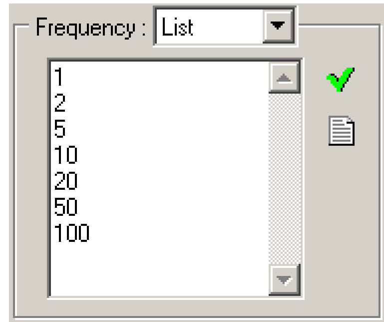
AC Source. Enter the name of AC source component, or select the name from drop-down list. Any voltage source, current source, or label can be used as AC source. The AC source component may have any model (except **File** and **SubCir**), so that there is no need to set the model of the component to **Sin:** the model will be set to **Sin** automatically and restored back what AC analysis complete. When AC analysis is being performed, the DC voltage/current of the component will be set to its DC value at t=0, and AC voltage/current required for AC analysis will be added to that DC level. AC source component name will be marked with **(AC)** text on the schematic and in the Components window. AC source component can also be selected from Schematic context menu, and by Components Window **Set AC source** button  .

Frequency. Select the method of how frequency simulation points are specified. When simulation starts, AC window frequency range is automatically set to the range between minimum and maximum specified frequencies.



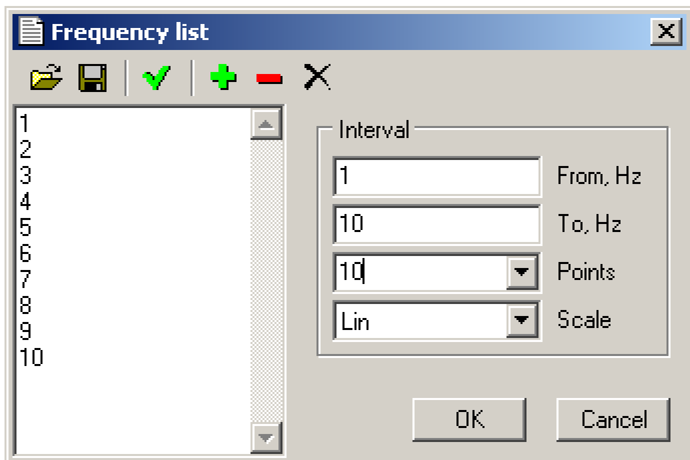
- **Interval**. Specify frequency interval, number of simulation points, and scale.
  - **From, Hz**. Start frequency.
  - **To, Hz**. End frequency.
  - **Points**. Number of simulation points.
  - **Scale**. Frequency scale:
    - **Log.** Logarithmic.
    - **Lin**. Linear.

- ○ **Use current screen**. Click to use current AC screen left frequency, right frequency, and scale as a new simulation interval. **From**, **To**, and **Scale** parameters will be set according to what is currently displayed on the AC graph.



- **List**. Specify custom frequencies. This method can be used to specify more frequency points in the range where AC response is changing fast, in order to optimize simulation time (especially for **Sweep AC Source** simulation method). Enter frequencies one number per line. Frequencies can be entered in any order. Click **Sort and refresh** button to verify and sort entered numbers.

  Click **Edit frequency list** button to edit frequency list in the **Frequency List** dialog box.



The dialog box allows entering frequencies in the list manually, as well as performing the following operations:

- ○ **Open frequency list** from the text file.
- ○ **Save frequency list** to the text file.
- ○ **Sort and refresh** frequency list.
- ○ **Add interval.** Add frequencies specified in the **Interval** box (**From**, **To**, **Points**, and **Scale** parameters) to the frequency list. Frequencies currently existing in the list will not be removed.
- ○ **Remove interval.** Remove frequencies specified in the **Interval** box (**From**, **To**) from the frequency list. All frequencies in the **From…To** interval will be removed.
- ○ **Delete all.** Clear frequency list.

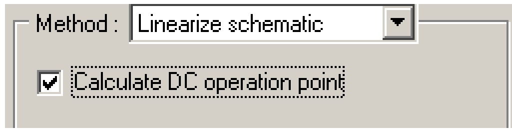
Gamma and VSWR parameters.

- **Z0, Ohm.** Characteristic impedance for Gamma and VSWR traces.

- **Advanced.** Click to open Advanced Settings dialog box. See **Transient settings** chapter for details.

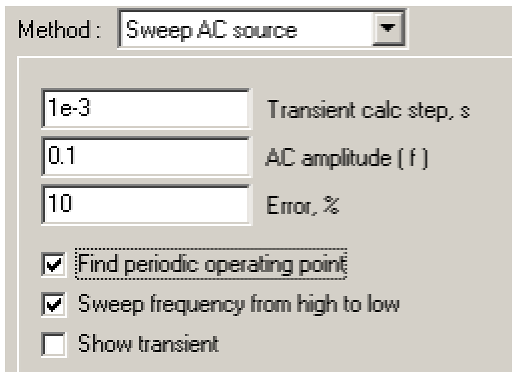
- **Method.** Select AC simulation method. See **Simulation Algorithm** chapter for details on simulation methods.

    - **Linearize schematic.**



        - **Calculate DC operating point**. If selected, DC operating point will be calculated prior to AC analysis. This option does not required for linear circuits, or if Initial Conditions for all components are manually defined.

    - **Sweep AC source**.



        - **Transient calc step, s**. Transient calculation step.
        - **AC amplitude (f).** Amplitude of AC source. **V** for voltage source and label, **A** for current source. Amplitude can be a constant, or a function of frequency **f**. For example:

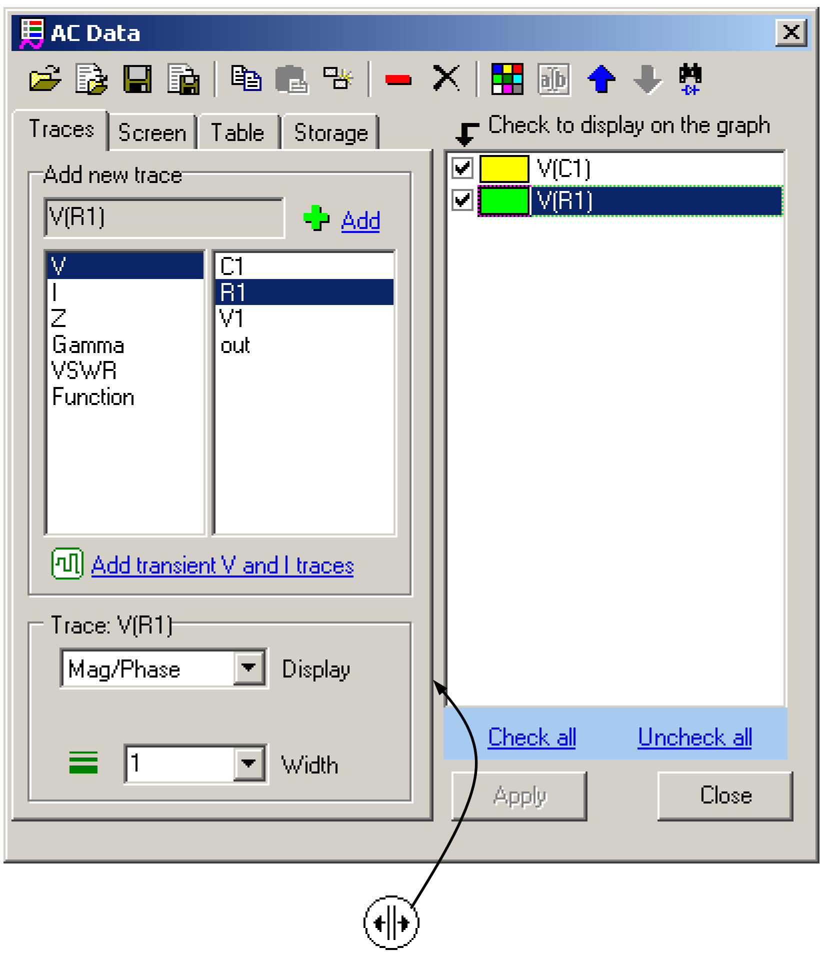
```
0.1
1m*f
1000/f
```

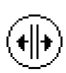
        - **Error, %.** Expected error of AC analysis. See **Sweep AC source method** chapter for details.
        - **Find periodic operating point.** Run transient prior to AC analysis, until periodic steady state is reached.
        - **Sweep frequency from high to low.** Perform AC simulation starting from higher frequencies.
        - **Show transient.** Show transient data during AC simulation, and keep transient data after AC simulation is complete.

AC Data

Click **AC data** toolbar button  , or select **AC | Data** command. **AC Data** window will show up. The window always shows data of active document (schematic). Switching to another document automatically updates the data in the window. The window consists of Toolbar, Trace list, and 4 pages used for the following operations:

- **Traces:** add traces, set up individual trace scales and width.
- **Screen:** set up graph scales, gridlines, and other screen options.
- **Table:** configure data table**.**
- **Storage:** manage storage data.



Move cursor over "splitter" area , then press left mouse button and drag to resize panes.

Trace list shows all currently available traces. Checkbox indicates the following trace property depending on selected page:

- **Trace** and **Screen** page – trace is shown on the graph.
- **Table** page – trace is shown in the table.
- **Storage** page – storage is allowed for trace.

One or more traces can be selected in the list using mouse, **Ctrl**, and **Shift** keys. Click **Check all** to check all al traces, **Uncheck all** to uncheck all traces. Most of toolbar commands apply to selected traces only. Please note: **selected** trace is highlighted in the list, and trace selection state is not related to trace checkbox state. On the screenshot above, both traces are "checked", and only V(R1) is selected.

Double-click on the trace to change the color of the trace.

This chapter describes toolbar commands and **Traces** page only. Other pages are described in the **AC Window** chapter (**Graph**, **Data table**, and **Storage**).

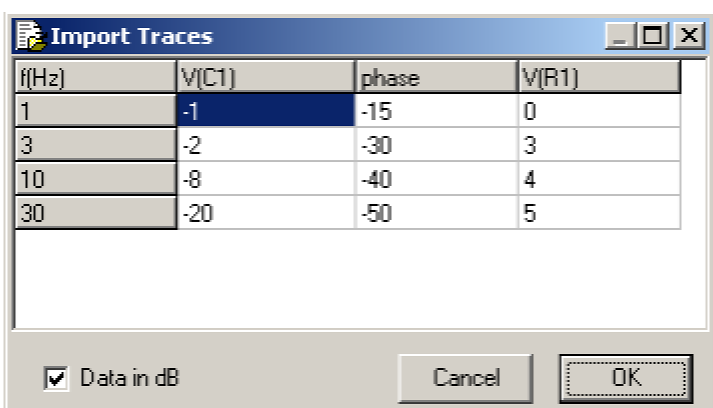
Toolbar

Toolbar button commands apply to all or selected traces. Some of those commands are also accessible through context menus in the **AC Window**.

- **Open file**. Load traces from "nlf" data file.
- **Import traces** from text or "csv" file. Format of the file is similar to the export format. The first line is a header line: it may have any text in the first column, and trace names in other columns. The first column consists of frequency (in Hz), other columns consist trace data. Trace data may have one column per trace, or two columns per trace. The first trace column is magnitude (absolute value, or dB), and it has trace name in the header line. The second trace column, if exists, is phase (in degrees), and it has "phase" text in the header line. If second trace column does not exist, trace phase is set to zero. If trace name consists of symbols other than numbers and letters, it should be enclosed in quotes. The data and names can be comma, space, or Tab separated. For example:

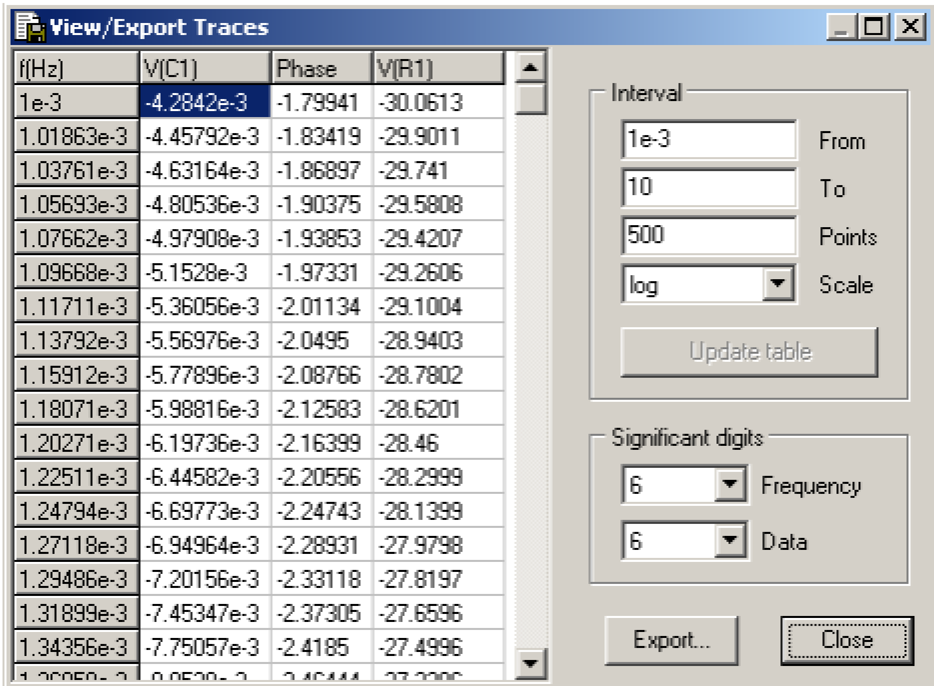
```
f(Hz),V(C1),phase,V(R1)
1,-1,-15,0
3,-2,-30,3
10,-8,-40,4
30,-20,-50,5
```

When file is loaded, its contents will be displayed in the **Import Traces** dialog box, for verifying:



Check **Data in dB** checkbox, if magnitude of the traces is in dB. Click **OK** to confirm import.

- **Save selected traces** into "nlf" data file. Only traces selected (highlighted) in the Trace list will be saved into the file.
- **View/Export selected traces.** Only traces selected (highlighted) in the Trace list will be viewed and exported into text or "csv" file. **View/Export** dialog box will show up:



Selected traces are shown as a text in the table. Initially, traces are shown in the frequency interval between cursors, or, if cursors are disabled, in the full screen. Change **From**, **To, Points,** and **Scale** values and press **Enter**, or click **Update table** button to update table data. Number of **significant digits** for frequency and data columns can be specified.

Click **Export** to save the table into the text file as comma-separated values.

- **Copy selected traces** to the clipboard. Only traces selected (highlighted) in the Trace list will be copied in the clipboard.
- **Paste traces** from the clipboard. Traces from the clipboard will be added to Trace list.
- **Duplicate selected traces.** This operation is equivalent to **Copy**/**Paste** operations. Only traces selected (highlighted) in the Trace list will be duplicated.
- **Remove selected traces.** Only traces selected (highlighted) in the Trace list will be removed.
- **Delete all traces.**
- **Select color** of selected trace. Only one trace should be selected. **Double-click** on the trace in the Trace list performs the same operation.

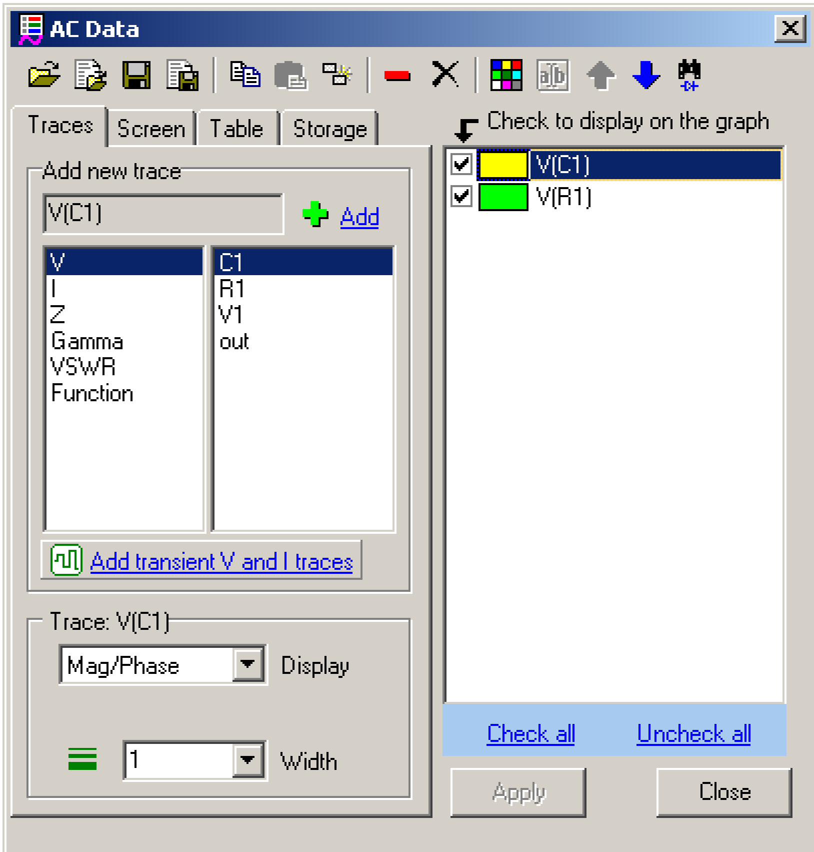
- **Rename trace.** Only one trace should be selected. Only Z, Gamma, VSWR traces, and traces loaded from data file, imported from text file, or pasted from clipboard can be renamed. Renaming the trace of **Function** type changes the function. **Rename** dialog box will show up:

Enter new trace name and click **OK**.

- **Move selected traces up.** This operation changes the order of traces in the list, on the graph, and in the data table.
- **Move selected traces down.** This operation changes the order of traces in the list, on the graph, and in the data table.
- **Find component.** If selected trace is V, I, Z, Gamma or VSWR, click to show the component on the schematic. The component will be selected (highlighted) and centered on the screen.

Traces

Traces page of **AC Data** window is used to add traces and set up individual trace scales and width.



Add new trace. Select trace type in the left list:

- **V** – voltage.
- **I** – current.
- **Z** – impedance.
- **Gamma** – reflection coefficient.
- **VSWR** – Voltage Standing-Wave Ratio.
- **Function** – arbitrary function.

If **V** or **I** trace is selected, the right list will show components available for this trace type: the model of the component should support selected type. Select component and click **Add** button  , or double-click on the component name to add new trace to the trace list. The name of the trace consists of the letter followed by the name of component in parentheses:

```
V(R1), I(C2)
```

If **Z**, **Gamma**, or **VSWR** trace is selected, click **Add** button  to add new trace to the trace list. Traces show Z, Gamma, and VSWR relative to AC source.

If **Function** trace is selected, enter function in the edit window and click **Add** button to add new trace to the trace list. The function may consists of arithmetic operators and functions, component parameters, V and I on the component, and the following variables:

`f` – current AC frequency.
`w` – angular AC frequency, w = 2πf.
`s` or `p` – Laplace parameter, s = p = j*2πf .

For example:

```
V(C1)/I(C1)
V(X1.V1)
1/(1+s)
V(R1)*f
```

The name of the trace is the function itself: so that renaming the trace will change the function.

V and **I** traces can also be added from Schematic context menu, and by Components Window toolbar buttons.

The following individual trace parameters can be set:

- **Display.** Specifies what is displayed on the graph and shown in the Data table:
  - **Mag/Phase.** Magnitude and phase.
  - **Mag.** Magnitude only (absolute value)
  - **Phase.** Phase only.
  - **Re.** Real part. Can be used to display R of Z trace.
  - **Im.** Imaginary part. Can be used to display X of Z trace.
- **Width.** Width of the trace line in pixels.

Select one or more traces in the Trace list, change parameters and press **Enter**, or click **Apply** button. If selected traces have different values for some parameter, the corresponding field will be left blank. Leave the field blank to keep individual values unchanged, or enter a new value to apply it to all selected traces.

To display both **Re** and **Im** of the trace, add this trace into trace list twice, then select **Re** for one trace, and **Im** for another trace.

Performing simulation

Use Menu commands, Toolbar buttons, or hotkeys to perform AC simulation.

- **Start AC** (**AC | Start**, or **F9).** When AC simulation starts, the **AC Window** opens up, and the frequency range and scale (log/lin) of the screen is set to the values specified in the **AC Settings** dialog box. While simulation is running, results are immediately displayed in the **AC Window**, and a progress bar is shown in the status bar of the window.

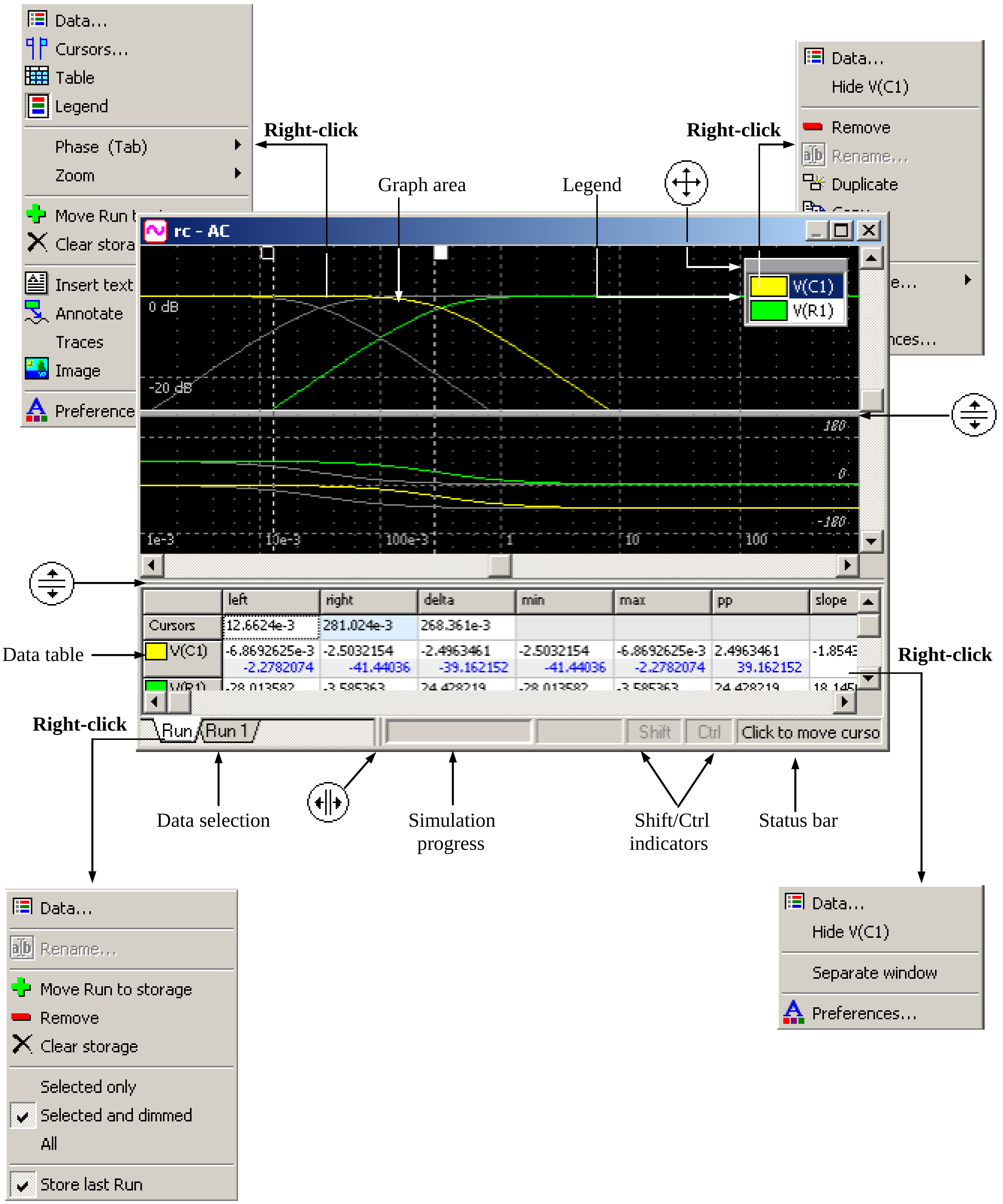
- **Stop AC.** While AC simulation is running, this is **the only** button available.

- **AC Log** (**AC | Log**). Log information shown in the dialog box may be useful for troubleshooting. The last log is saved into schematic file. When submitting schematic file to Customer Service for help, please save schematic after simulation, in order to have last log included into the file. Click **Copy to clipboard** button to save log text into clipboard.

- **Sweep** (**AC | Sweep**) allows running series of AC runs while changing component parameter or variable in specified range, and storing AC data in the storage. Sweep is performed using script commands, and is configured on **Sweep** page of **Tools** window. See **Tools**, **Sweep** chapter for details.

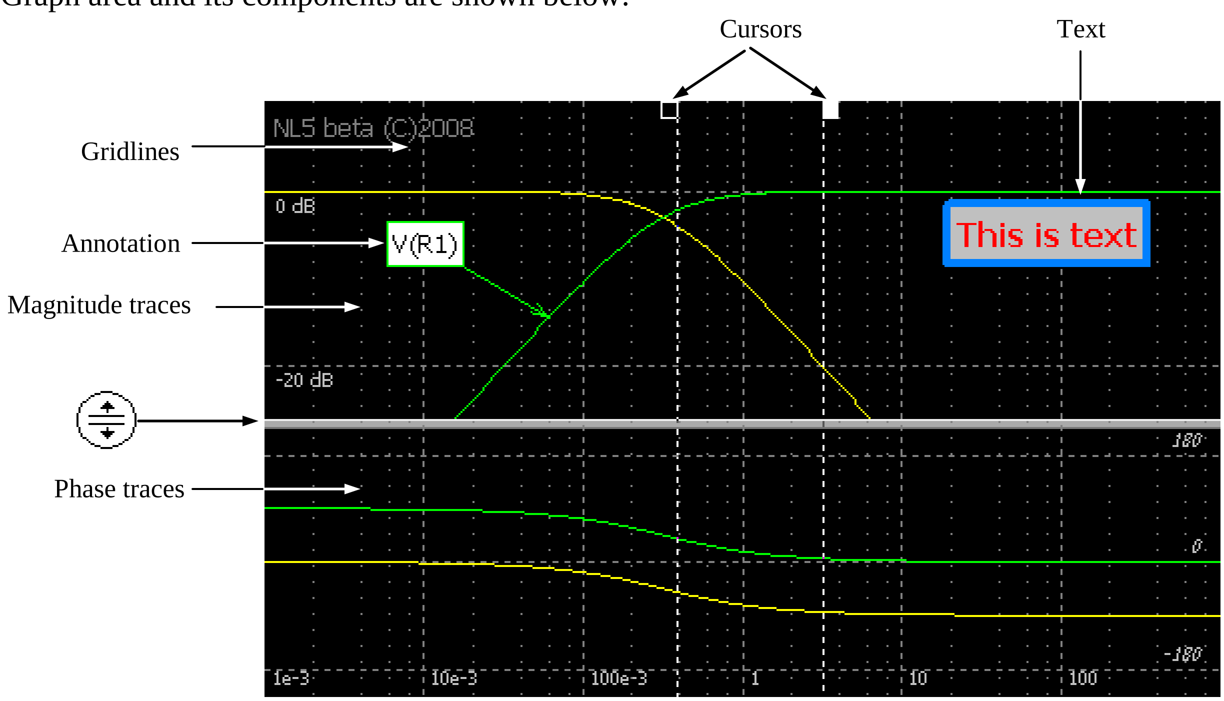
AC window

Typical view of AC window and its main components are shown below:



- **Graph** area contains traces with annotations, cursors, and text.
- **Legend** window contains list of traces shown on the graph. Click on the gray header bar of the legend window and drag to move legend window.
- **Data table** contains cursors/screen information and calculated traces data.
- **Data selection** area contains last simulation and storage data tabs. Click on a tab to select **Run** or storage data.
- **Simulation progress** shows current simulation status as a progress bar.
- **Shift/Ctrl indicators** are highlighted when **Shift** and/or **Ctrl** key are depressed.
- **Status bar** shows hint related to current position of mouse pointer and **Shift/Ctrl** state.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize Storage selection area.
- Move mouse pointer over "splitter" area , then press left mouse button and drag to resize data table, and magnitude/phase separator.
- **Right-click** on the graph, legend, Data table, or Data selection area to see context menu with relevant commands.
- Common properties of **AC Window**, such as colors, fonts, and some options, can be customized on **Graphs, Table, Annotation,** and **Text** pages of **Preferences** dialog box. Properties specific to the document (schematic) can also be set up in **AC Data** window.

Graph area and its components are shown below:



Graph

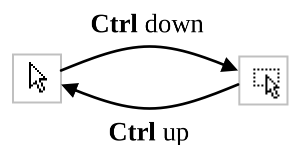
Graph navigation can be performed by commands available in the AC context menus, AC toolbar buttons, shortcuts, keyboard keys, and mouse. Very often, the same operation can be performed by different ways. For instance, zooming graph in/out can be done using keyboard keys only, mouse only, or both. It is user's choice to use the most effective and convenient one.

There are 3 graph operation modes:

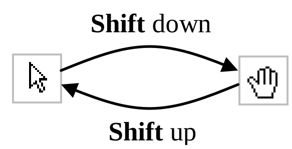
- ***Cursors*****.** Moving cursors.
- ***Zoom***. Zooming graph using mouse.
- ***Scrolling***. Scrolling the graph.

The mode can be selected by clicking the button on the AC toolbar. Also, there are quick ways to switch from ***Cursors*** mode to ***Zoom*** and ***Scrolling*** modes:

- Press and hold **Ctrl**, click and drag mouse to zoom the graph. Release **Ctrl** to return to ***Cursors*** mode:



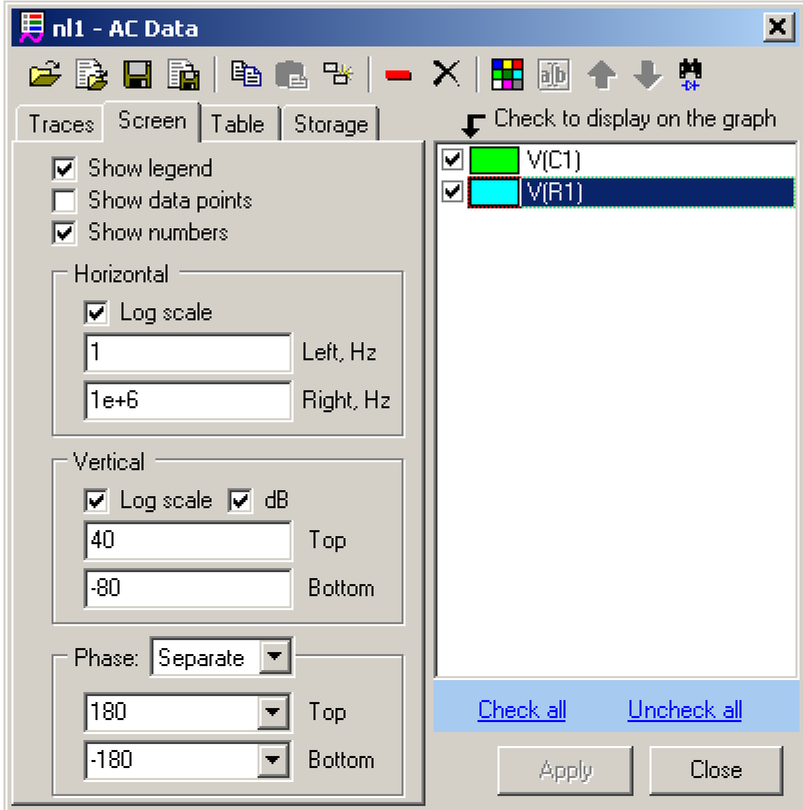
- Press and hold **Shift**, click and drag mouse to scroll the graph. Release **Shift** to return to ***Cursors*** mode:

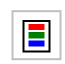


Traces are shown on the graph with their individual width and colors defined on the **Traces** page of **AC Data** window. All traces have the same horizontal and vertical scale.

Gridlines spacing is selected automatically so that last significant digit step is 1, 2, or 5, and distance between gridlines is approximately equal to the value specified on **Graphs** page of **Preferences** dialog box as **Gridlines interval** in pixels. If scale is logarithmic, gridlines are automatically adjusted to provide best view.

Scales, gridlines, and some other options of the graph can be changed on **Screen** page of **AC Data** window:



- **Show legend**. Select to show **Legend** window. Also use **Legend** button  on the Toolbar or context menu command.
- **Show data points.** Select to mark calculated data points of all traces as small squares.
- **Show numbers.** Select to show scale numbers.

Horizontal. Set up horizontal scale and gridlines.

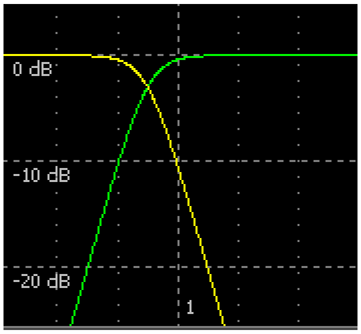
- **Log scale**. Select for logarithmic frequency scale.
- **Left**. Frequency at the left edge of the screen.
- **Right.** Frequency at the right edge of the screen.

Vertical. Set up vertical scale and gridlines.

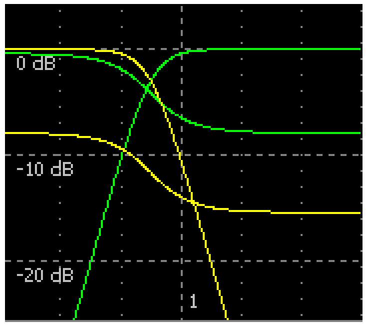
- **Log scale.** Select for logarithmic magnitude scale.
- **dB.** Select to show vertical scale in decibels.
- **Top.** Magnitude at the top of the screen.
- **Bottom.** Magnitude at the bottom of the screen.

Phase. Select phase display mode:

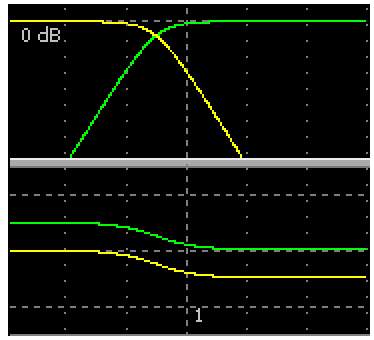
- **Off.** Do not show phase.
- **On.** Show magnitude and phase in the same area of the graph.
- **Separate.** Show magnitude and phase in separate areas of the graph.



Phase Off



Phase On

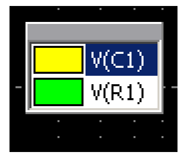


Separate

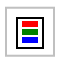
Press **Tab** in the **AC Window** to toggle Phase display mode.

- **Top.** Phase at the top of the screen.
- **Bottom.** Phase at the bottom of the screen.

Legend

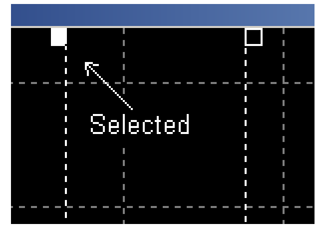


Legend window contains list of traces shown on the graph.

- To show/hide legend click **Legend** button  on the Toolbar or context menu, or use **Show legend** checkbox in the **Screen** page of **AC Data** window.
- **Click** on the trace to select the trace. Selected trace will be shown on top of all traces.
- **Double-click** on the trace to select trace and to open **AC Data** window.
- **Right-click** to select trace and open context menu. The menu will contain some common commands, and commands related to selected trace.
- **Click** on the gray header bar of the legend and drag to move the window.
- Legend font size can be selected on the **Legend** page of **Preferences**.

.

Cursors

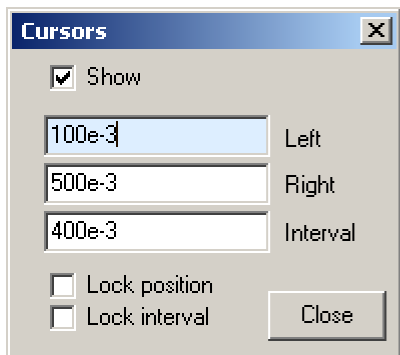


Cursors are used mostly for selecting frequency interval on the graph for data table calculations. Selected (active) cursor is shown with solid colored square on the top. To show/hide (enable/disable) cursors click **Show/hide cursors** Toolbar button .

Select ***Cursors*** mode ( ) to work with cursors on the graph.

- **Double-click** on the graph to set both cursors to the same point. This will also show cursors, if they were not shown.
- **Click** on the graph to move nearest cursor to this point.
- **Click and drag** to select and move cursor.

To place cursors to specific positions and for other options right-click on the graph and select **Cursors** command from context menu. **Cursors** dialog box will show up:



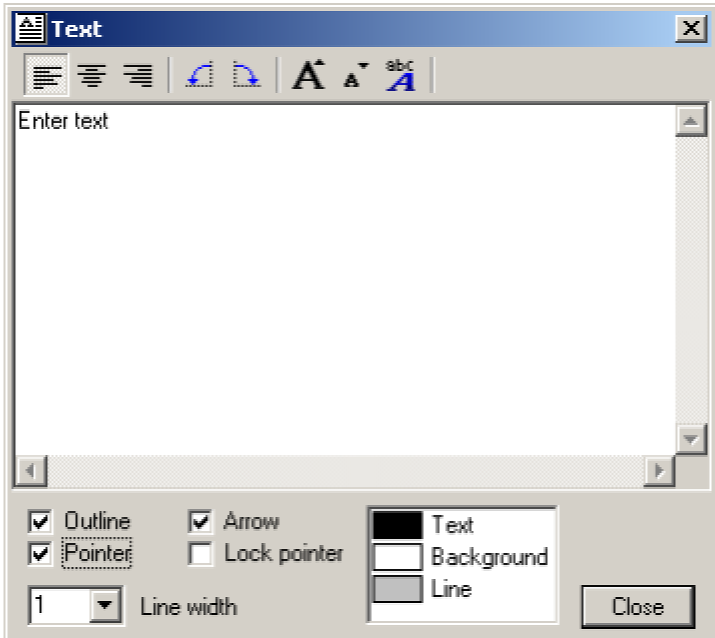
- **Show**. Select checkbox to show cursors
- **Left, Right, Interval**. Enter new cursors position or interval; press **Enter** to apply or **Esc** to cancel. If interval changed, an active cursor will move.
- **Lock position**. Lock cursors at current position, cursors can not be moved.
- **Lock interval**. Keep current interval between cursors. If one cursor is being moved, another one will automatically follow it to maintain specified interval.

The following toolbar buttons can be used to move cursors:

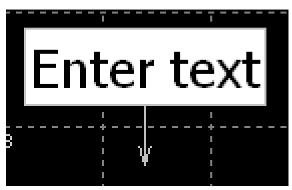
- **Right maximum.** Move selected cursor to the nearest right maximum of selected trace.
- **Left maximum.** Move selected cursor to the nearest left maximum of selected trace.
- **Right minimum.** Move selected cursor to the nearest right minimum of selected trace.
- **Left minimum.** Move selected cursor to the nearest left minimum of selected trace.
- **Right unity gain.** Move selected cursor to the nearest right frequency with unity gain (magnitude=1) of selected trace.
- **Left unity gain.** Move selected cursor to the nearest left frequency with unity gain (magnitude=1) of selected trace.

Text.

To add text on the graph **right-click** on the graph and select **Insert Text** command from the context menu. **Text** dialog box will show up:



Enter text in the text box. The text will be simultaneously shown on the graph:



The text can be formatted using toolbar buttons and controls:

Alignment. Set alignment of multi-line text.

- **Align left.**
- **Center.**
- **Align right**.

Orientation. Change orientation of the text.

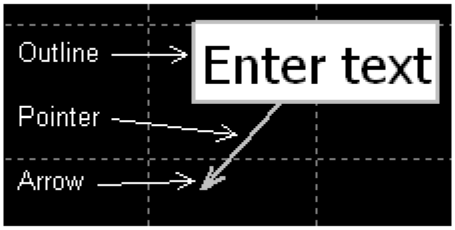
- **Rotate left.**
- **Rotate right.**

Font. Change size of the font or select specific font type and options.

- **Larger font.**

- **Smaller font.**
- **Select font.**

Outline and pointer options



- **Outline.** Draw outline rectangle.
- **Pointer.** Draw pointer line from the text to specified point.
- **Arrow.** Draw pointer line with arrow.
- **Lock pointer.** Lock the end of the pointer: the end of the pointer will not move even when text is being moved.

- **Line width.** Specify line width of the outline and pointer.
- **Color.** Double-click on the item in the list to change the color.

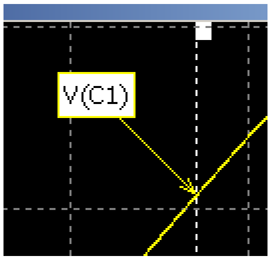
If graph is zoomed or scrolled, the text stays at the same place, anchored to left-top corner of the graph window. To move the text, click on the text and drag. If pointer is locked, only text will move. To move the pointer only, click on the end of the pointer and drag.

To edit the text, **double-click** on the text, or **right-click** on the text and select **Edit text** command from context menu. The same **Text** dialog box will show up.

To delete the text, **right-click** on the text and select **Delete text** command from context menu.

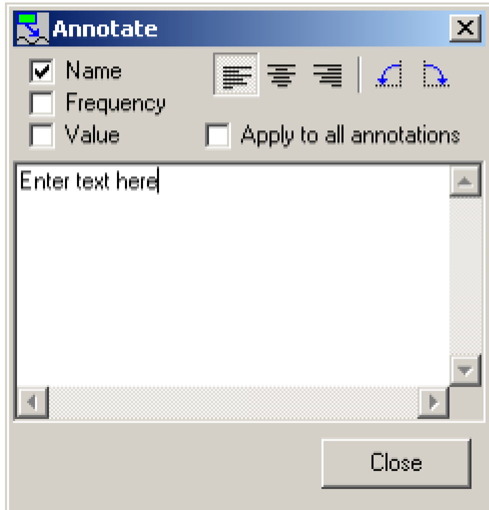
Annotation

Annotation is a text with a pointer, which always points to the same data point of a trace, even when graph is zoomed or scrolled. Annotation belongs to a trace, so if trace is deleted, all trace annotations are deleted as well. Annotation is also deleted if trace data is cleared. For instance, if annotation is added to simulation (**Run**) trace, and a new simulation is started, the annotation will disappear, since the trace data is cleared at simulation start.



To add annotation, set active cursor to the time point where annotation is needed, right-click on the graph, select **Annotate**, then select **Selected trace** or **All traces** command. The same buttons are available in the AC toolbar. Annotation(s) will be added only if trace exists at cursor's frequency. If cursors are disabled, annotation will be added approximately at 1/3 of a screen.

Annotation font, colors, number of significant digits, and some other properties can be specified at the **Annotation** page of **Properties**. To change annotation text and annotation-specific properties, **double-click** on the annotation, or **right-click** on the annotation and select **Edit annotation** command from context menu. **Annotate** dialog box will show up.



Enter text in the text box. The text will be simultaneously shown on the annotation. The following options and formatting are available:

- **Name.** Display trace name in the text.
- **Frequency.** Display frequency of the annotation in the text.
- **Value.** Display trace magnitude and phase (is phase trace is shown) value in the text.

Alignment. Set alignment of multi-line text.

- **Align left.**
- **Center.**
- **Align right**.

Orientation. Change orientation of the text.

- **Rotate left.**
- **Rotate right.**

- **Apply to all annotations.** Select to apply current settings to all annotations on the graph.

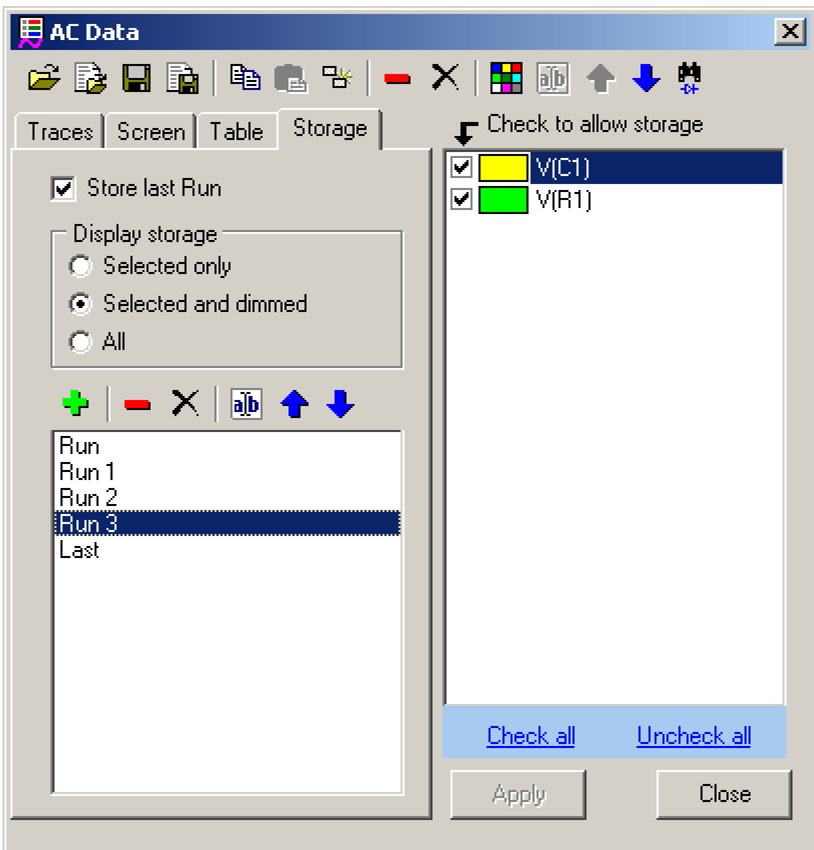
To move annotation text keeping pointer at the same trace point, click on the annotation text and drag. To move the pointer, click on the end of the pointer and drag. The pointer will change frequency, yet following trace amplitude. Annotation text will move with the pointer.

To delete annotation, **right-click** on the annotation and select **Delete annotation** command from context menu. To delete all annotations, **right-click** on the graph, select **Annotate**, then select **Delete all** command from context menu.

Storage

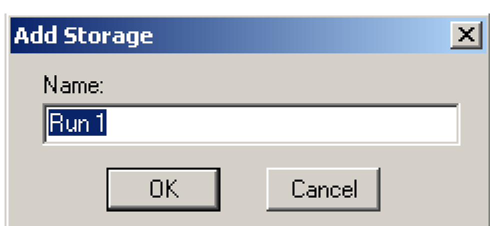
The results of the last simulation run are always shown in the **Run** tab of the **AC Window**. Last run data can be moved into storage under the name **Last**, so that it can be compared with other simulation runs. Each storage data has a tab on the **Data selection** area assigned to it. Storage data can be selected by clicking on the tab. Storage data belongs to the **trace**, so that if trace is deleted, storage data for this trace will be deleted as well.

List of available storage data, commands, and storage display selection can be found on **Storage** page of **AC Data** window:



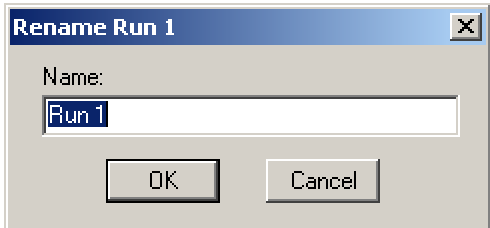
To access some commands **right-click** on the graph or storage tab, then select command from context menu.

- **Move Run to storage.** Move current simulation data into storage. **Add Storage** dialog box will show up:



Enter new storage name or leave suggested default name and click **OK**. A new tab with storage name will be created in the **Data selection** area.

- **Remove** selected storage. Last **Run** data can be removed as well.
- **Clear storage.** Delete all storage data.
- **Rename** selected run. **Rename** dialog box will show up:



Enter new storage name and click **OK**.

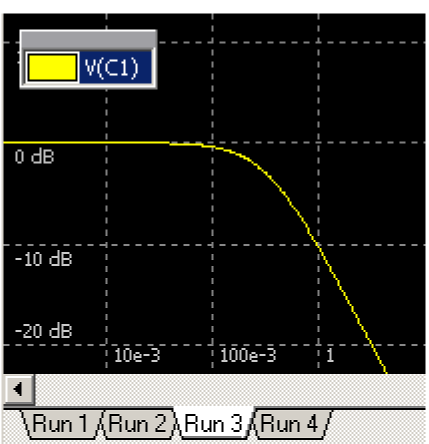
- **Move selected up.**
- **Move selected down.**

- **Store last Run.** Select this option to compare new simulation with previous one. When new simulation starts, current simulation data will be moved into storage under the name "Last". If run with the name "Last" already exists, it will be overwritten. Thus, previous simulation data will always be in the storage under the name "Last".

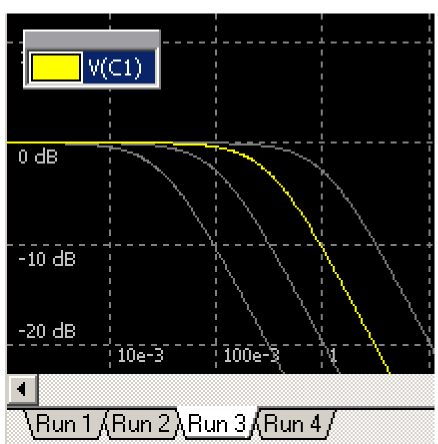
Display storage

- **Selected only.** Only selected data is shown on the graph.
- **Selected and dimmed.** Selected data is displayed with normal trace colors, other data is displayed with dimmed color, specified on **Graphs** page of **Preferences** dialog box.
- **All.** All data is displayed with normal trace colors.

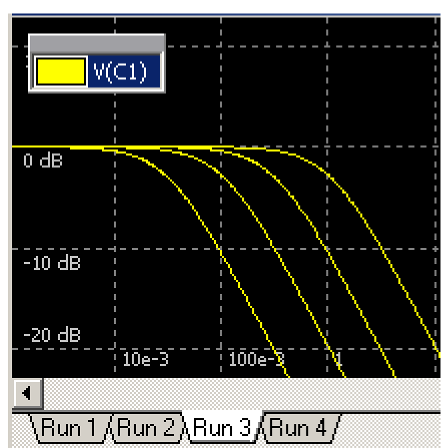
Example:



Selected only



Selected and dimmed

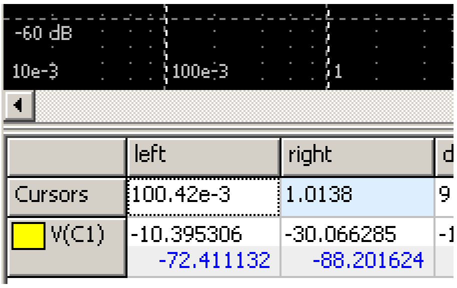


All

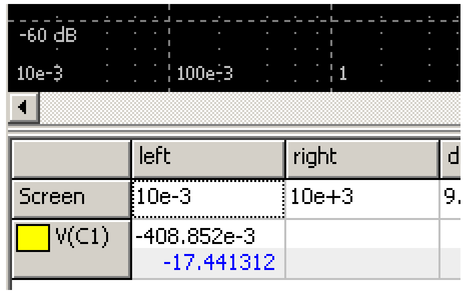
When **Storage** page is selected, checkboxes in the trace list specify traces with storage allowed.

Data table

The Data table shows cursors position, trace values, and some characteristics of the traces calculated between cursors, such as: mean, max, min, and more. If cursors are disabled, the table shows the data at the left and right edges of the screen, and values calculated between left and right edges of the screen:



Cursors enabled, active cursor is highlighted



Cursors disabled, screen is used

- To show/hide Data table click **Table** toolbar button, or **right-click** on the graph and select **Table** command from context menu.
- **Click** on the trace row to select the trace. Selected trace will be shown on top of all traces.
- **Double-click** on the table to open **AC Data** window.
- **Right-click** to open context menu. The menu will contain some common commands, and commands related to selected trace.
- Colors, fonts, and number of significant digits used in the table can be customized on **Table** page of **Preferences** dialog box**.** Phase and magnitude have separate font and color settings.

The table can be displayed on the bottom of the **AC Window**, or as a separate window: **right-click** on the table and select **Separate window** command:

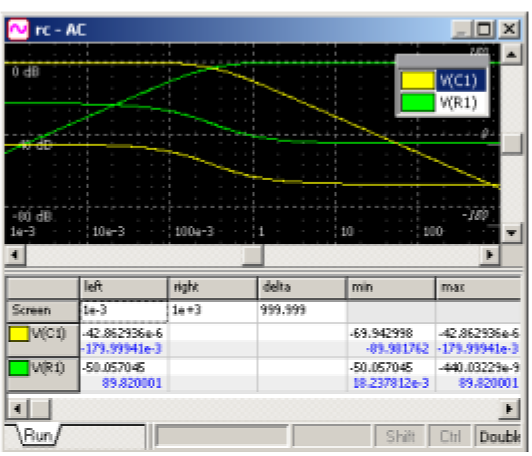


Table in the AC Window

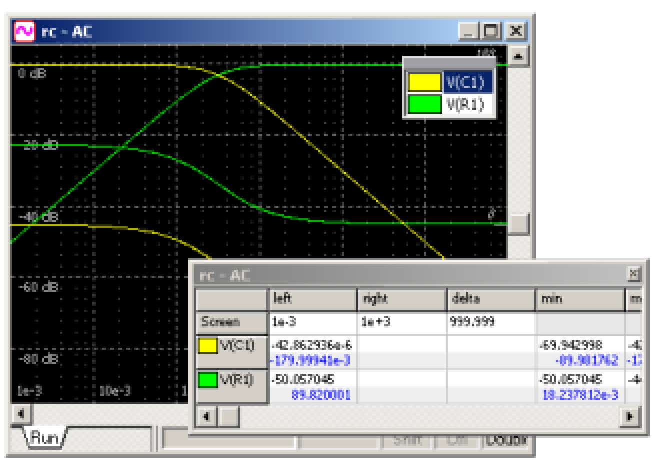
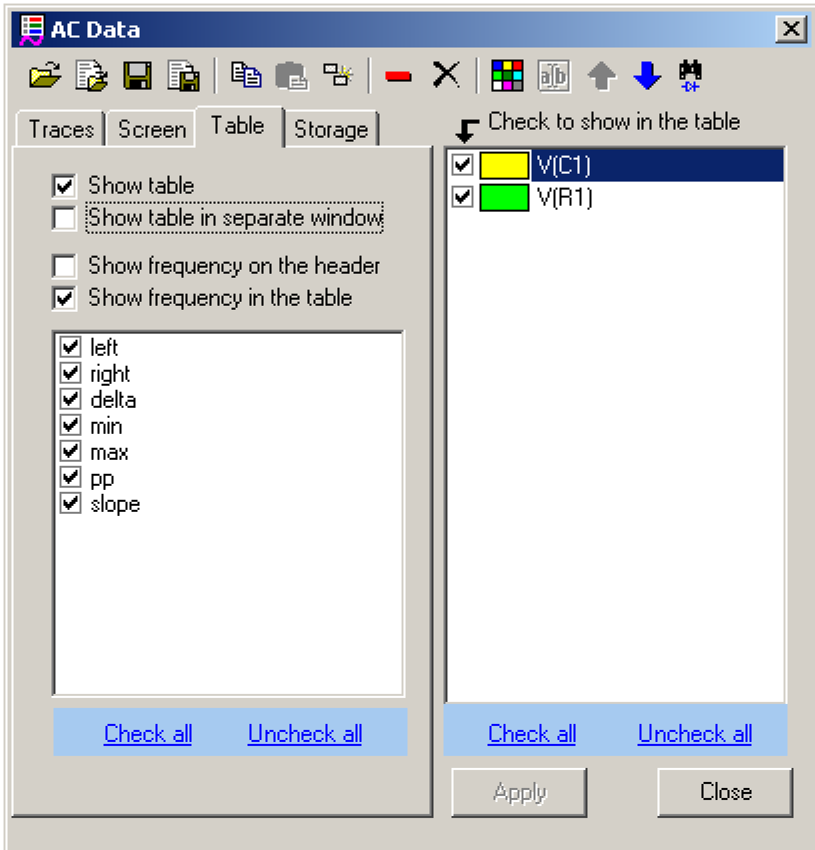
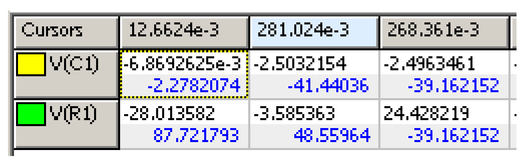


Table in separate window

The values shown in the table, as well as other table options, can be selected on **Table** page of **AC Data** window:

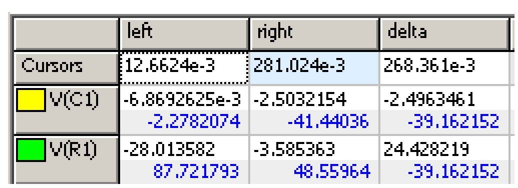


- **Show table.** Select to show table.
- **Show table in separate window.** If selected, the table will be displayed as a separate window.
- **Show frequency on the header.** If selected, cursors position will be shown in the header line, in the **left**, **right**, and **delta** columns.



Cursors	12.6624e-3	281.024e-3	268.361e-3
V(C1)	-6.8692625e-3 -2.2782074	-2.5032154 -41.44036	-2.4963461 -39.162152
V(R1)	-28.013582 87.721793	-3.585363 48.55964	24.428219 -39.162152

- **Show frequency in the table.** If selected, cursors positions will be shown in a separate row.



	left	right	delta
Cursors	12.6624e-3	281.024e-3	268.361e-3
V(C1)	-6.8692625e-3 -2.2782074	-2.5032154 -41.44036	-2.4963461 -39.162152
V(R1)	-28.013582 87.721793	-3.585363 48.55964	24.428219 -39.162152

- **Table values.** Select values to display in the table:
  - **left** – trace value at left cursor.
  - **right** – trace value at right cursor.
  - **delta – right** minus **left.**
  - **min** – trace minimum between cursors.

- **max** – trace maximum between cursors.
- **pp** – trace peak-to-peak value between cursors.
- **slope** – magnitude slope between cursors, in dB/decade. If cursors are located at the same frequency, slope is calculated as magnitude derivative at this frequency. Otherwise slope is calculated as $(\mathbf{Mag}_{\mathbf{right}} - \mathbf{Mag}_{\mathbf{left}}) / (\mathbf{f}_{\mathbf{right}} - \mathbf{f}_{\mathbf{left}})$.

When **Table** page is selected, checkboxes in the trace list specify traces shown in the table.

Scrolling and Zooming

To scroll graph use any of the following methods:

- Move mouse pointer to the left or right edge of the graph. Mouse pointer will take "big arrow" shape. Click or hold left mouse button to scroll graph.
- ***Cursors*** mode : hold **Shift** key, then click and drag graph.
- ***Scrolling*** mode : click and drag graph.
- Hold **Ctrl** key and rotate **mouse wheel** to scroll horizontally.
- Hold **Shift** key and rotate **mouse wheel** to scroll vertically.
- Press **Right** and **Left** keys.
- ***Zoom*** mode : **double-click** to center this point.

To zoom graph use any of the following methods:

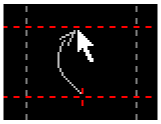
- Rotate **mouse wheel** to zoom horizontally.
- Hold **Ctrl** and **Shift** key and rotate **mouse wheel** to zoom vertically.
- Click toolbar buttons, or use keyboard shortcuts, or **right-click** on the graph, select **Zoom**, then select command:
  - **Horizontal Zoom-in (Ctrl-PgUp).**
  - **Horizontal Zoom-out (Ctrl-PgDn).**
  - **Fit the screen horizontal (Ctrl-Home).**
  - **Fit cursors to screen.**
  - **Vertical Zoom-in (PgUp).**
  - **Vertical Zoom-out (PgDn).**
  - **Fit the screen vertical (Home).**
  - **Fit the screen (Shift-Home).**
  - **Zoom-in (Shift-PgUp).**
  - **Zoom-out (Shift-PgDn).**

To zoom selected area

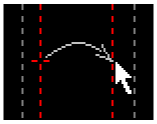
- ***Zoom*** mode : click and drag to select area.
- ***Cursors*** mode : hold **Ctrl** key, then click and drag to select area.

Selection area depends on how mouse pointer is moving relative to starting point.

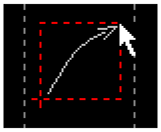
- If mouse pointer is moving only up or down, two horizontal lines will be shown. When left button released, selected area will be zoomed-in vertically.



- If mouse pointer is moving only left or right, two vertical lines will be shown. When left button released, selected area will be zoomed-in horizontally.



- If mouse pointer is moving diagonally, rectangle will be shown. When left button released, selected rectangle area will be zoomed-in to fit the screen.



In the phase graph area only horizontal scroll and zoom can be performed.

To Undo and Redo scrolling and zooming click toolbar buttons:

- **Undo** scrolling or zooming.
- **Redo** scrolling or zooming.

AC commands

The following commands, buttons, and shortcuts are available in the Main menu, Main Toolbar, AC toolbar, and AC context menus.

- **Open/Show AC window (F8)**.
- **AC Settings.** Show AC Settings dialog box.
- **AC Data.** Open AC Data window.
- **Start AC (F9).** Start AC analysis.
- **Stop.** Stop AC analysis.
- **Log.** Show AC log.
- **Sweep**.
- **Preferences.** Open **Preferences** dialog box.

Toolbar and some context menus:

- **Cursors** mode.
- **Zoom** mode.
- **Scrolling** mode.
- **Horizontal Zoom-in (Ctrl-PgUp).**
- **Horizontal Zoom-out (Ctrl-PgDn).**
- **Fit the screen horizontal (Ctrl-Home).**
- **Fit cursors to screen.**
- **Vertical Zoom-in (PgUp).**
- **Vertical Zoom-out (PgDn).**
- **Fit the screen vertical (Home).**
- **Fit the screen (Shift-Home).**
- **Undo scale (Backspace).** Undo scale.
- **Redo scale.**
- Show/hide **Cursors**.
- Show/hide **Data Table**.
- Show/hide **Legend**.
- **Right maximum.** Move selected cursor to the nearest right maximum of selected trace.
- **Left maximum.** Move selected cursor to the nearest left maximum of selected trace.
- **Right minimum.** Move selected cursor to the nearest right minimum of selected trace.
- **Left minimum.** Move selected cursor to the nearest left minimum of selected trace.

- **Right unity gain.** Move selected cursor to the nearest right frequency with unity gain (magnitude=1) of selected trace.
- **Left unity gain.** Move selected cursor to the nearest left frequency with unity gain (magnitude=1) of selected trace.

Graph commands (context menu):

- Open **Cursors** dialog box.
- **Phase (Tab)** ►
  - **Off**. Do not show phase.
  - **On**. Show magnitude and phase in the same area of the graph.
  - **Separate**. Show magnitude and phase in separate areas of the graph.
- **Traces** ► (Commands apply to all traces displayed on the graph)
  - **Open.** Load traces from "nlf" data file
  - **Import** traces from text or "csv" file.
  - **Save** traces into "nlf" data file.
  - **View/Export.** View traces as a text and save into text or "csv" file.
  - **Copy** traces to clipboard.
  - **Paste** traces from clipboard.
- **Image** ►
  - **Copy to clipboard.** Copy image of AC window to clipboard.
  - BMP **Save as BMP.** Save image of AC window to file in BMP format.
  - JPG **Save as JPG.** Save image of AC window to file in JPG format.

Storage commands:

- **Move Run to storage**.
- **Remove** selected storage.
- **Clear storage.**
- **Rename** selected storage.
- **Store last Run.** Move current Run into storage "Last" when new simulation starts.
- **Selected only**
- **Selected and dimmed**
- **All**

} Storage display mode

Annotation commands:

- **Annotate selected trace.**
- **Annotate all traces.**
- **Edit annotation.**

- **Delete annotation.**
- **Delete all.**

Text commands:

- **Insert text** on the graph.
- **Edit text.**
- **Delete text.**

Data table commands:

- **Hide *trace name*:** do not show trace in the Data table.
- **Separate window.** Show Data table in the AC Window or as a separate window.

Legend commands:

- **Hide *trace name*:** do not show trace on the graph.
- **Remove** selected trace.
- **Rename** selected trace.
- **Duplicate** selected trace.
- **Copy** selected trace to clipboard.
- **Paste** traces from clipboard.
- **Find component:** V, I, Z, Gamma, and VSWR traces only. Click to show the component on the schematic.

Keyboard keys and shortcuts

The following keyboard keys and shortcuts can also be used:

- **Tab.** Toggle phase display mode.
- **Left, Right.** Scroll graph.
- **Up, Down.** Select trace.
- **Shift-PgUp.** Zoom-in.
- **Shift-PgDn.** Zoom-out.

Mouse operation

The following mouse operation can be used.

- **Right-click.** Open context menu.
- **Mouse-wheel.** Horizontal zoom-in/zoom-out.
- **Ctrl-mouse wheel**. Scroll horizontally.

- **Shift-mouse wheel**. Scroll vertically.
- **Ctrl-Shift-mouse-wheel.** Vertical zoom-in/zoom-out.

Cursors mode :

- **Click (left button).** If cursors visible, move nearest cursor.
- **Click and drag.**
  - On annotation: move annotation text or pointer.
  - On text: move text or pointer.
  - Otherwise: move cursor.
- **Double-click.**
  - On annotation: edit annotation.
  - On text: edit text.
  - Otherwise: show cursors, move both cursors.

Zoom mode :

- **Click and drag.** Select and zoom.
- **Double-click.** Center screen.

Scrolling mode :

- **Click and drag.** Scroll graph.
- **Double-click.** Center screen.

AC Tools

AC Tools offer different ways of presenting simulation results. To open Tool go to **AC | Tools**, then select the line with required Tool.

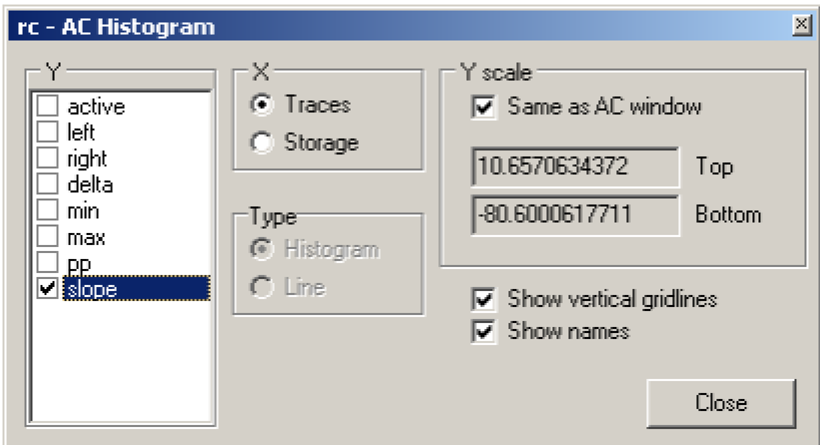
The following Tools are currently available:

- Histogram
- Smith chart
- Nyquist plot
- Nichols plot
- Markers

Histogram

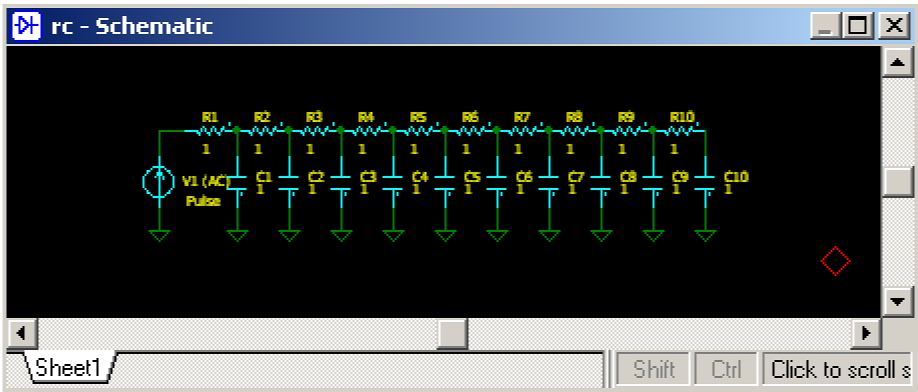
Histogram presents trace values and some characteristics of the traces calculated between cursors (or on the screen, if cursors are disabled), in a graphical format. Histogram can also show "cross-section" of traces or storage data.

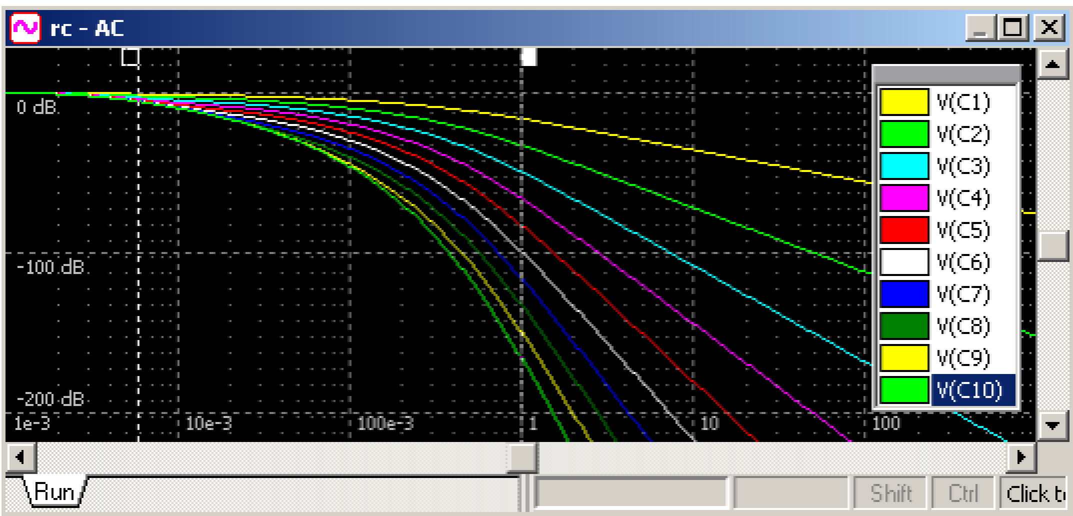
- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:

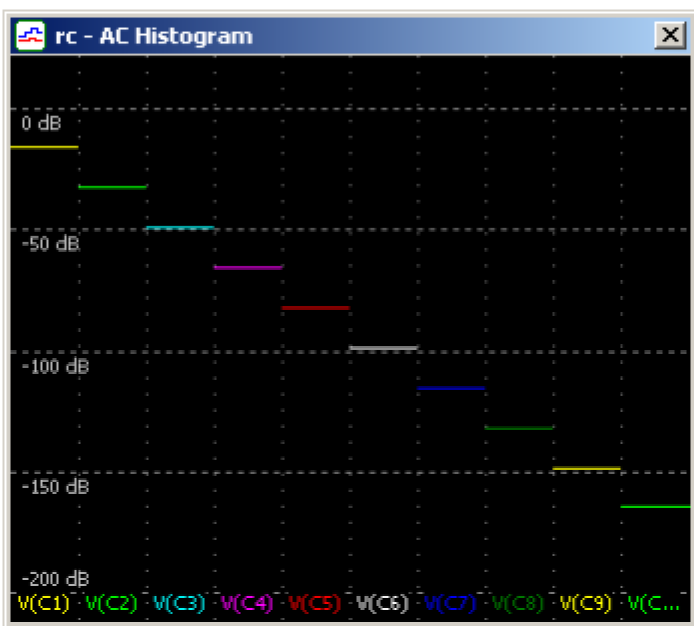


  - **Y** values. Select variables to be displayed on Y axis. "**Active"** is currently selected cursor (left or right). Other values are similar to what is displayed in the AC Data Table.
  - **X**. Select histogram mode: what is shown on X axis.
    - **Traces**. Show "cross-section" of all traces currently displayed on the graph.
    - **Storage**. Show "cross-section" of the storage for all traces currently displayed on the graph.
  - **Type**. Select histogram type for **Storage** mode:
    - **Histogram.**
    - **Line**.
  - **Y scale**. By default, histogram Y scale is the same as AC screen vertical scale. Uncheck **Same as AC window** checkbox to enter Y scale **Top** and **Bottom.**
  - **Show vertical gridlines.** Check to show vertical gridlines dividing histogram data.
  - **Show names.** Check to show trace or storage names on the X axis.

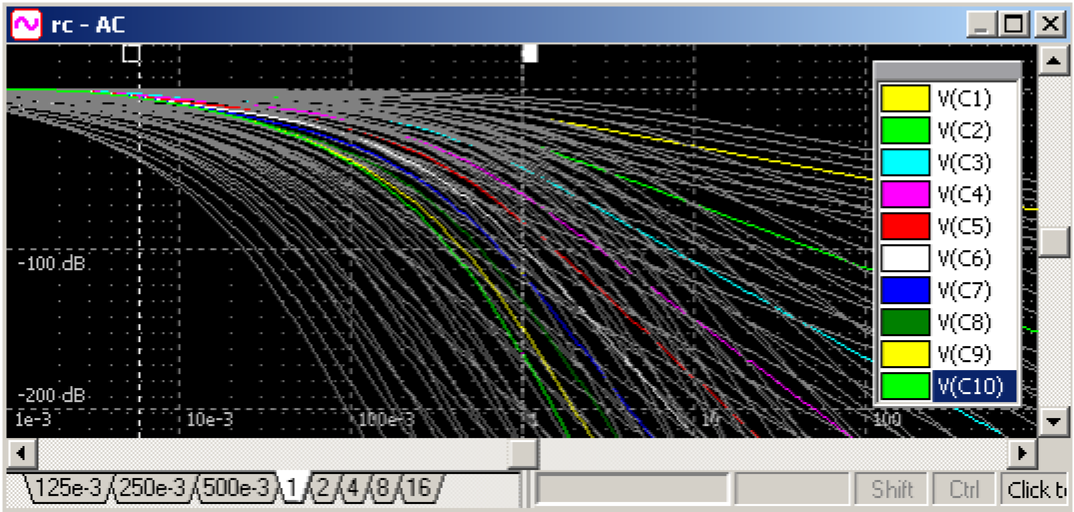
Traces mode, or "traces cross-section", can be used to display "spatial" distribution of the signal in the schematic. The following histogram shows attenuation on each stage of RC filter at 1 Hz.

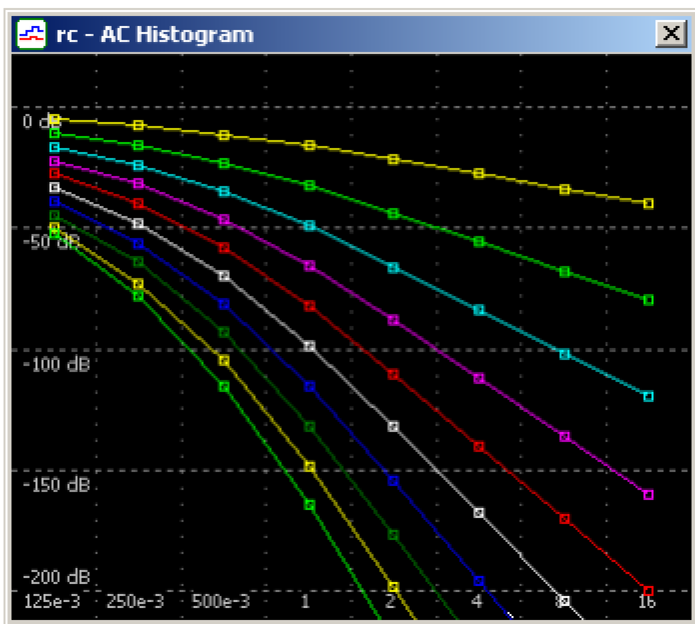






Storage mode, or "storage cross-section", can be used to display how trace values at specific time depend on schematic parameters. The following example shows modeling of previous schematic, with resistors changed from 0.125 to 16, with X2 step using AC sweep. Each run is saved into storage. X axis of histogram is storage data (i.e. resistance). The lines of different colors show attenuation on each stage of RC filter at f=1 Hz (active cursor), as a function of resistance.

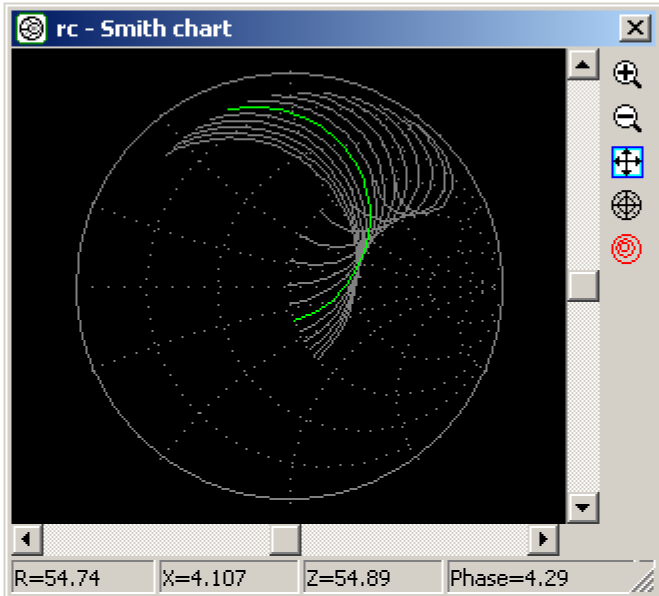




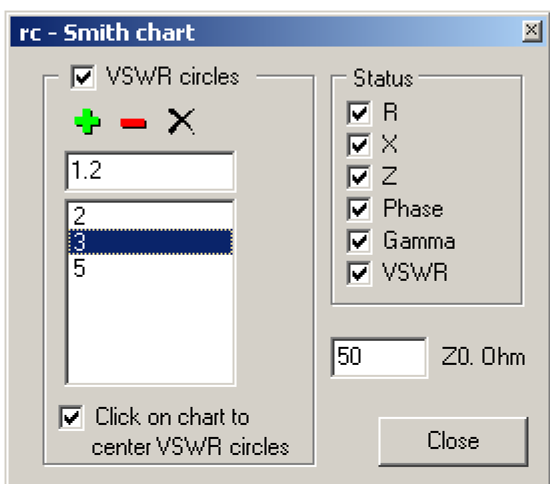
One should notice that "cross-section" of Storage histogram at R=1 is the same as upper line of Trace histogram shown in the previous example.

Smith Chart

This is a standard Smith Chart, plotting complex reflection coefficient. Please note that Smith chart is supposed to plot Z (impedance) traces only. In fact it plots all types of AC traces, interpreting them as complex impedance. The chart shows traces between cursors only (or on the screen, if cursors are disabled).

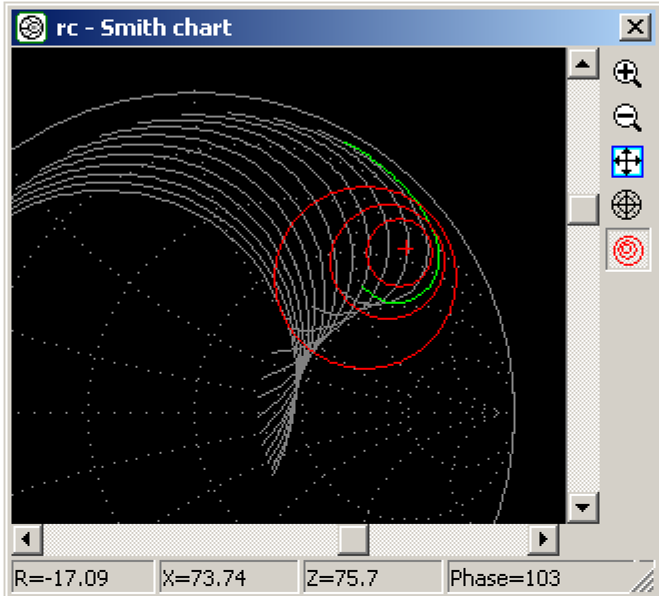


- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:



  - **VCWR circles**. Show circles with specified VSWR values around selected point. Values are shown in the VSWR list.
    - **Add** VSWR circle to the list.
    - **Remove** VSWR circle from the list.
    - **Delete** all.

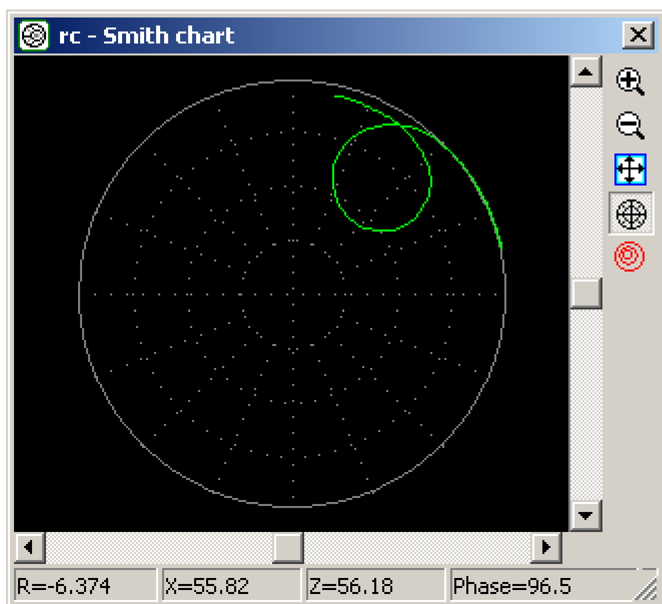
- **Click on chart to center VSWR circles**. If checked, VSWR circles will be shown around arbitrary selected point on the chart:



If unchecked, VSWR circles will be shown around canter of the chart.

- **Status.** Show selected values in the status bar. The values are shown for mouse pointer position.
- **Z0, Ohm.** Characteristic impedance of the chart.

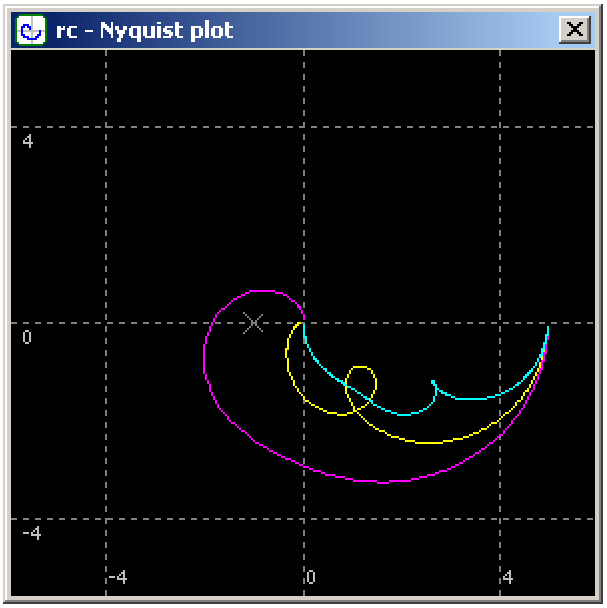
- **Zoom in.**
- **Zoom out.**
- **Fit the screen.**
- **Polar grid.**
- Show/hide **VSWR circles.**



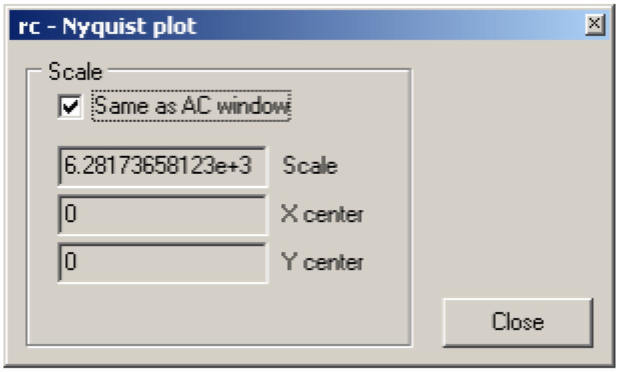
Polar grid

Nyquist plot

Nyquist plot shows complex AC response in polar coordinates. The diagram shows traces between cursors only (or on the screen, if cursors are disabled). "X" marker shows unit gain point with –180 degrees phase.



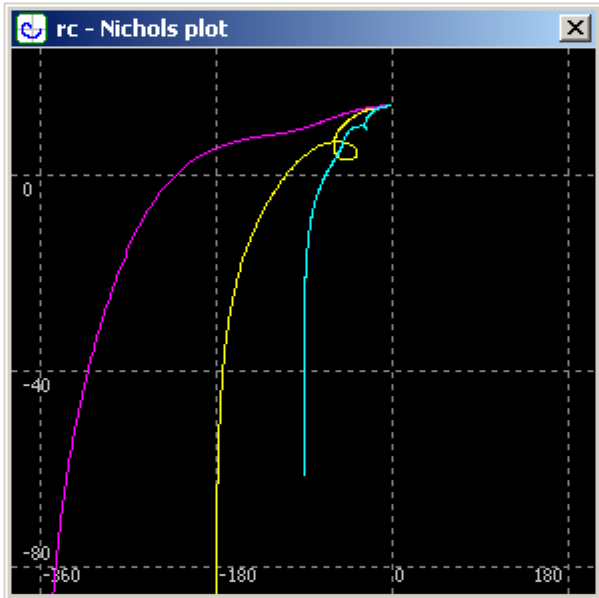
- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:



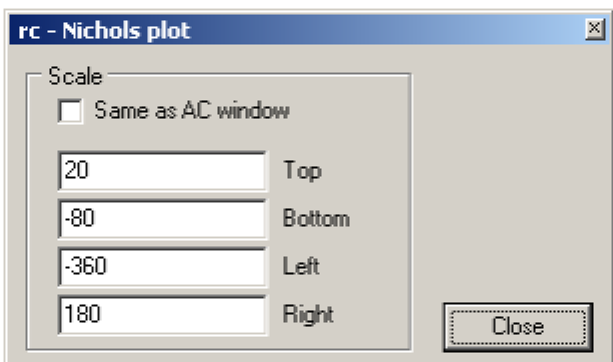
By default, plot **Scale**, **X center** and **Y center** are obtained from AC graph vertical scale. Uncheck **Same as AC window** checkbox and enter scale and individual X and Y center values.

Nichols plot

Nichols plot shows logarithm of the magnitude as a function of the phase. The diagram shows traces between cursors only (or on the screen, if cursors are disabled).



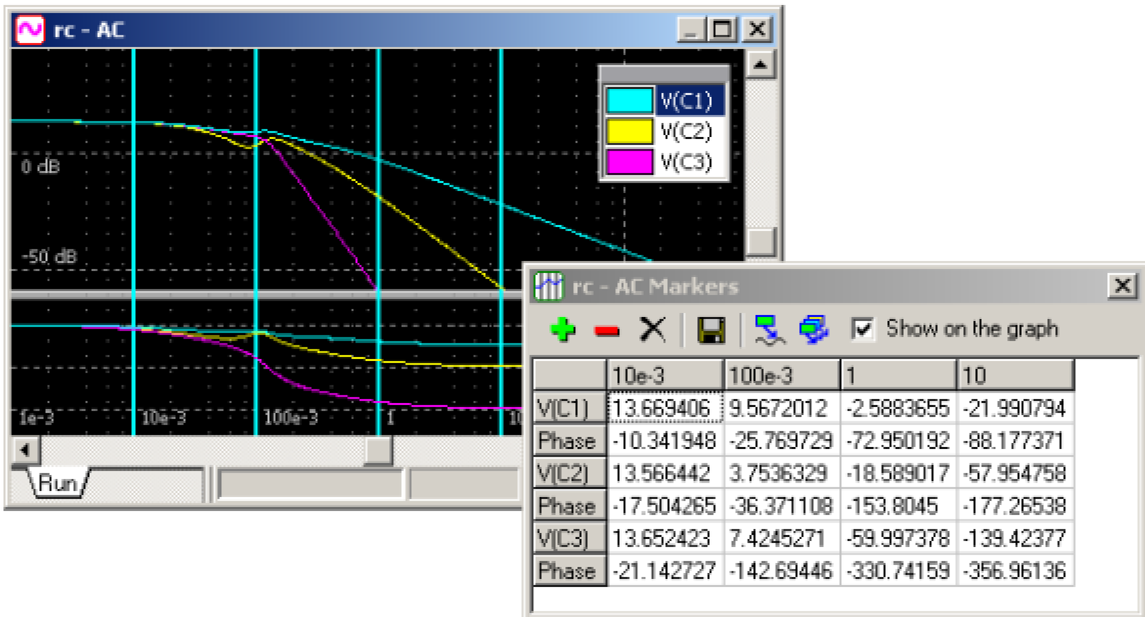
- **Right-click** on the window to access relevant commands.
- **Double-click** on the window to open Configuration dialog box:



By default, plot **Top, Bottom, Left,** and **Right** are the same as AC graph magnitude and phase scales. Uncheck **Same as AC window** checkbox and enter new scales if needed.

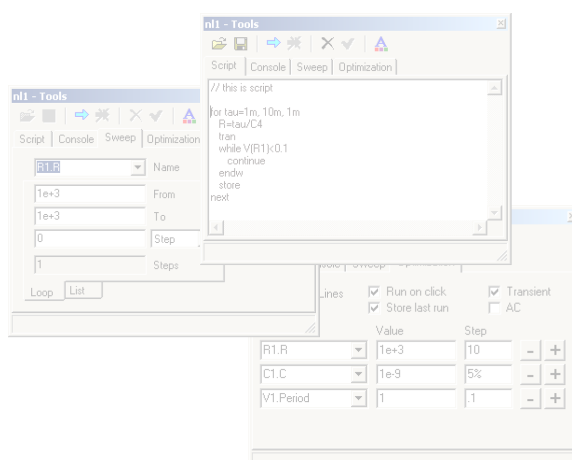
Markers

Markers tool provides a convenient way to monitor traces amplitudes at specified points. Unlike cursors, markers always stay at specified position. Number of markers is not limited. Below you can see 4 markers shown on the AC graph (at 0.01Hz, 0.1Hz, 1Hz, and 10Hz) with traces magnitude and phase displayed in the Markers table.



- **Markers table** consists of the traces displayed on the graph.
- **Add** new marker. Marker frequency is a frequency of selected cursor.
- **Remove** marker from the table. Select (click) any cell in the table, which belongs to the marker's column, then click the button. On the example above, the first marker (f=10e-3) will be removed.
- **Delete** all markers.
- **Export** markers table in the text or "csv" file.
- **Annotate** selected trace at markers positions on the AC graph.
- **Annotate all traces** at markers positions on the AC graph.
- **Show on the graph**. Select to show markers on the AC graph. Markers width and color can be changed on the **Graphs** page of **Preferences** dialog box.

VI. Tools


nl1 - Tools
Script
Console
Sweep
Optimization
// this is script
for tau=1m, 10m, 1m
R=tau/C4
tran
while V(R1)<0.1
continue
endw
store
next
Name
From
To
Step
Steps
Loop
List
Run on click
Store last run
Transient
AC
Value
Step
R1.R
C1.C
V1.Period

Tools are used to perform and automate complex analysis by means of **script**. Most of the Tools are located in the **Tools** window. Use the following Main Menu commands (**Tools** menu) or Toolbar buttons to open Tools:

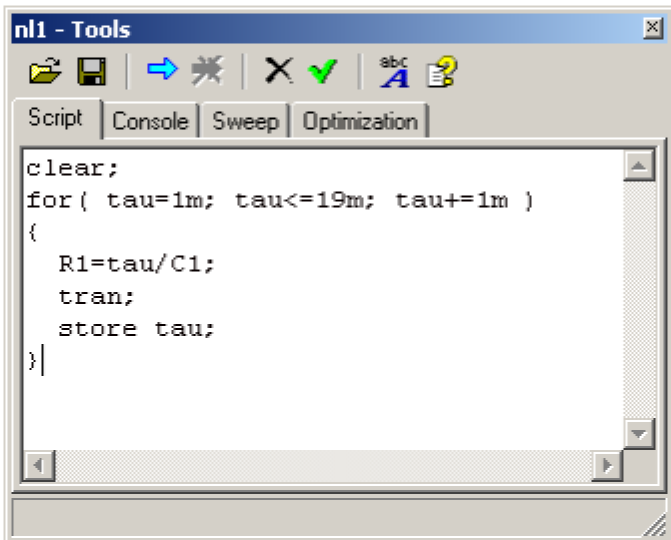
- **Script.** Open, save, edit, and run script.
- **Console.** Shows script execution log. Also has a command line, which works as calculator, and allows executing some script commands.
- **Sweep.** Performs multiple transient and/or AC analysis runs while sweeping component parameter.
- **Optimization.** Performs transient and/or AC analysis while manually iterating selected parameters.
- **HTTP Link.** Configure and start NL5 HTTP server to provide link with external applications.

Script

Script page allows opening, editing, saving, and running of **script.** Script is a program written on simplified C language. In addition to standard operators and mathematical functions, it may use NL5-specific **script commands**, which allows changing schematic parameters, running simulation, analyzing data, and saving simulation results.

Script commands can also be executed from command line in the **Console** page of Tools window, and through **HTTP link** from external applications. This allows using NL5 as an "add-on" simulation engine with popular engineering tools such as MATLAB®, Python, and others.

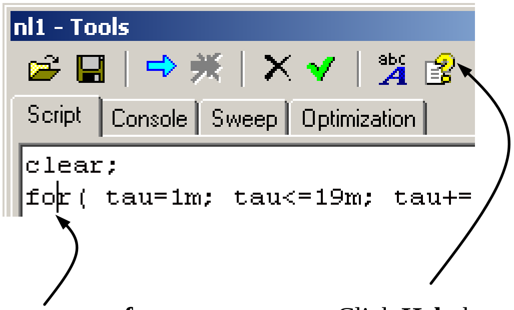
Click **Script** Toolbar button or select **Tools | Script** command in the Main Menu to open **Script** page of **Tools** window:

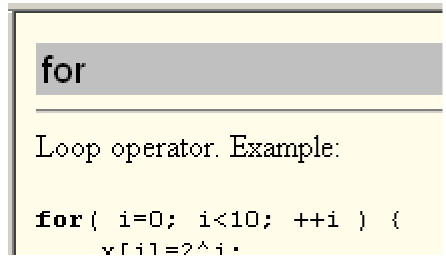


Toolbar buttons perform the following operations:

- **Open script** from the text file.
- **Save script** to the text file.
- **Run script**.
- **Stop script** execution.
- **Clear** text.
- Check script code
- **Font.** Select font of the text.

- **Help (F1)**. Click to open Help. To see Help topic on specific operator, function, or command, place cursor on the word in the script text and click the button, or press **F1**.





Place cursor on **for** operator

Click **Help** button or press **F1**

Help on **for** operator is displayed

Script syntax

Script is using simplified C language (see **Attachment 4** for reference and examples). All operators (**Attachment 2**) and functions (**Attachment 3**) can be used in the script. In addition, NL5-specific script commands (**Attachment 5**) can be used to run simulation, analyze data, and save simulation results.

Component parameters and schematic variables can be used in expressions and can be modified similar to script local variables:

```
R1=tau/C1;
freq=1./V1.period;          // "freq" is schematic variable
```

Trace and cursors data can be used in expressions. The data correspond to the values displayed in the Transient or AC Data table. In order to use trace data, the trace should be added into the Transient or AC data, but does not need to be displayed on the graph or in the data table. For example:

```
x = V(out).max - V(out).min;
if( V(C1).(3.45)>threshold ) break;
charge=I(C1).mean * delta;
```

Script commands. Script commands are used to open schematic, control simulation process (run and continue simulation), and save simulation data in different formats. See **Attachment 5** for commands list and examples. Please note that unlike standard C function calls, script commands parameters do not need to be enclosed into parentheses. However, for consistency of the code, parentheses can be used as well. For example:

```
open(rc.nl5);
tran(0,1,1m);
close();
```

Running script

To run the script use any of the following methods.

- Running script from **Tools** window:
  - Select **Script** page.
  - Enter script code, or click **Open** button , select and open script file.
  - Click **Run script** button to run script.

- Running from Windows environment:
  - Drag and drop script file icon onto NL5 icon.

- Running from command line or another application:
  - Start NL5 with the name of script file as a parameter. For example:

```
nl5.exe myscript.txt
```

When script is running, no changes can be done on the schematic, transient, or AC windows. Only **Stop script** button is available. Script log and error messages are displayed on the **Console** page. Script is applied to the active document.

When transient simulation command **tran** (or **cont**) is issued by the script, transient mode is automatically set to "non-continuous", so that transient will always pause at the end of the screen. The next script command will not be executed until transient simulation is complete.

When AC simulation command **AC** is issued by the script, the next script command will not be executed until AC simulation is complete.

Script examples

Set component parameters. Component parameters have been calculated in external application (for instance, Excel), or entered manually and saved into the text file in the ***name***=***value*** format:

```
R1=5.1;
C1=12e-9;
V3.period=0.01
```

Run the script to apply new parameters to components.

Sweep parameter. Component parameter is changing in specified range, transient analysis performed for each parameter, results placed into storage:

```
for( R1=1; R1<=10; R1+=1 )
{
  tran;
  store R1;
}
```

Sweep parameter from the list. Component parameter is assigned value from the list, AC analysis performed for each parameter, results placed into storage:

```
for( V1.period=1m,2m,10m,50,100m )
{
  ac;
  store V1.period;
}
```

Sweep variable. Local variable is changing in some range, component parameters modified, transient analysis performed, results placed into storage:

```
for( freq=1; freq<=10; freq*=1.1 )
{
  V2.period=1/freq;
  R2=1/(freq*C5);
  tran;
  store freq;
}
```

Wait for condition. Transient is running until peak-to-peak value of the trace is less than specified threshold. When done, Initial Conditions are saved.

```
threshold=1e-6;
tran;
while( v(c1).pp>threshold ) cont;
saveic;
```

Perform analysis for specified file, save data, exit application. Schematic file is loaded into NL5, component parameters changed, transient analysis performed, traces exported into "csv" file, NL5 closed. This script can be executed from command line.

```
open lcr.nl5;
R1=100;
C1=1n5;
tran;
export data.csv;
exit;
```

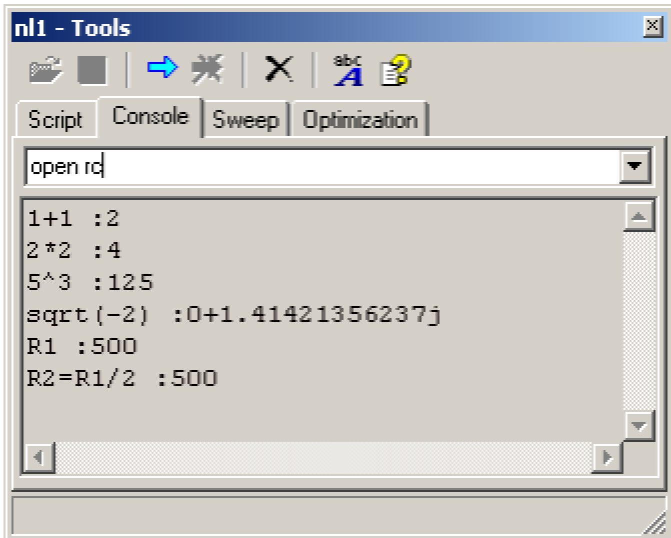
Perform analysis for specified file, log data, exit application. Schematic file is loaded into NL5, component parameter swept, transient analysis performed, traces data logged into text file, NL5 closed. This script can be executed from command line.

```
open lcr.nl5;
logdata lcrdata.csv, r1, v(r1).mean, v(r1).rms;
for( R1=100; R1<=1000; R1+=100 )
{
  tran;
  logdata;
}
exit;
```

Console

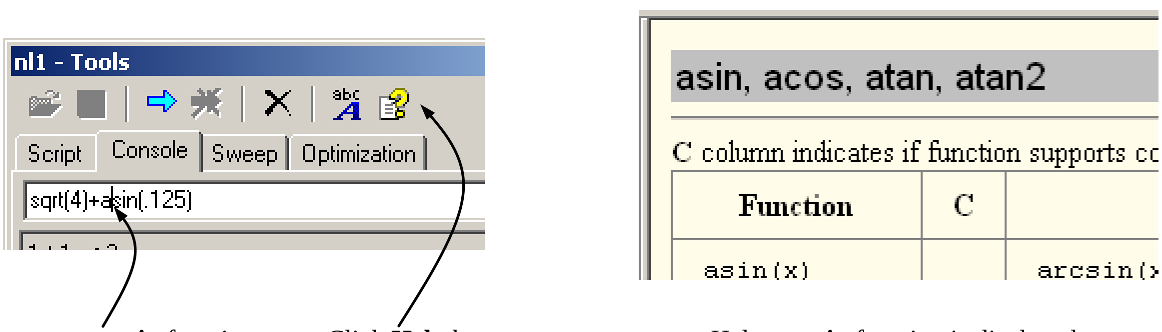
Console page displays log of script execution. It also has a command line, which works as calculator and allows executing some script commands.

Click **Console** Toolbar button or select **Tools | Script** command in the Main Menu to open **Console** page of **Tools** window.



Toolbar buttons perform the following operations:

- **Clear** log.
- **Font.** Select font of the text.
- **Help (F1).** Click to open Help. To see Help topic on specific function or command, place cursor on the word in the command line or in the log window and click the button, or press **F1**.



Place cursor on **asin** function Click **Help** button or press **F1** Help on **asin** function is displayed

Command line

Command line is used to evaluate expressions. Expression may contain:

- Numbers (including complex numbers).
- Component parameters and schematic variables of active document.
- Arithmetic operators and functions.
- Trace data.

Enter expression in the command line and press **Enter** to evaluate. Results will be displayed in the log area. For example:

```
2*2                  : 2*2=4
sin(45)              : sin(45)=707.106781187e-3
R1*C1                : R1*C1=15
sqrt(-2)             : sqrt(-2)=0+1.41421356237j
V(R1).mean           : V(R1).mean=0.15425
```

To change component parameters or variables of active document, enter parameter name followed by equal sign and expression:

```
R1=1k
R1=1000/C1
V1.model=pulse
```

You can also execute script commands from command line: all script commands except **logdata** and **exit** are allowed. In addition, **pause** command will pause transient simulation. Commands are applied to active document. For example:

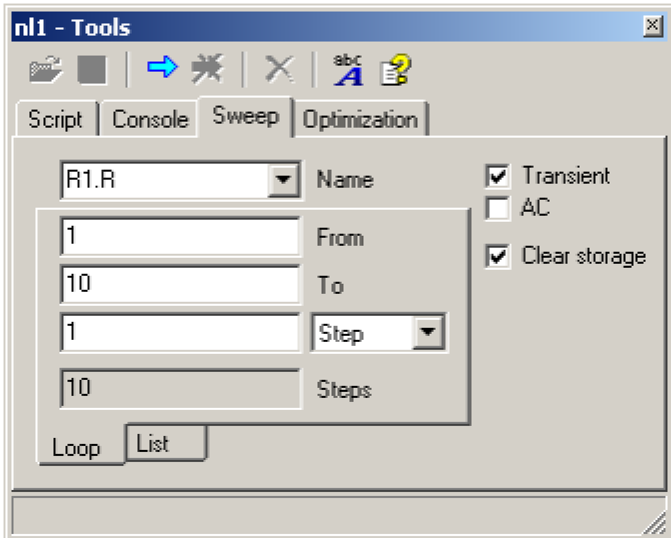
```
open rc.nl5
tran 0,1m,.1u
store
cont
```

Previous expressions and commands can be retrieved from drop-down list, or by pressing **Up** and **Down** arrow keys. Press **Esc** to clear command line.

Sweep

Sweep page provides automatic generation of and running of script code, which changes component parameter in specified range, performs transient and/or AC analysis, and stores traces into storage.

Click **Sweep** Toolbar button  or select **Tools | Sweep** command in the Main Menu to open **Sweep** page of **Tools** window:



Toolbar buttons perform the following operations:

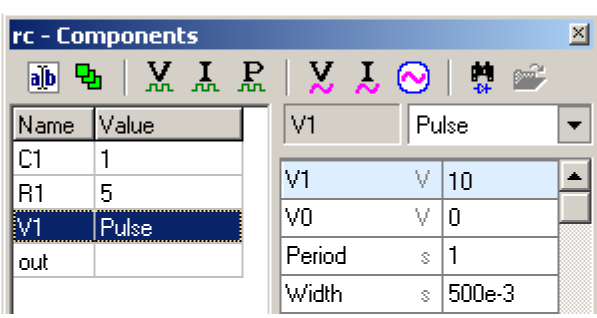

- **Run** sweep.


- **Stop** sweep execution.

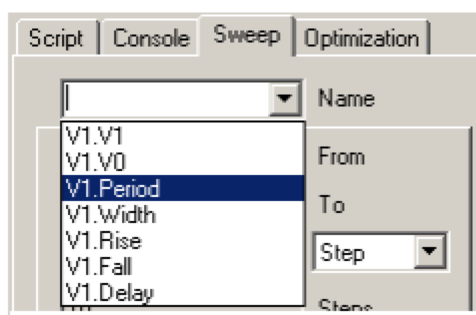

- **Clear** List type sweep text.

There are two types of sweep: **Loop** and **List**. Set up the following parameters for both sweep types:

- **Name**. Name of the parameter to be changed. Enter parameter name manually, or select from drop-down list. The list shows all numerical parameters of the component, which is currently selected in the schematic or in the Components Window. For example:



Select component V1



Select parameter V1.Period

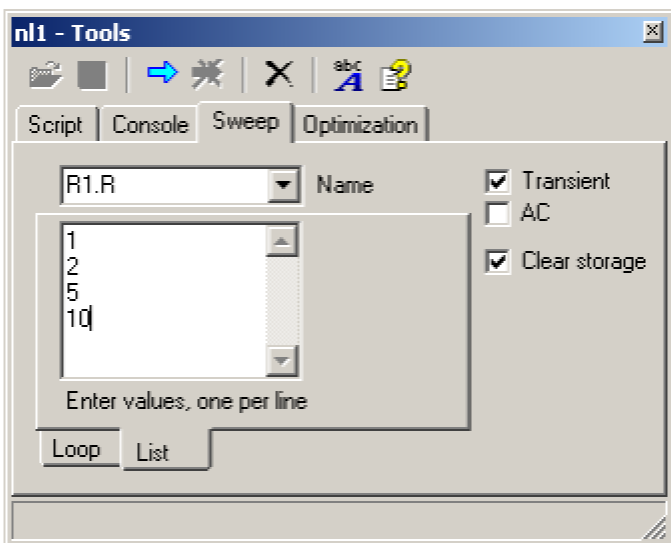
- **Transient**. Select to run transient.
- **AC**. Select to run AC analysis.
- **Clear storage**. Select to clear storage when sweep starts. If nor cleared, sweep runs will be added to existing storage.
- Select **Loop** or **List** sweep type on the bottom tab.

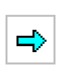
For **Loop** sweep enter the following parameters:

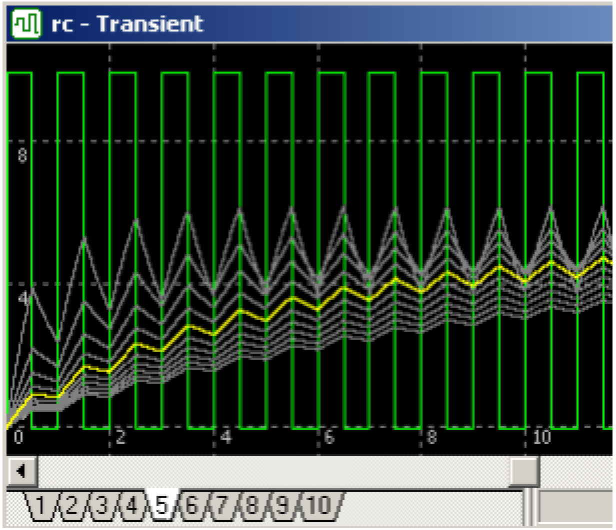
- **From**. Start parameter's value.
- **To**. End parameter's value.
- **Step type**:
    - **Step.** Parameter is incremented (or decremented) by **Step** value.
    - **Step, %.** Parameter is changed by specified **percentage**.
    - **Step, X**. Parameter is **multiplied** by specified value.
- **Steps** field shows total number of steps to be performed.

For **List** sweep enter the following parameters:

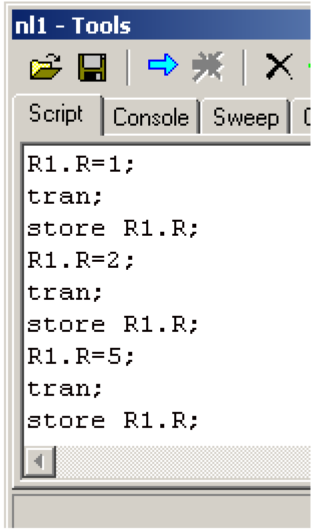
- **List** of parameter values in the text field, one value per line.



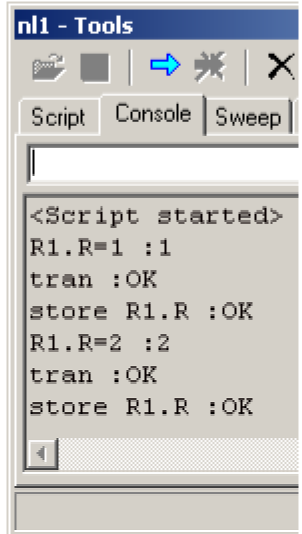
Click **Run script** button  to run sweep. Analysis results will be saved into storage. Only traces with enabled storage will be saved. **Script** page contains text of the script that was executed, and **Console** page contains log of the script execution.



Sweep results in the storage



Script text



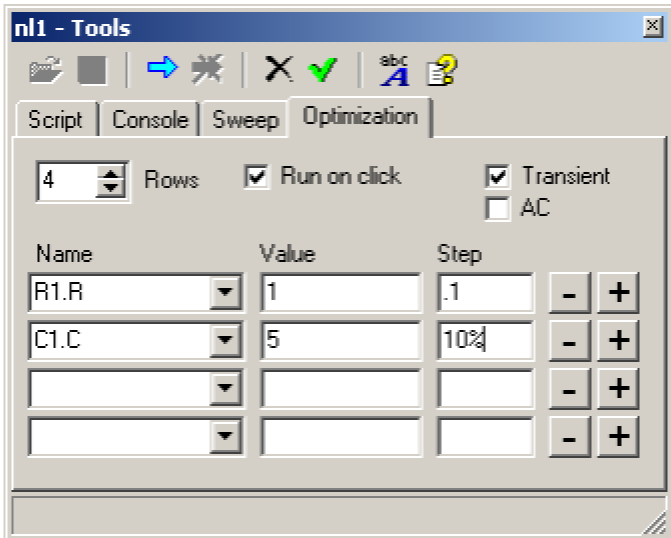
Execution log

Transient | Sweep and **AC | Sweep** commands open **Sweep** page with Transient or AC checkboxes selected, respectively.

Optimization

Optimization performs transient and/or AC analysis while manually iterating selected parameters.

Click **Optimization** Toolbar button or select **Tools | Optimization** command in the Main Menu to open **Optimization** page of **Tools** window:



Toolbar buttons perform the following operations:

- **Run** analysis.
- **Stop** analysis execution.
- **Clear** optimization table.
- **Clean up and update** optimization table. Removes rows with empty name field, moves rows up to fill up gaps, fills in **Value** fields with current component values.

The following parameters can be specified:

- **Rows**. Number of parameters to iterate, 1 to 16. If all rows do not fit the window, warning message will show up in the status bar: "*Resize window to see more rows*".
- **Run on click**. If selected, specified analysis will be started immediately when **Plus** or **Minus** button of any parameter is clicked. Otherwise, click button to run analysis.
- **Transient**. Select to run transient.
- **AC**. Select to run AC analysis.
- **Name** fields contain names of the parameters to be changed. Enter parameter's name manually, or select from drop-down list. The list shows all numerical parameters of the component, which is currently selected in the schematic or in the Components Window. (See example in the **Sweep** chapter).
- **Value** fields contain current value of specified parameter. To update fields click **Clean up and update** button.

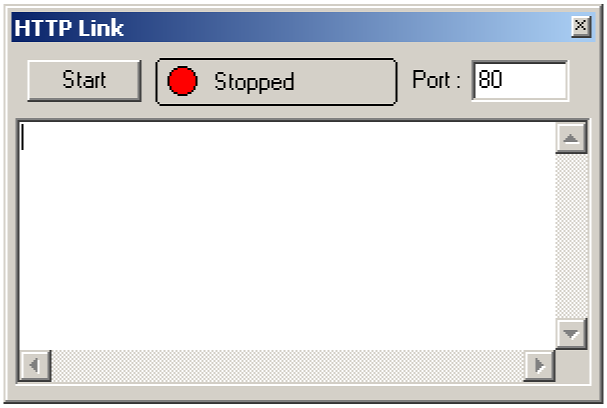
- **Step** fields specify how the value will change when **Plus** or **Minus button** is clicked. If **Step** is a number, the number will be added to current value. If **Step** is a number with '%' character at the end, the value will be changed by specified percentage.
- **Plus/Minus** buttons. When clicked, the value will be modified by specified step. If **Run on click** option is selected, specified analysis will be performed.

HTTP link

HTTP link provides NL5 link with external application. NL5 serves as a "server", running built-in HTTP server. External application is a "client". NL5 and a client application may run on the same computer or on different computers, communicating through local network or Internet.

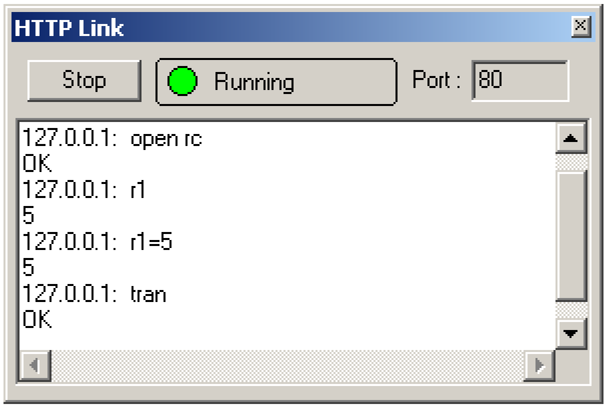
Starting HTTP server

Select **Tools | HTTP Link** command in the Main Menu to open **HTTP Link** window:



Default **Port** number is 80, which is standard port for HTTP protocol. If this port is already used by another application (most likely HTTP server), select any other available port number. Make sure the client application uses the same port number as NL5 in URL request. Please note that port number can be changed only before HTTP link is started the first time. To change the port number after that, close NL5 application and start it again. If NL5 and client application are running on different computers, make sure that any existing firewalls and routers are properly configured to allow TCP communication through specified port.

To start HTTP server click **Start** button. While server is running, the window will display IP address of the client, command received, and result sent to the client:



The window can be closed at any time without affecting server operation. To stop HTTP server click **Stop** button.

HTTP server can also be started automatically at NL5 startup using "–http" switch in command line:

```
>nl5.exe -http
```

Sending URL request

Client applications can issue commands and obtain data from NL5 by sending URL request with parameters, and receiving back response in the text format. The "URL read" function name, syntax, and parameters may vary for different applications. For example, in the MATLAB `urlread` function can be used:

```
s = urlread('url')
```

For other applications refer to Manual or Function Reference for HTTP or URL read functions with parameters.

URL string has the following general format:

http://*host_name:port*/?cmd=*command*

where:

- ***host_name*** is the name or IP address of the computer where NL5 is running. If client application is running on the same computer with NL5, *host_name* can be "127.0.0.1", or "localhost".
- ***port*** is the port number. It should be the same as port number specified in NL5. If default HTTP port number (80) is used, it can be omitted, so that URL string will look as follows:

  http://*host_namet*/?cmd=*command*

- ***command*** is a script command or an expression.

NL5 executes command or evaluates expression and responds with a text, which can be:

- "OK",
- result of the expression in text format,
- comma-separated string for trace data request,
- error message.

For example:

Request:	**Response:**
`http://localhost/?cmd=open rc.nl5`	`"OK"`
`http://127.0.0.1/?cmd=R1`	`"100"`
`http://192.168.0.1/?cmd=C1=1n`	`"2.2e-9"`
`http://public025:2119/?cmd=V(C1) 0,1,.2`	`"0,9.99999424754,9.9999944731,…"`
`http://localhost/?cmd=open test.nl5`	`"Error opening file test"`

If client application allows, the following modifications can be applied to URL string, making it more simple and readable:

- Text "http://" can be omitted.
- Text "cmd=" can be omitted.
- Several commands can be issued in one URL. Commands are separated by the '&' character, and the response consists of comma-separated responses for each command. For example:

For example:

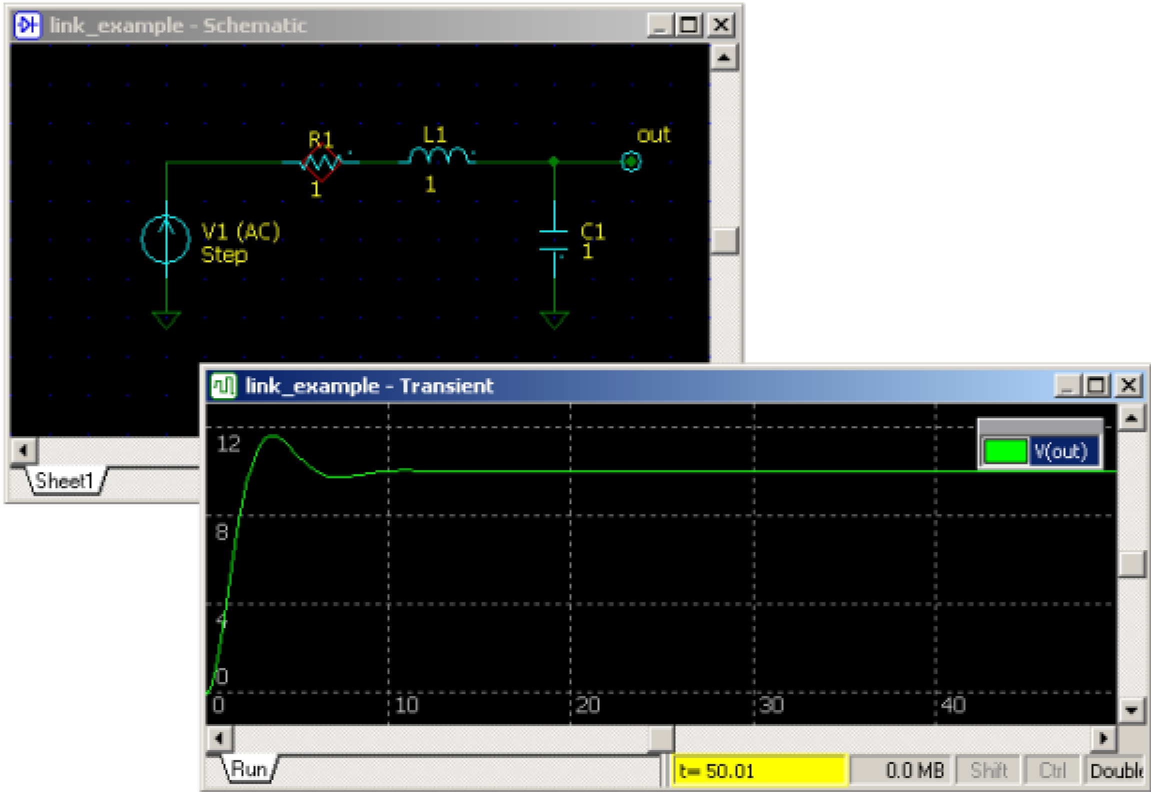
Request:	**Response:**
`127.0.0.1/?open rc`	`"OK"`
`192.168.0.101/?R1&C1&L1`	`"100,2.2e-9,100e-3"`
`http://public025:222/?store R1`	`"OK"`
`localhost/?V(C1).mean`	`"1.27978684602"`

Running simulation

When transient simulation command **tran** (or **cont**), or AC simulation command **AC** is issued through HTTP link, NL5 sends "OK" response immediately without waiting for simulation to complete. This is done in order to avoid "time-out" condition in case of long simulation time. The "client" should wait for simulation end by periodically sending command **ready** and checking NL5 response. Response "0" means that simulation is still running, response "1" means that simulation is complete and a new command can be issued. If the error occurs during simulation, NL5 responds with error message.

NL5-MATLAB link example

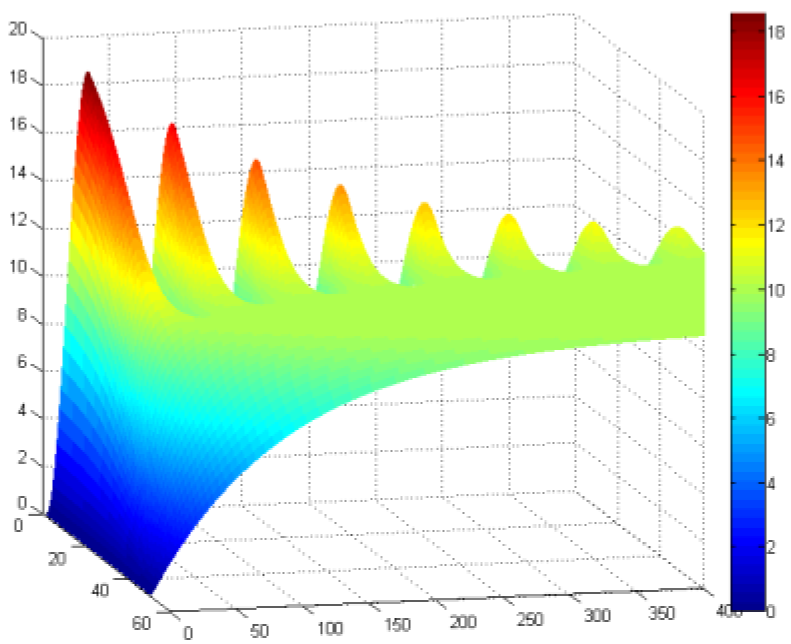
The following example shows using NL5 HTTP link with MATLAB. NL5 has HTTP server running, and schematic "link_example.nl5" loaded. Transient window shows schematic response on V1 step function for R1=1:



The code listed below has been loaded into MATLAB from the file "link_example.m" and executed:

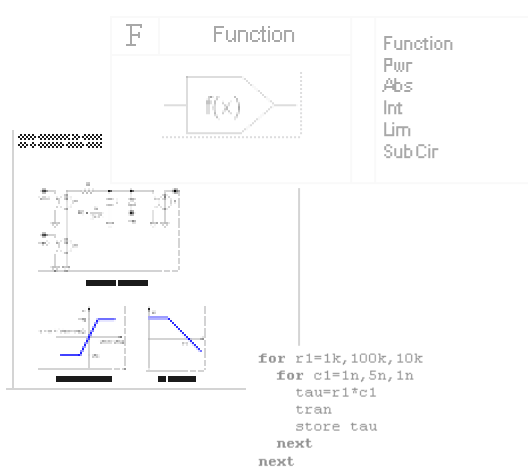
```
clear
clc
close all
R=logspace(-1,1);
Header='http://127.0.0.1/?';
Cmd2=[Header,'tran'];
Cmd3=[Header,'ready'];
Cmd4=[Header,'V(out) 0,50,.1'];
for k=1:length(R)
    Cmd1=[Header,'R1=',num2str(R(k))];
    urlread(Cmd1);
    urlread(Cmd2);
    Response='0';
    while strcmp(Response,'0')
        Response=urlread(Cmd3);
    end
    Graph(k,:)=str2num(urlread(Cmd4));
end
Graph=Graph';
surf(Graph)
shading flat
colormap jet
colorbar
ylim([0 400])
```

The code changes R1 in the range 0.1…10 with logarithmic step, calculates transient, reads V(out) trace data, and displays V(out) at 3-D graph as a function of time and R1 value:



Example schematic and MATLAB files are located in the `Examples/MATLAB` folder of the NL5 complete package download zip file.

VII. Attachments



1. Component Types, Models and Parameters

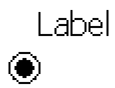
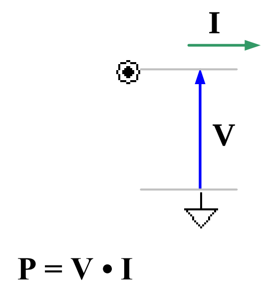
Component types are described in the following format:

Letter	Type	Models	Model 1 Model 2 …	Traces	Picture of V, I, P traces
Symbol					
Views	View 1 View 2 … (optional)		Description of views (optional)		

Model and parameters are presented as follows:

Letter	Type		Examples/Components/Example file name
Model	Parameter	Units	Description
Model name	Par 1	Unit 1	Par 1 description
	Par 2	Unit 2	Par 2 description
	…	…	…
Model specific view (optional)	Detailed description of the model and parameters.		

Model example files are located in the `Examples/Components` folder of the NL5 complete package download zip file.

	Label	Models	Models list	Traces	Traces diagram
			Label V Pulse Step Sin PWL Function File SubCir		

	Label		Label/Label_Label.nl5
Model	Parameter	Units	Description
Label	VIC	V	Initial condition: initial voltage.
	R	Ohm	Initial resistance.

Label. This model can be used:

- As a voltage trace probe point.
- For connecting schematic points without wires, including points at different sheets.
- To apply initial voltage during DC operating point calculation.

When calculating DC operating point, if initial voltage "VIC" is not blank, the temporary voltage source "VIC" is connected to the label through initial resistor "R". When DC operating point is found, the voltage source is removed.

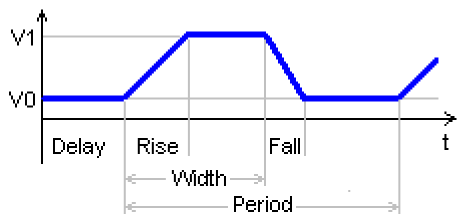
Please note: "VIC" is not a constant voltage source! This voltage will be removed after DC operating point calculation, and the label will be floating! For constant voltage source use **V** model.

	Label		Label/Label_V.nl5
Model	Parameter	Units	Description
V	V	V	Voltage.

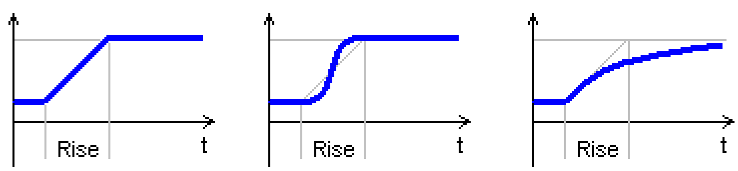
Constant voltage source. Voltage = "V".

	Label		Label/Label_Pulse.nl5
Model	Parameter	Units	Description
Pulse	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	Period	s	Period.
	Width	s	Pulse width.
	Slope		Slope type: Linear/Cos/Exp
	Rise	s	Pulse rise length.
	Fall	s	Pulse fall length.
	Delay	s	Delay before first pulse starts.

Pulse voltage source. Pulses start after “Delay” time. “Rise” time is included into “Width”, “Fall” time is not included into “Width”. Almost every parameter can be set to zero and infinity (“inf”); otherwise the error message will be displayed.



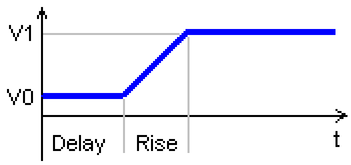
Slope type applies both to pulse rise and fall. The following slope types are available:



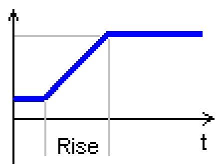
Linear **Cos (cosine)** **Exp (exponential)**

	Label		Label/Label_Step.nl5
Model	Parameter	Units	Description
Step	V1	V	Step On voltage.
	V0	V	Step Off voltage.
	Slope		Slope type: Linear/Cos/Exp
	Rise	s	Step rise length.
	Delay	s	Delay before step starts.

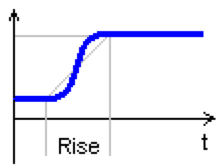
Step voltage source. Step starts after “Delay” time.



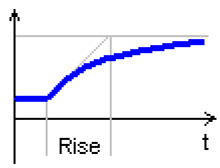
The following slope types are available:



Linear



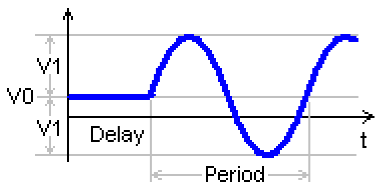
Cos (cosine)

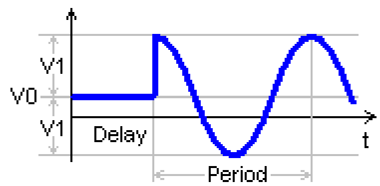


Exp (exponential)

	Label		Label/Label_Sin.nl5
Model	Parameter	Units	Description
Sin	V1	V	Voltage amplitude.
	V0	V	Voltage baseline.
	Period	s	Period.
	Phase	deg	Phase.
	Delay	s	Delay before sine signal starts.

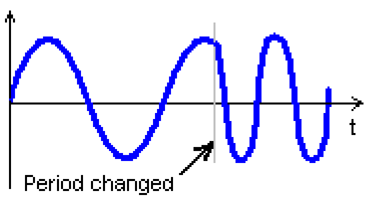
Sine voltage source. Sine signal starts after "Delay" time. "Phase" is sine phase in degrees at the moment when signal starts:





Phase = 0 **Phase = 90**

If transient is paused, sine period changed, then transient is continued, the phase of the signal remains continuous, providing smooth sine signal of variable frequency:



	Label	Label/Label_PWL.nl5

Model	Parameter	Units	Description
PWL	pwl		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Piece-wise linear voltage source. Signal is defined by "pwl" parameter in the csv ("comma-separated values") format, as follows:

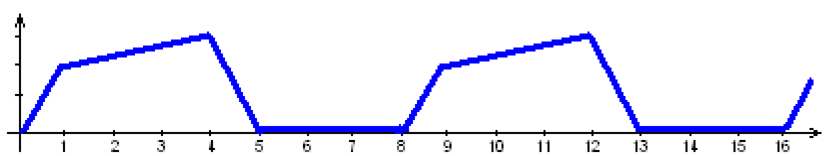
t0,V0,t1,V1,...,tn,Vn

where all t and V can be numerical values or expressions. If t<t0, signal is V0. If t0<t<t1, signal value is linearly interpolated between V0 and V1, etc. If t>tn, then signal value is Vn if "Cycle" parameter is set to "No", otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
pwl = 0,0,1,2,4,3,5,0,8,0
```

If "Cycle" = Yes, "Delay" = 0, the following voltage will be generated:



	Label	Label/Label_Function.nl5

Model	Parameter	Units	Description
Function	F(t)	V	Function

Arbitrary function. F(t) defines voltage as a function of the following variables:

t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(t) is blank, voltage is zero.

Example:

F(t) = sin(t) * (1+cos(t*.01))
F(t) = V(R1) * I(R1)

Please note that **V**, **I**, and **P** variables are taken at previous calculation step. This may affect stability of the schematic with closed loop.

	Label	Label/Label_File.nl5

Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Voltage source defined in the text file. "File" parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Signal is defined in the csv ("comma-separated values") format, as follows:

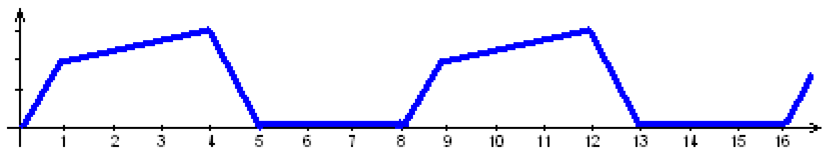
```
<if first line does not start with a number, it is ignored >
t0,V0
t1,V1
.....
tn,Vn
```

where all t and V can be numerical values or expressions. If t<t0, signal is V0. If t0<t<t1, signal value is linearly interpolated between V0 and V1, etc. If t>tn, then signal value is Vn if "Cycle" parameter is set to "No", otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
0,0
1,2
4,3
5,0
8,0
```

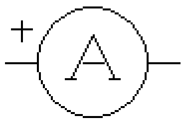
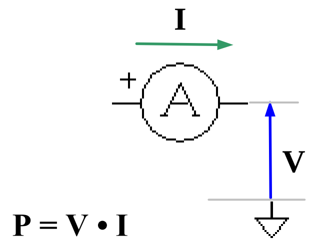
If "Cycle" = Yes, "Delay" = 0, the following voltage will be generated:



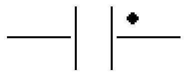
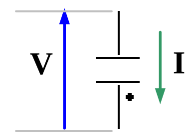
	Label	Label/Label_SubCir.nl5

Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

A	Amperemeter	Models	Amperemeter	Traces	
					

A	Amperemeter	A/ A_Amperemeter_Amperemeter.nl5
Model	No parameters	
Amperemeter	**Amperemeter.** Short circuit. In addition to current, amperemeter can measure voltage relative to ground, and power to grounded load.	

C	Capacitor	Models		Traces	
			C PWL Function SubCir		 P = V • I

C	Capacitor		C/C_Capacitor_C.nl5
Model	Parameter	Units	Description
C	C	F	Capacitance
	IC	V	Initial condition: voltage. Leave blank if IC not defined.

Linear capacitor. I = C*dV/dt.

When calculating DC operating point, if "IC" is defined, capacitor is replaced with voltage source equal to IC. If "IC" is not defined (blank), capacitor is temporarily removed (open circuit), DC operating point calculated, and then the voltage found across the capacitor is assigned to the capacitor as its initial voltage.

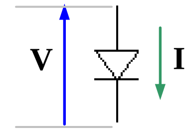
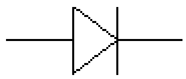
C	Capacitor		C/C_Capacitor_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, C(V)
	IC	V	Initial condition: voltage. Leave blank if no IC defined.

Piece-wise linear capacitor. "pwl" string defines capacitance as a function of voltage. See *Working with PWL* chapter for details.

When calculating DC operating point, if "IC" is defined, capacitor is replaced with voltage source equal to IC. If "IC" is not defined (blank), capacitor is temporarily removed (open circuit), DC operating point calculated, and then the voltage found across the capacitor is assigned to the capacitor as its initial voltage.

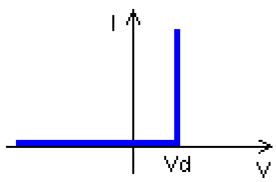
C	Capacitor		C/C_Capacitor_Function.nl5
Model	Parameter	Units	Description
Function	Z(s)	Ohm	Impedance as a function of ***s*** parameter.
	Impedance function in *s* domain. For transient, constant impedance Z(0) is used. For linearized AC analysis, complex impedance Z(s) is used. The following variables can be used in the function: ***f*** – current AC frequency, Hz ***w*** – angular AC frequency, ***w = 2πf*** . ***s*** or ***p*** – Laplace parameter, ***s = p = j*2πf***. Example: Z(s) = 1.0/3p/s + 1.5p - 3 pF capacitor in series with 1.5 pH inductor. Z(s) = (1.0/10n/s)||1k - 10 nF capacitor in parallel with 1 kOhm resistor.		

C	Capacitor		C/C_Capacitor_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

D	Diode	Models	Diode Storage Soft PWL SubCir	Traces	 P = V • I
					

D	Diode		D/D_Diode_Diode.nl5
Model	Parameter	Units	Description
Diode	Vd	V	Forward voltage drop.
	IC		Initial condition: On/Off.

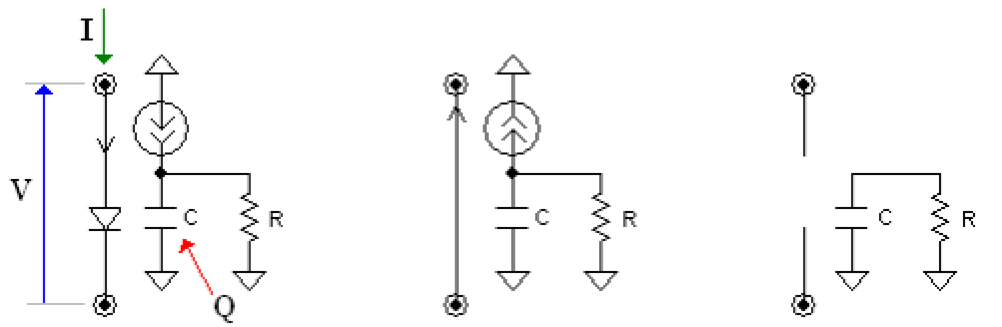
Ideal diode. If V >= "Vd", diode is On (short circuit). Otherwise diode is Off (open circuit, I=0).



When calculating DC operating point diode is set to the state specified in "IC".

D	Diode		D/D_Diode_Storage.nl5
Model	Parameter	Units	Description
Storage	Vd	V	Forward voltage drop.
	t	s	Recombination time constant.
	IC		Initial condition: Off/On.
	ICQ	C (A*s)	Initial condition: charge.

Charge storage diode. Simplified equivalent schematic of the model is the following:

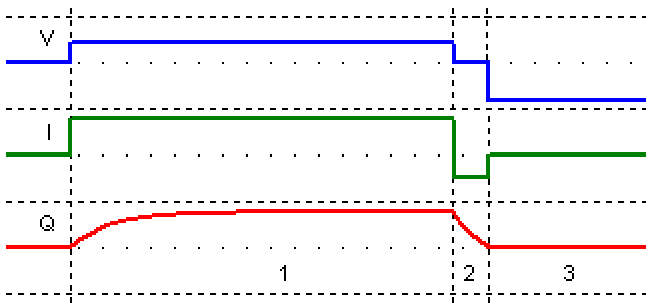


1. Forward current
V = "Vd", I > 0, Q > 0

2. Reverse current
V = 0, I < 0, Q > 0

3. No current
V < 0, I = 0, Q = 0

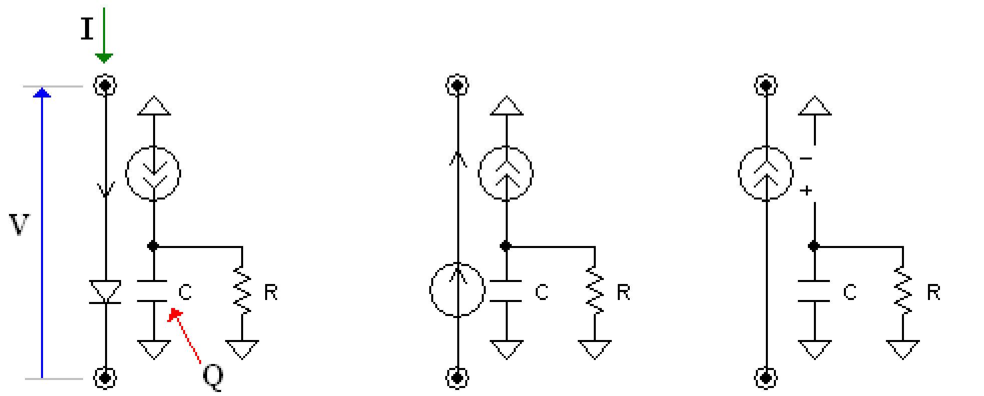
The diode has internal capacitor C and resistor R, with the time constant RC = "t", Q is the charge on the capacitor. In mode 1, forward current flows through the diode and forward voltage drop is "Vd". At the same time, the current equal to forward current is charging capacitor C. In mode 2, reverse current is applied to the diode, and capacitor C is being discharged by the current equal to reverse current. As long as charge Q on the capacitor is positive, the diode is a short circuit with zero voltage drop. Finally, when charge drops to zero, the diode switches to mode 3, with zero current and negative voltage drop (open circuit). V, I, and Q waveforms are shown on the graph:



When calculating DC operating point the diode is set to the state specified in "IC", and internal charge Q is set to specified "ICQ" value.

D	Diode		D/D_Diode_Soft.nl5
Model	Parameter	Units	Description
Soft	Vd	V	Forward voltage drop.
	t	s	Recombination time constant.
	ts	s	Soft recovery time constant.
	IC		Initial condition: Off/On.
	ICQ	C (A*s)	Initial condition: charge.

Soft recovery charge storage diode. Simplified equivalent schematic of the model is the following:



1. Forward current
V = "Vd", I > 0, Q > 0
C = 1, R = "t"

2. Reverse current
V = "Vd", I < 0, Q > 0
C = 1, R = "t"

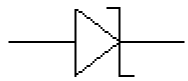
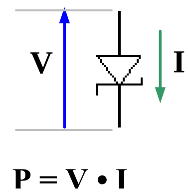
3. Soft recovery
V < "Vd", I < 0, Q > 0
C = 1, R = "ts"

The diode has internal capacitor C=1 and resistor R. Time constant RC is equal either recombination time constant "t", or soft recovery time constant= "ts". Q is the charge on the capacitor. In mode 1, forward current flows through the diode and forward voltage drop is "Vd". At the same time, the current equal to forward current is charging capacitor C. In mode 2, reverse current is applied to the diode, and capacitor C is being discharged by the current equal to reverse current. Voltage drop on the diode is still "Vd". At the moment when reverse current is equal or less than charge divided by soft recovery time constant "ts", a mode 3 is turned on. In mode 3, capacitor C is being exponentially discharged by the current through resistor R with time constant "ts" (plus small constant current to ensure full discharge - not shown on the picture). Reverse diode current is proportional to the charge. As soon as charge drops to zero, the diode switches to mode 4 (not shown), with zero current and negative voltage drop (open circuit). See demo schematic D/D_Diode_Soft.nl5 for examples of soft recovery waveforms.

When calculating DC operating point the diode is set to the state specified in "IC", and internal charge Q is set to specified "ICQ" value.

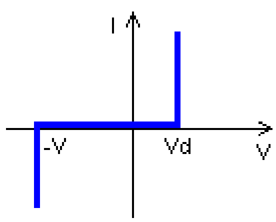
D	Diode		D/D_Diode_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, R(V)
	Piece-wise linear diode. "pwl" string defines resistance as a function of voltage. See *Working with PWL* chapter for details.		

D	Diode		D/D_Diode_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

D	Zener	Models	Zener PWL SubCir	Traces	
					

D	Zener	D/D_Zener_Zener.nl5	
Model	Parameter	Units	Description
Zener	V	V	Breakdown voltage drop.
	Vd	Vd	Forward voltage drop.
	IC		Initial condition: Minus/Off/Plus.

Ideal zener. If V <= -"V" or V >= "Vd", zener is On (short circuit). Otherwise zener is Off (open circuit, I=0).



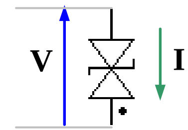
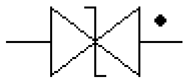
When calculating DC operating point zener is set to the state specified in "IC".

D	Zener	D/D_Zener_PWL.nl5	
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, R(V)

Piece-wise linear zener. "pwl" string defines resistance as a function of voltage. See *Working with PWL* chapter for details.

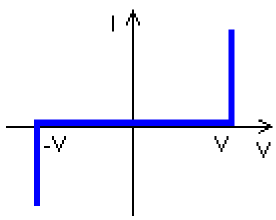
D	Zener	D/D_Zener_SubCir.nl5	
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

D	Bidirectional zener	Models	Zener PWL SubCir	Traces	 P = V • I
					

D	Bidirectional zener		D/D_BZener_Zener.nl5
Model	Parameter	Units	Description
Zener	V	V	Breakdown voltage drop.
	IC		Initial condition: Minus/Off/Plus.

Ideal bidirectional zener. If V <= -"V" or V >= "V", zener is On (short circuit). Otherwise zener is Off (open circuit, I=0).



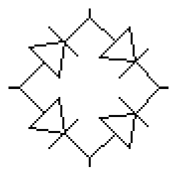
When calculating DC operating point zener is set to the state specified in "IC".

D	Bidirectional zener		D/D_BZener_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, R(V)

Piece-wise linear zener. "pwl" string defines resistance as a function of voltage. See *Working with PWL* chapter for details.

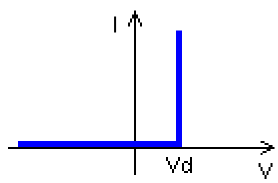
D	Bidirectional zener		D/D_BZener_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

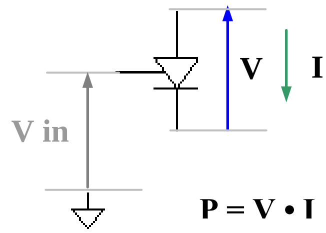
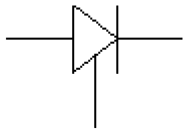
D	Bridge rectifier	Models	Diode	Traces	$P = V_1 \bullet I_1 + V_2 \bullet I_2 + V_3 \bullet I_3 + V_4 \bullet I_4$
					

D	Bridge rectifier		D/D_Bridge_Diode.nl5
Model	Parameter	Units	Description
Diode	Vd	V	Forward voltage drop.

Bridge rectifier with ideal diodes. For each diode, if V >= "Vd", diode is On (short circuit). Otherwise diode is Off (open circuit, I=0).



When calculating DC operating point all diodes are Off.

D	Logic controlled thyristor	Models	Thyristor SubCir	Traces	
					

D	Logic controlled thyristor		D/D_LCT_Thyristor.nl5
Model	**Parameter**	**Units**	**Description**
Thyristor	Vd	V	Forward voltage drop.
	Ihold	A	Holding current.
	IC		Initial condition: Off/On.

Ideal thyristor. Thyristor has two states:

- Off state (non-conducting): open circuit.
- On state (conducting): ideal diode with "Vd" forward voltage drop.

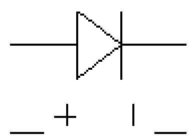
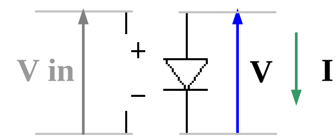
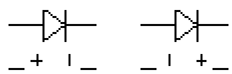
If control voltage "Vin" is greater than logical threshold, thyristor is in On state (ideal diode). When control voltage drops below logical threshold, thyristor stays in On state as long as current I exceeds holding current "Ihold", and voltage V is not negative.

To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point thyristor is set to the state specified in "IC".

D	Logic controlled thyristor		D/D_LCT_SubCir.nl5
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

D	Voltage controlled thyristor	Models	Thyristor SubCir	Traces	
					 P = V • I
Views					

D	Voltage controlled thyristor		D/D_VCT_Thyristor.nl5
Model	Parameter	Units	Description
Thyristor	Vd	V	Forward voltage drop.
	Ihold	A	Holding current.
	Threshold	V	Voltage threshold.
	IC		Initial condition: Off/On.

Ideal thyristor. Thyristor has two states:

- Off state (non-conducting): open circuit.
- On state (conducting): ideal diode with "Vd" forward voltage drop.

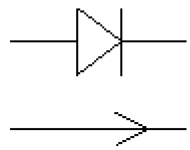
If control voltage "Vin" is greater than "Threshold", thyristor is in On state (ideal diode). When control voltage drops below "Threshold", thyristor stays in On state as long as current I exceeds holding current "Ihold", and voltage V is not negative.

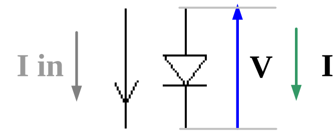
When calculating DC operating point thyristor is set to the state specified in "IC".

D	Voltage controlled thyristor		D/D_VCT_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

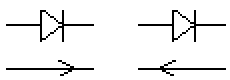
D	Current controlled thyristor	Models	Thyristor SubCir	Traces	





P = V • I

Views



D	Current controlled thyristor		D/D_CCT_Thyristor.nl5
Model	Parameter	Units	Description
Thyristor	Vd	V	Forward voltage drop.
	Ihold	A	Holding current.
	Threshold	A	Current threshold.
	IC		Initial condition: Off/On.

Ideal thyristor. Thyristor has two states:

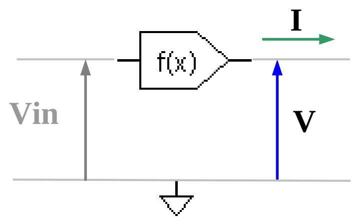
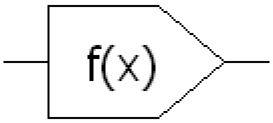
- Off state (non-conducting): open circuit.
- On state (conducting): ideal diode with "Vd" forward voltage drop.

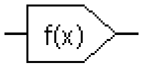
If control current "Iin" is greater than "Threshold", thyristor is in On state (ideal diode). When control current drops below "Threshold", thyristor stays in On state as long as current I exceeds holding current "Ihold", and voltage V is not negative.

When calculating DC operating point thyristor is set to the state specified in "IC".

D	Current controlled thyristor		D/D_CCT_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

<table>
<tr><td>F</td><td>Function</td><td rowspan="2">Models</td><td rowspan="2">Function
Pwr
Abs
Int
Lim
SubCir</td><td rowspan="2">Traces</td><td rowspan="2">P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
</table>

F	Function		F/F_Function_Function.nl5
Model	Parameter	Units	Description
Function 	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.

Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – input voltage Vin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x*** and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain at each frequency.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

F	Function	F/F_Function_Pwr.nl5

Model	Parameter	Units	Description
Pwr	power		Power.
	K	V/V	Gain.
	IC	V	Initial condition: output voltage.

"Signed" power function. V = K * pwr(Vin, power).

The function is calculated as follows:

if power = 0:
    if Vin < 0 . . . : V = -K
    if Vin = 0 . . . : V = 0
    if Vin > 0 . . . : V = K

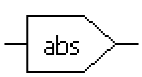
if power ≠ 0:

    if Vin < 0 . . . : $V = - K * (-Vin)^{power}$
    if Vin = 0 . . . : V = 0
    if Vin > 0 . . . : $V(= K * Vin^{power}$

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

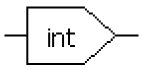
F	Function	F/F_Function_Abs.nl5

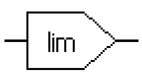
Model	Parameter	Units	Description
Abs	K	V/V	Gain.
	IC	V	Initial condition: output voltage.



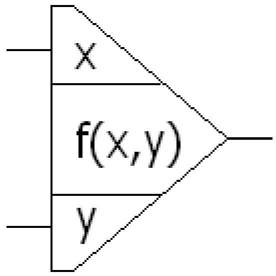
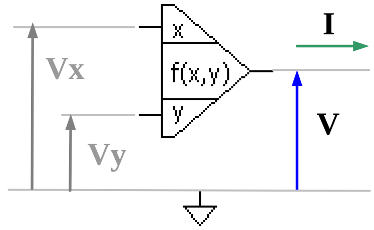
Absolute value. V = K * abs(Vin).

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

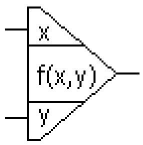
F	Function		F/F_Function_Int.nl5
Model	**Parameter**	**Units**	**Description**
Int 	resolution	V	Resolution.
	K	V/V	Gain.
	IC	V	Initial condition: output voltage.
	Rounding function. V = K * round(Vin, resolution). Round to the nearest multiple of "resolution". If resolution = 1, round to the nearest integer. When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.		

F	Function		F/F_Function_Lim.nl5
Model	**Parameter**	**Units**	**Description**
Lim 	Max	V	Maximum.
	Min	V	Minimum.
	IC	V	Initial condition: output voltage.
	Limiting function. The function is calculated as follows: if Vin < Min . . . : V = Min if Vin > Max . . : V = Max Otherwise . . . . . : V = Vin When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.		

F	Function		F/F_Function_SubCir.nl5
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

F	Function-2	Models		Traces	
			Function Mul Div Sum Sub Max Min GT LT Pwr Mag Phase SubCir		 P = V • I

F	Function-2		F/F_Function-2_Function.nl5
Model	Parameter	Units	Description
Function	F(x,y)	V	Output as function of the inputs.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.



Arbitrary function.

Transient analysis. F(x,y) defines output voltage as a function of the following variables:

x – input voltage Vx
y – input voltage Vy
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x,y) is blank, output is zero. F(s) is ignored.

Example:
F(x,y) = sqrt(x*x+y*y)
F(x,y) = x * y * sin(t)
F(x,y) = P(r1)+P(r2)

Please note that input voltages ***x***, ***y***, and variables **V**, **I, P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

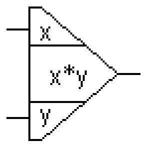
AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x,y) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain..

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

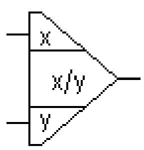
F	Function-2		F/F_Function-2_Mul.nl5
Model	Parameter	Units	Description
Mul	K	V/V	Gain.
	IC	V	Initial condition: output voltage.



Multiplication. V = K * Vx * Vy.

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Div.nl5
Model	Parameter	Units	Description
Div	K	V/V	Gain.
	IC	V	Initial condition: output voltage.

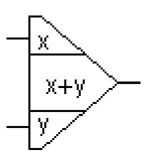


Division. V = K * Vx / Vy.

If Vy = 0, V = 0.

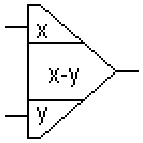
When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

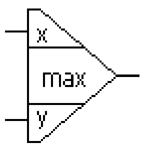
F	Function-2		F/F_Function-2_Sum.nl5
Model	Parameter	Units	Description
Sum	K	V/V	Gain.
	IC	V	Initial condition: output voltage.

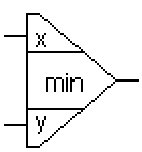


Addition. V = K * (Vx + Vy).

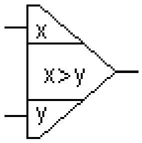
When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Sub.nl5
Model	Parameter	Units	Description
Sub	K	V/V	Gain.
	IC	V	Initial condition: output voltage.
			**Subtraction.** V = K * (Vx - Vy). When calculating DC operating point output is set to specified output voltage “IC”. When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Max.nl5
Model	Parameter	Units	Description
Max	K	V/V	Gain.
	IC	V	Initial condition: output voltage.
			**Maximum.** V = K * max(Vx, Vy). if Vx >= Vy . . : V = K * Vx if Vx < Vy . . . : V = K * Vy When calculating DC operating point output is set to specified output voltage “IC”. When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Min.nl5
Model	Parameter	Units	Description
Min	K	V/V	Gain.
	IC	V	Initial condition: output voltage.
			**Minimum.** V = K * min(Vx, Vy). if Vx >= Vy . . : V = K * Vy if Vx < Vy . . . : V = K * Vx When calculating DC operating point output is set to specified output voltage “IC”. When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_GT.nl5
Model	Parameter	Units	Description
GT	IC	V	Initial condition: output voltage.



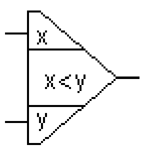
Greater than. V = Vx > Vy ? High : Low.

if Vx <= Vy . . : V = Low
if Vx > Vy . . . : V = High

High and Low are logical levels. To see and set logical levels go to **Transient | Settings**, or **AC | Settings** then click **Advanced** button.

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_LT.nl5
Model	Parameter	Units	Description
LT	IC	V	Initial condition: output voltage.



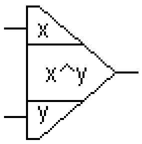
Less than. V = Vx < Vy ? High : Low.

if Vx < Vy . . . . : V = High
if Vx >= Vy . . . : V = Low

High and Low are logical levels. To see and set logical levels go to **Transient | Settings**, or **AC | Settings** then click **Advanced** button.

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Pwr.nl5
Model	**Parameter**	**Units**	**Description**
Pwr	K	V/V	Gain.
	IC	V	Initial condition: output voltage.



“Signed” power function. V = K * pwr(Vx, Vy).

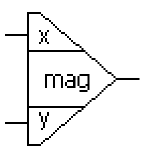
The function is calculated as follows:

if Vy = 0:
  if Vx < 0. . . : V = -K
  if Vx = 0. . . : V = 0
  if Vx > 0. . . : V = K

if Vy ≠ 0:
  if Vx < 0. . . : $V = -K * (-Vx)^{Vy}$
  if Vx = 0. . . : V = 0
  if Vx > 0. . . : $V = K * Vx^{Vy}$

When calculating DC operating point output is set to specified output voltage “IC”. When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

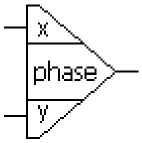
F	Function-2		F/F_Function-2_Mag.nl5
Model	**Parameter**	**Units**	**Description**
Mag	K	V/V	Gain.
	IC	V	Initial condition: output voltage.



Magnitude. $V = K * sqrt(Vx^2 + Vy^2)$.

When calculating DC operating point output is set to specified output voltage “IC”. When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_Phase.nl5
Model	Parameter	Units	Description
Phase	K	V/V	Gain.
	IC	V	Initial condition: output voltage.



Phase. V = K * phase(Vx, Vy).

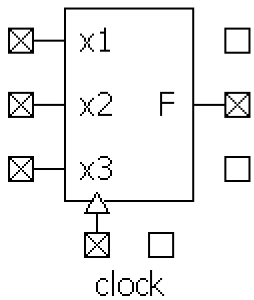
V in Volts is equal to phase of a vector Vx + jVy in degrees.
If Vx = 0 and Vy = 0: V = 0.

When calculating DC operating point output is set to specified output voltage "IC". When calculating transient, output voltage is always delayed by one calculation step. This may affect stability of the schematic with closed loop.

F	Function-2		F/F_Function-2_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

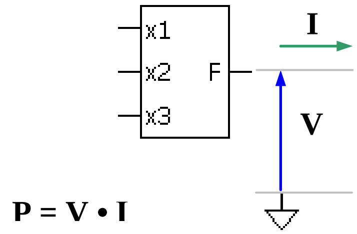
F Custom function



Models

Function

Traces

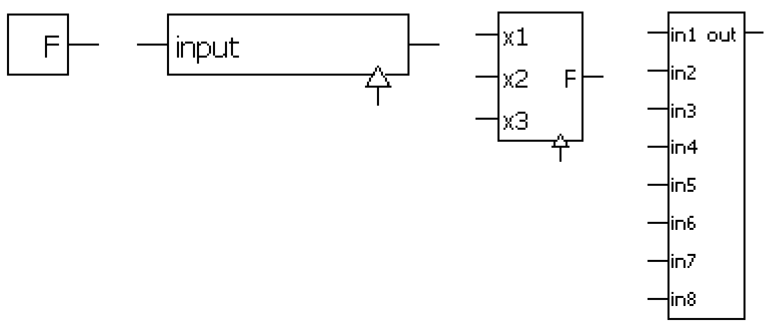


This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:

- arbitrary size up to 32(width) X 8(height),
- up to 8 inputs on the left side,
- one output on the right side,
- one or no clock pins on the bottom side.
- custom input and output names.

Examples of Custom function component:



F	Custom function		F/F_Custom_Function_Ex1.nl5 F/F_Custom_Function_Ex2.nl5 F/F_Custom_Function_Ex3.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the inputs.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.

Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

pin_name – input voltage on the input pin "pin_name"
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = max(x1,x2,x3)
F(x) = (in1+in2)*V(R1)

If ***clock*** pin does not exist, the model operates in "continuous" mode: the function is calculated and applied to the output on every calculation step. Please note that input voltages and variables **V**, **I, P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

If ***clock*** pin exists, the model operates in "synchronized" mode: the function is calculated and applied to the output only on rising edge of logical clock signal. As a result, "synchronized" mode provides faster simulation than "continuous" mode.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:

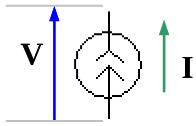
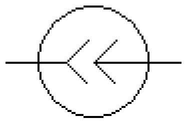
f – current AC frequency, Hz
w – angular AC frequency, $w = 2\pi f$.
s or ***p*** – Laplace parameter, $s = p = j*2\pi f$.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

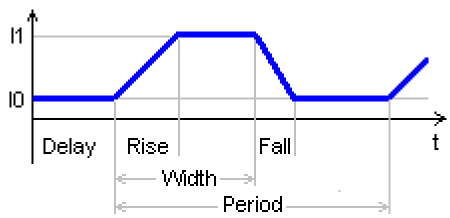
If ***clock*** pin exists, F(s) is ignored, and transfer function of the model is zero.

<table>
<tr><td>I</td><td>Current source</td><td rowspan="2">Models</td><td rowspan="2">I
Pulse
Step
Sin
PWL
Function
File
SubCir</td><td rowspan="2">Traces</td><td rowspan="2">
P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
</table>

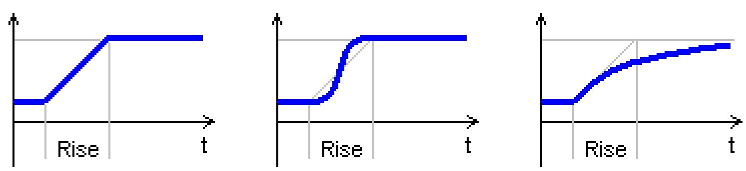
I	Current source		I/I_CS_I.nl5
Model	**Parameter**	**Units**	**Description**
I	I	A	Current.
	Constant current source. Current = "I".		

I	Current source		I/I_CS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	I1	A	Pulse On current.
	I0	A	Pulse Off current.
	Period	s	Period.
	Width	s	Pulse width.
	Slope		Slope type: Linear/Cos/Exp
	Rise	s	Pulse rise length.
	Fall	s	Pulse fall length.
	Delay	s	Delay before first pulse starts.

Pulse current source. Pulses start after "Delay" time. "Rise" time is included into "Width", "Fall" time is not included into "Width". Almost every parameter can be set to zero and infinity ("inf"), otherwise the error message will be displayed.



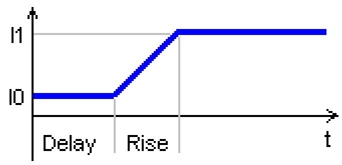
Slope type applies both to pulse rise and fall. The following slope types are available:



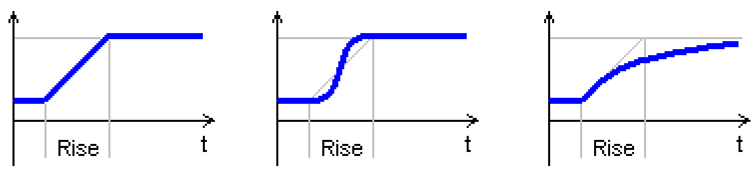
Linear **Cos (cosine)** **Exp (exponential)**

I	Current source		I/I_CS_Step.nl5
Model	Parameter	Units	Description
Step	I1	A	Step On current.
	I0	A	Step Off current.
	Slope		Slope type: Linear/Cos/Exp
	Rise	s	Step rise length.
	Delay	s	Delay before step starts.

Step current source. Step starts after "Delay" time.



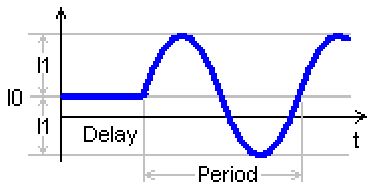
The following slope types are available:



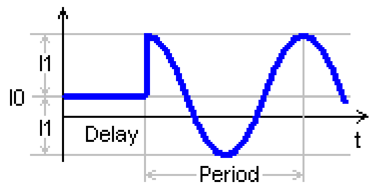
Linear **Cos (cosine)** **Exp (exponential)**

I	Current source		I/I_CS_Sin.nl5
Model	Parameter	Units	Description
Sin	I1	A	Current amplitude.
	I0	A	Current baseline.
	Period	s	Period.
	Phase	deg	Phase.
	Delay	s	Delay before sine signal starts.

Sine current source. Sine signal starts after “Delay” time. “Phase” is sine phase in degrees at the moment when signal starts:

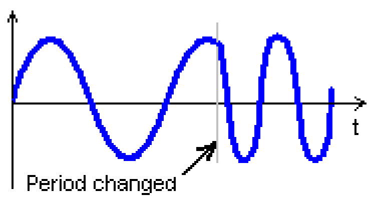


Phase = 0



Phase = 90

If transient is paused, sine period changed, then transient is continued, the phase of the signal remains continuous, providing smooth sine signal of variable frequency:



I	Current source		I/I_CS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Piece-wise linear current source. Signal is defined by "pwl" parameter in the csv ("comma-separated values") format, as follows:

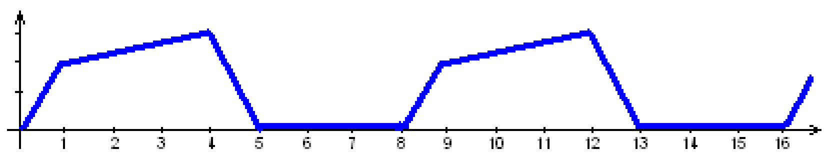
t0,I0,t1,I1,...,tn,In

where all t and I can be numerical values or expressions. If t<t0, signal is I0. If t0<t<t1, signal value is linearly interpolated between I0 and I1, etc. If t>tn, then signal value is In if "Cycle" parameter is set to "No", otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
pwl = 0,0,1,2,4,3,5,0,8,0
```

If "Cycle" = Yes, "Delay" = 0, the following current will be generated:



I	Current source		I/I_CS_Function.nl5
Model	Parameter	Units	Description
Function	F(t)	A	Function

Arbitrary function. F(t) defines current as a function of the following variables:

t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(t) is blank, current is zero.

Example:

F(t) = sin(t) * (1+cos(t*.01))
F(t) = V(R1) * I(R1)

Please note that **V**, **I**, and **P** variables are taken at previous calculation step. This may affect stability of the schematic with closed loop.

I	Current source		I/I_CS_File.nl5
Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Current source defined in the text file.“File“ parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Signal is defined in the following format:

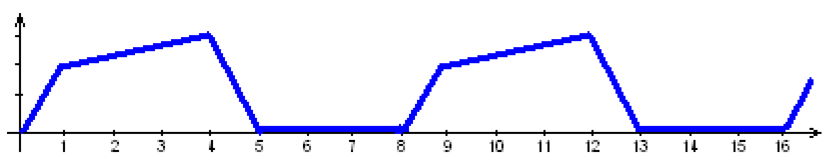
```
<if first line does not start with a number, it is ignored>
t0,I0
t1,I1
.....
tn,In
```

where all t and I can be numerical values or expressions. If t<t0, signal is v0. If t0<t<t1, signal value is linearly interpolated between I0 and I1, etc. If t>tn, then signal value is In if “Cycle” parameter is set to “No”, otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by “Delay” time.

Example:

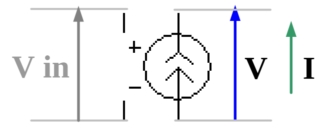
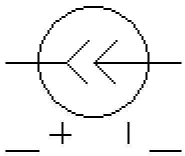
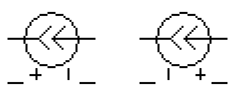
```
0,0
1,2
4,3
5,0
8,0
```

If “Cycle” = Yes, “Delay” = 0, the following current will be generated:



I	Current source		I/I_CS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

I	Voltage controlled current source	Models	Linear Function PWL VCO Pulse PWM SubCir	Traces	 **P = V • I**
 					
Views					

I	Voltage controlled current source			I/I_VCCS_Linear.nl5
Model	Parameter	Units	Description	
Linear	K	A/V	Gain	
	Linear voltage controlled current source. I = K * Vin.			

I	Voltage controlled current source		I/I_VCCS_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	A	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	A	Initial condition: output current.

Arbitrary function.

Transient analysis. F(x) defines output current as a function of the following variables:

- ***x*** – input voltage Vin
- ***t*** - current time
- **V(*name*)** - voltage on the component ***name***
- **I(*name*)** - current through the component ***name***
- **P(*name*)** – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x*** and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:

- ***f*** – current AC frequency, Hz
- ***w*** – angular AC frequency, ***w = 2πf*** .
- ***s*** or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output current "IC".

I	Voltage controlled current source		I/I_VCCS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Vin)
	Piece-wise linear voltage controlled current source. Source gain K is defined by "pwl" string as a function of input voltage Vin. See *Working with PWL* chapter for details.		

I	Voltage controlled current source		I/I_VCCS_VCO.nl5
Model	Parameter	Units	Description
VCO	I1	A	Current amplitude (Sin), or Pulse On current (Pulse).
	I0	A	Current baseline (Sin), or Pulse Off current (Pulse).
	dFdV	Hz/V	Gain.
	Phase	deg	Phase.
	Type		Signal type: Sin/Square/Triangle/Sawtooth.
	Voltage controlled oscillator. Output current is a signal with frequency equal to: f(Hz) = dFdV * Vin. For Sine signal, "I0" is baseline, and "I1" is amplitude. For Square, Triangle, and Sawtooth signals, "I0" is Off level, "I1" is On level. "Phase" is additional phase of the signal, in degrees.		

I	Voltage controlled current source		I/I_VCCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	V	Voltage threshold.
	I1	A	Pulse On current.
	I0	A	Pulse Off current.
	One-shot pulse generator. When increasing input voltage Vin crosses "Threshold" value, current pulse of "Width" duration is generated. "I0" is pulse Off level, "I1" is pulse On level. If increasing Vin crosses "Threshold" value while pulse is generated, the pulse is restarted.		

I	Voltage controlled current source		I/I_VCCS_PWM.nl5
Model	Parameter	Units	Description
PWM	I1	A	Pulse On current.
	I0	A	Pulse Off current.
	F	Hz	Frequency.
	Phase	deg	Phase.
	Vmax	V	Input voltage corresponding to 100% duty.

Voltage controlled Pulse-Width Modulator. Output current is a pulse signal of frequency "F" shifted by "Phase". Input voltage Vin is sampled at the beginning of each cycle of the signal, and width of the output pulse during this cycle is calculated according to the equation:

width = 1/F * (Vin / Vmax)

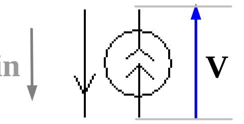
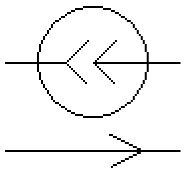
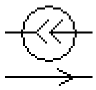
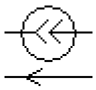
or

duty = 100% * (Vin / Vmax);

If the width is equal or less than zero, a short "On" pulse with the width equal to the minimum calculation step at that moment will be generated. If the width is equal or greater than period of frequency "F", a short "Off" pulse at the end of the period will be generated. Due to that, the frequency of the output signal is always "F". Such a signal can be, for instance, divided by D-trigger to create a signal with duty cycle less than 50%.

I	Voltage controlled current source		I/I_VCCS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

I	Current controlled current source	Models	Linear Function PWL CCO Pulse PWM SubCir	Traces	 $P = V \bullet I$
					
Views	 				

I	Current controlled current source		I/I_CCCS_Linear.nl5
Model	Parameter	Units	Description
Linear	K	A/A	Gain
	Linear current controlled current source. I = K * Iin.		

I	Current controlled current source		I/I_CCCS_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	A	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	A	Initial condition: output current.

Arbitrary function.

Transient analysis. F(x) defines output current as a function of the following variables:

x – input current Iin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input current ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output current "IC".

I	Current controlled current source		I/I_CCCS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Iin)
	Piece-wise linear current controlled current source. Source gain K is defined by "pwl" string as a function of input current Iin. See *Working with PWL* chapter for details.		

I	Current controlled current source		I/I_CCCS_CCO.nl5
Model	Parameter	Units	Description
CCO	I1	A	Current amplitude (Sin), or Pulse On current (Pulse).
	I0	A	Current baseline (Sin), or Pulse Off current (Pulse).
	dFdI	Hz/A	Gain.
	Phase	deg	Phase.
	Type		Signal type: Sin/Square/Triangle/Sawtooth.
	Current controlled oscillator. Output current is a signal with frequency equal to: f(Hz) = dFdI * Iin. For Sine signal, "I0" is baseline, and "I1" is amplitude. For Square, Triangle, and Sawtooth signals, "I0" is Off level, "I1" is On level. "Phase" is additional phase of the signal, in degrees.		

I	Current controlled current source		I/I_CCCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	A	Current threshold.
	I1	A	Pulse Off current..
	I0	A	Phase.
	One-shot pulse generator. When increasing input current Iin crosses "Threshold" value, current pulse of "Width" duration is generated. "I0" is pulse Off level, "I1" is pulse On level. If increasing Iin crosses "Threshold" value while pulse is generated, the pulse is restarted.		

I	Current controlled current source		I/I_CCCS_PWM.nl5
Model	Parameter	Units	Description
PWM	I1	A	Pulse On current.
	I0	A	Pulse Off current.
	F	Hz	Frequency.
	Phase	deg	Phase.
	Imax	A	Input current corresponding to 100% duty.

Current controlled Pulse-Width Modulator. Output current is a pulse signal of frequency "F" shifted by "Phase". Input current Iin is sampled at the beginning of each cycle of the signal, and width of the output pulse during this cycle is calculated according to the equation:

width = 1/F * (Iin / Imax)

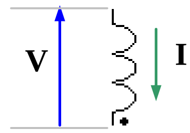
or

duty = 100% * (Iin / Imax);

If the width is equal or less than zero, a short "On" pulse with the width equal to the minimum calculation step at that moment will be generated. If the width is equal or greater than period of frequency "F", a short "Off" pulse at the end of the period will be generated. Due to that, the frequency of the output signal is always "F". Such a signal can be, for instance, divided by D-trigger to create a signal with duty cycle less than 50%.

I	Current controlled current source		I/I_CCCS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

<table>
<tr><td>L</td><td>Inductor</td><td rowspan="2">Models</td><td rowspan="2">L
PWL
Function
SubCir</td><td rowspan="2">Traces</td><td rowspan="2">

P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
</table>

L	Inductor		L/L_Inductor_L.nl5
Model	Parameter	Units	Description
L	L	H	Inductance
	IC	A	Initial condition: current. Leave blank if IC not defined.

Linear inductor. V = L*dI/dt.

When calculating DC operating point, if "IC" is defined, inductor is replaced with current source equal to IC. If "IC" is not defined (blank), inductor is temporarily replaced by short circuit, DC operating point calculated, and then the current through short circuit is assigned to the inductor as its initial current.

L	Inductor		L/L_Inductor_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, L(I)
	IC	A	Initial condition: current. Leave blank if IC not defined.

Piece-wise linear capacitor. "pwl" string defines inductance as a function of current. See *Working with PWL* chapter for details.

When calculating DC operating point, if "IC" is defined, inductor is replaced with current source equal to IC. If "IC" is not defined (blank), inductor is temporarily replaced by short circuit, DC operating point calculated, and then the current through short circuit is assigned to the inductor as its initial current.

L	Inductor		L/L_Inductor_Function.nl5
Model	Parameter	Units	Description
Function	Z(s)	Ohm	Impedance as a function of ***s*** parameter.

Impedance function in *s* domain. For transient, constant impedance Z(0) is used. For linearized AC analysis, complex impedance Z(s) is used. The following variables can be used in the function:

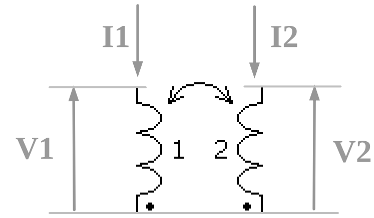
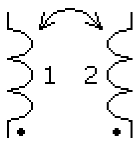
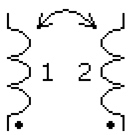
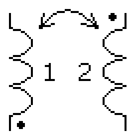
- ***f*** – current AC frequency, Hz
- ***w*** – angular AC frequency, ***w = 2πf*** .
- ***s*** or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:

Z(s) = 3n*s + 0.5 - 3 nH inductor in series with 0.5 Ohm resistor.
Z(s) = (1u*s)||(1.0/10p/s) - 1 mkH inductor in parallel with 10 pF capacitor.

L	Inductor		L/L_Inductor_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

L	Coupled inductors	Models	L	Traces	
					
Views	 				

L	Coupled inductors		L/L_CoupledInductors_L.nl5
Model	Parameter	Units	Description
L	L1	H	L1 inductance
	L2	H	L2 inductance
	K		Coupling coefficient (-1…1)
	IC1	A	L1 initial condition: current. Leave blank if IC1 not defined.
	IC2	A	L2 initial condition: current. Leave blank if IC2 not defined.

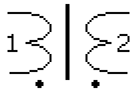
Coupled linear inductors.

V1 = L1*dI1/dt + M*dI2/dt
V2 = M*dI1/dt + L2*dI2/dt

Where M = K*sqrt(L1*L2) is mutual inductance.

When calculating DC operating point, initial conditions IC1 and IC2 are independently applied to corresponding inductors L1 and L2, similar to how it is done for the component **L** (inductor).

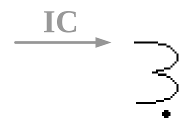
L Custom coupled inductors



Models

L
SubCir

Traces



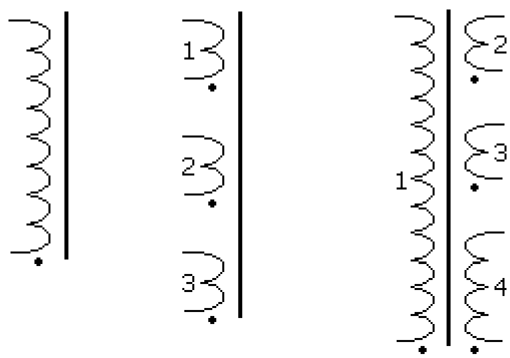
Views

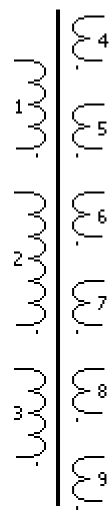
This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:

- height from 2 to 32,
- up to 9 windings (total) on both sides,
- arbitrary length of a winding.

Examples of Custom coupled inductors component:





L	Custom coupled inductors		L/L_CustomCoupledInductors_L.nl5
Model	Parameter	Units	Description
L	L1	H	L1 inductance
	…	H	…
	LN	H	LN inductance
	K12		L1-L2 coupling coefficient (-1…1)
	…		…
	K(N-1)N		L(N-1)-LN coupling coefficient (-1…1)
	IC1	A	L1 initial condition: current. Leave blank if IC1 not defined.
	…	A	…
	ICN	A	LN initial condition: current. Leave blank if ICN not defined.

Custom coupled inductors.

V1 = L1*dI1/dt + M12*dI2/dt + … + M1N*dIN/dt
V2 = M12*dI1/dt + L2*dI2/dt + … + M2N*dIN/dt
…
VN = M1N*dI1/dt + M2N*dI2/dt + … + LN*dIN/dt

Where Mij = Kij*sqrt(Li*Lj) is mutual inductance, Mij = Mji.

When calculating DC operating point, initial conditions ICN are independently applied to corresponding inductors LN, similar to how it is done for the component **L** (inductor).

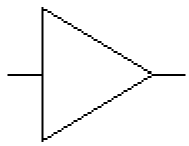
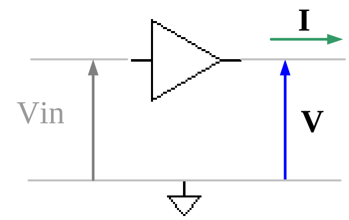
If only one winding is defined, a component behaves exactly as a linear inductor **L**.

Please be aware that coupling coefficients Kij should be properly specified within allowable range (-1…1) in order to represent a "physically-realizable" system. See NL5 website (http://nl5.sidelinesoft.com) and other public resources for more details.

If all coupling coefficients are equal to 1 (or -1), using Winding components W with one magnetizing inductor will give better performance and more stable solution.

L	Custom coupled inductors		L/L_CustomCoupledInductors_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	…		…
	PinN		Name of subcircuit label connected to pin N
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

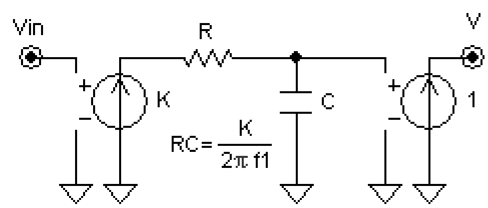
Subcircuit. See *Working with Subcircuits* chapter for details.

O	Amplifier	Models		Traces	
			Linear OpAmp Comparator Function PWL SubCir		 P = V • I

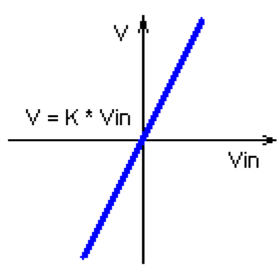
O	Amplifier		O/O_Amplifier_Linear.nl5
Model	Parameter	Units	Description
Linear	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	IC	V	Initial condition: output voltage.

Linear amplifier. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf").

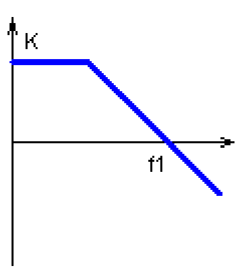
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



Static characteristic



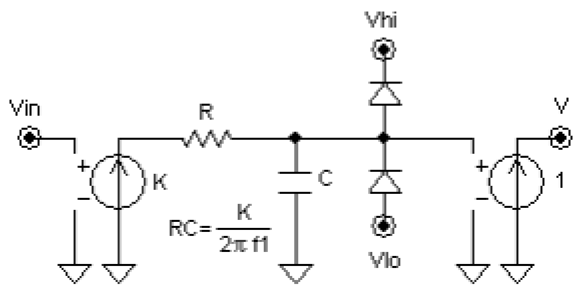
AC response

O	Amplifier		O/O_Amplifier_OpAmp.nl5
Model	Parameter	Units	Description
OpAmp	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	Vhi	V	Max output voltage.
	Vlo	V	Min output voltage.
	IC	V	Initial condition: output voltage.

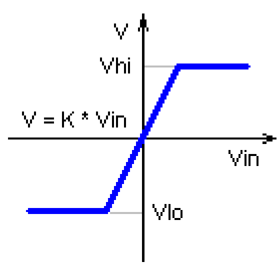
Linear amplifier with output limiter. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf"). Output voltage is limiting between "Vlo" and "Vhi".

When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.

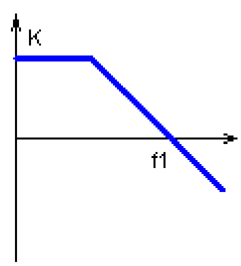
Please note: if both "K" and "f1" are set to infinity, the model may experience convergence problem. Use **Comparator** model instead.



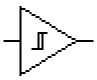
Equivalent schematic



Static characteristic



AC response

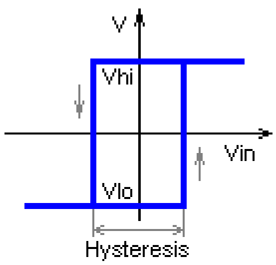
O	Amplifier		O/O_Amplifier_Comparator.nl5
Model	Parameter	Units	Description
Comparator 	Hysteresis	V	Hysteresis
	Vhi	V	Max output voltage.
	Vlo	V	Min output voltage.
	Delay	s	Output delay.
	IC		Initial condition: Low/High.

Comparator with hysteresis. Comparator output is set to "Vhi" or "Vlo" using following rules:

Vin > Hysteresis/2 . . . : V = Vhi
Vin < - Hysteresis/2 . . : V = Vlo
Otherwise . . . .: V = previous state

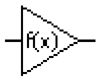
The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output.

When calculating DC operating point comparator output is set to "Vlo" or to "Vhi", according to selected "IC".



Static characteristic

O	Amplifier		O/O_Amplifier_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.



Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – input voltage Vin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

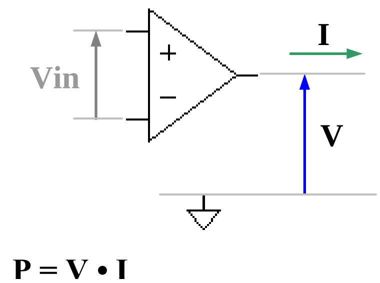
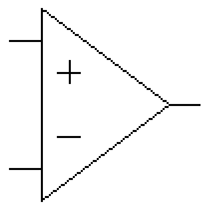
Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage “IC”.

O	Amplifier		O/O_Amplifier_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Vin)
	Piece-wise linear amplifier. Amplifier gain K is defined by "pwl" string as a function of input voltage Vin. See *Working with PWL* chapter for details.		

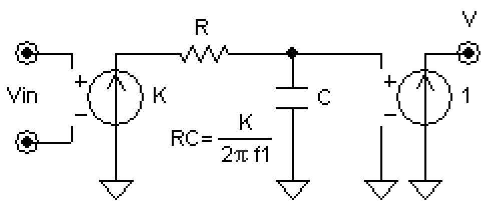
O	Amplifier		O/O_Amplifier_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

O	Differential amplifier	Models	Linear OpAmp Comparator Function PWL SubCir	Traces	
					

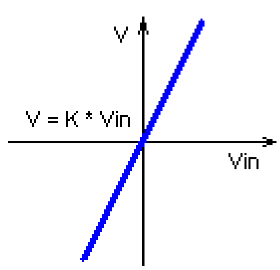
O	Differential amplifier		O/O_DiffAmp_Linear.nl5
Model	Parameter	Units	Description
Linear	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	IC	V	Initial condition: output voltage.

Linear differential amplifier. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf").

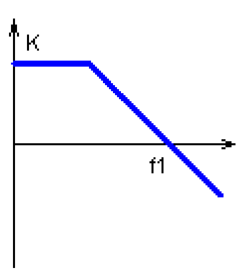
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



Static characteristic



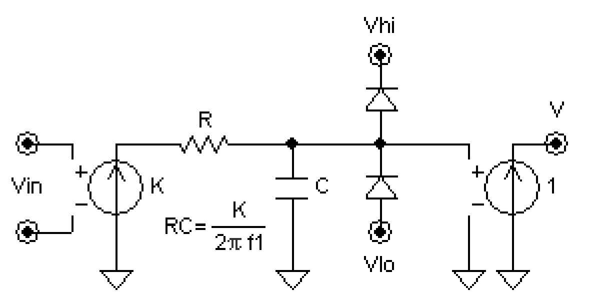
AC response

O	Differential amplifier		O/O_DiffAmp_OpAmp.nl5
Model	Parameter	Units	Description
OpAmp	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	Vhi	V	Max output voltage.
	Vlo	V	Min output voltage.
	IC	V	Initial condition: output voltage.

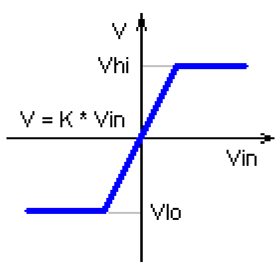
Linear amplifier with output limiter. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf"). Output voltage is limiting between "Vlo" and "Vhi".

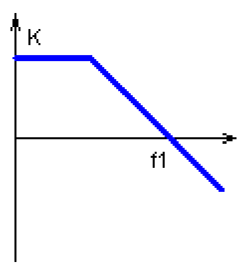
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.

Please note: if both "K" and "f1" are set to infinity, the model may experience convergence problem. Use **Comparator** model instead.

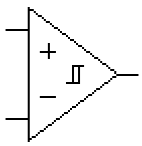


Equivalent schematic





Static characteristic **AC response**

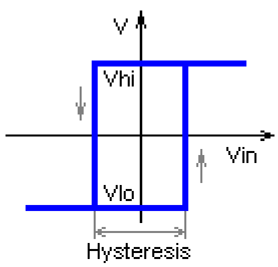
O	Differential amplifier		O/O_DiffAmp_Comparator.nl5
Model	Parameter	Units	Description
Comparator 	Hysteresis	V	Hysteresis
	Vhi	V	Max output voltage.
	Vlo	V	Min output voltage.
	Delay	s	Output delay.
	IC		Initial condition: Low/High.

Comparator with hysteresis. Comparator output is set to "Vhi" or "Vlo" using following rules:

Vin > Hysteresis/2. . . . : V = Vhi
Vin < - Hysteresis/2. . . : V = Vlo
Otherwise . . . . : V = previous state

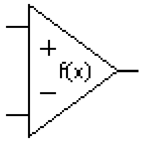
The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output.

When calculating DC operating point comparator output is set to "Vlo" or to "Vhi", according to selected "IC".



Static characteristic

O	Differential amplifier		O/O_DiffAmp_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.



Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – input voltage Vin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

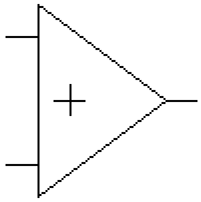
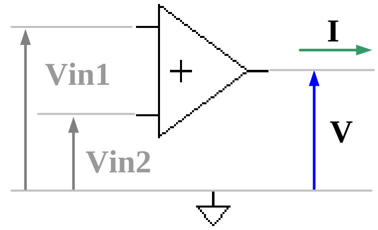
Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

O	Differential amplifier		O/O_DiffAmp_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Vin)
	Piece-wise linear amplifier. Amplifier gain K is defined by "pwl" string as a function of input voltage Vin. See *Working with PWL* chapter for details.		

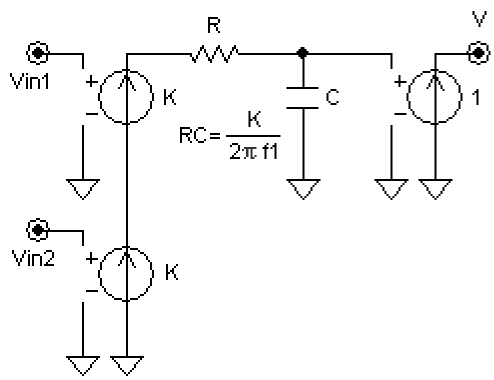
O	Differential amplifier		O/O_DiffAmp_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

O	Summing amplifier	Models	Linear OpAmp Function PWL SubCir	Traces	
					 **P = V • I**

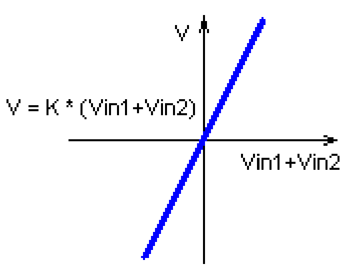
O	Summing amplifier		O/O_SumAmp_Linear.nl5
Model	Parameter	Units	Description
Linear	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	IC	V	Initial condition: output voltage.

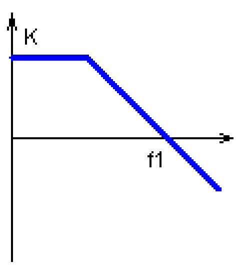
Linear summing amplifier. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf").

When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



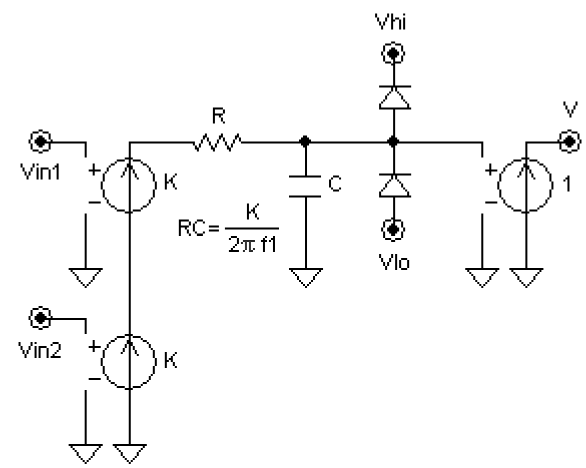


Static characteristic **AC response**

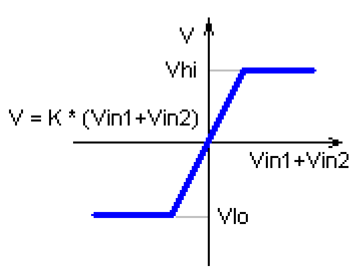
O	Summing amplifier		O/O_SumAmp_OpAmp.nl5
Model	Parameter	Units	Description
OpAmp	K	V/V	Gain
	f1	Hz	Unit gain frequency.
	Vhi	V	Max output voltage.
	Vlo	V	Min output voltage.
	IC	V	Initial condition: output voltage.

Linear amplifier with output limiter. "K" is open loop gain. Frequency response consists of one pole, "f1" is unit gain frequency. "K" and "f1" can be set to infinity ("inf"). Output voltage is limiting between "Vlo" and "Vhi".

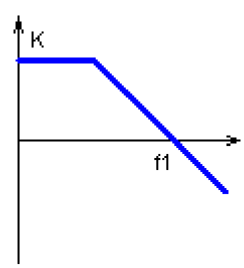
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, amplifier output is set to specified output voltage "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic

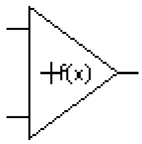


Static characteristic



AC response

O	Summing amplifier		O/O_SumAmp_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.



Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – voltage Vin1+Vin2
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

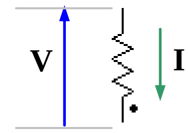
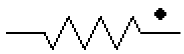
Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

O	Summing amplifier		O/O_SumAmp_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Vin1+Vin2)
	Piece-wise linear amplifier. Amplifier gain K is defined by "pwl" string as a function of sum of input voltages Vin1+Vin2. See *Working with PWL* chapter for details.		

O	Summing amplifier		O/O_SumAmp_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

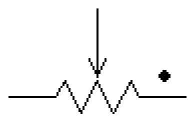
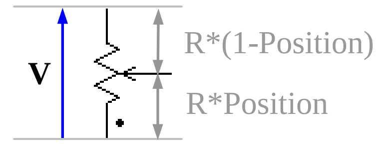
R	Resistor	Models	R PWL Function SubCir	Traces	 P = V • I
					

R	Resistor		R/R_Resistor_R.nl5
Model	Parameter	Units	Description
R	R	Ohm	Resistance
	Linear resistor. V = R*I.		

R	Resistor		R/R_Resistor_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, R(V)
	Piece-wise linear resistor. "pwl" string defines resistance as a function of voltage. See *Working with PWL* chapter for details.		

R	Resistor		R/R_Resistor_Function.nl5
Model	Parameter	Units	Description
Function	Z(s)	Ohm	Impedance as a function of ***s*** parameter.
	Impedance function in *s* domain. For transient, constant impedance Z(0) is used. For linearized AC analysis, complex impedance Z(s) is used. The following variables can be used in the function: ***f*** – current AC frequency, Hz ***w*** – angular AC frequency, ***w = 2πf*** . ***s*** or ***p*** – Laplace parameter, ***s = p = j*2πf***. Example: Z(s) = 10+3n*s - 10 Ohm resistor in series with 3 nH inductor. Z(s) = 10||(1.0/3p/s) - 10 Ohm resistor in parallel with 3 pF capacitor.		

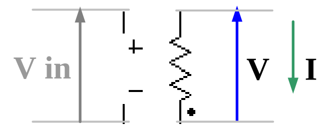
R	Resistor		R/R_Resistor_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

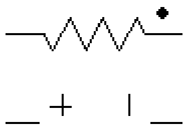
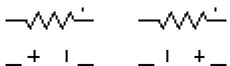
R	Potentiometer	Models	Potentiometer	Traces
				

R	Potentiometer		R/R_Potentiometer_Potentiometer.nl5
Model	Parameter	Units	Description
Potentiometer	R	Ohm	Resistance
	Position		Position of the wiper (0…1)

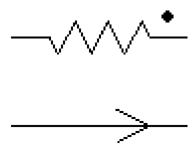
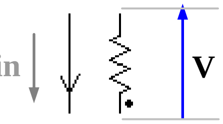
Potentiometer. Position of the wiper is referenced to the terminal with dot:

0 – wiper is connected to the terminal with dot
1 – wiper is connected to another terminal.

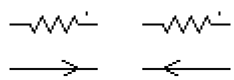
<table>
<tr><td>R</td><td>Voltage controlled resistor</td><td rowspan="2">Models</td><td rowspan="2">PWL</td><td rowspan="2">Traces</td><td rowspan="2">

P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
<tr><td>Views</td><td colspan="5"></td></tr>
</table>

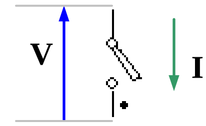
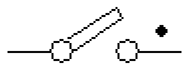
<table>
<tr><td>R</td><td colspan="2">Voltage controlled resistor</td><td>R/R_VCR_PWL.nl5</td></tr>
<tr><td>Model</td><td>Parameter</td><td>Units</td><td>Description</td></tr>
<tr><td rowspan="2">PWL</td><td>pwl</td><td></td><td>Comma-separated string, R(V)</td></tr>
<tr><td colspan="3">Piece-wise linear voltage controlled resistor. “pwl” string defines resistance as a function of control voltage Vin. See Working with PWL chapter for details.</td></tr>
</table>

R	Current controlled resistor	Models	Traces
		PWL	 $P = V \cdot I$

Views



R	Current controlled resistor		R/R_CCR_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, R(I)
	Piece-wise linear current controlled resistor. "pwl" string defines resistance as a function of control current Iin. See *Working with PWL* chapter for details.		

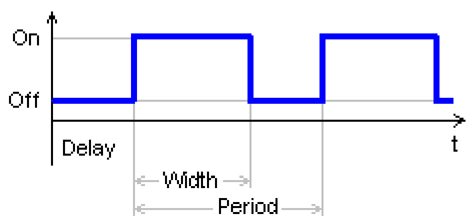
S	Switch	Models	Switch Pulse Step List File SubCir	Traces	 **P = V • I**
					

S	Switch		S/S_Switch_Switch.nl5
Model	Parameter	Units	Description
Switch	Switch		Switch state: Off/On.
	Switch. Off – open switch, infinite resistance. On – closed switch, zero resistance.		

S	Switch		S/S_Switch_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Period	s	Period.
	Width	s	Pulse width.
	Delay	s	Delay before first pulse starts.
	Active		Active switch state: Off/On.

Pulse switch. Switching starts after "Delay" time. Switch is in active state during "Width" time.

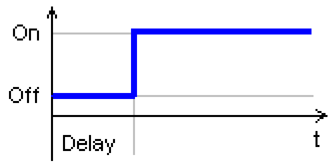
The following switching diagram is shown for "Active" = On:



S	Switch		S/S_Switch_Step.nl5
Model	Parameter	Units	Description
Step	Delay	s	Delay before active state.
	Active		Active switch state: Off/On.

Step switch. Switch is in active state after "Delay" time.

The following switching diagram is shown for "Active" = On:



S	Switch		V/S_Switch_List.nl5
Model	Parameter	Units	Description
List	List		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

List switch. Switching sequence is defined in the "List" parameter in the csv ("comma-separated values") format, as follows:

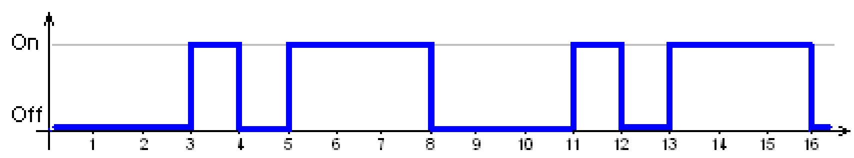
t0,s0,t1,s1,...,tn,sn

s0…sn defines switch state: positive number corresponds to On state, zero or negative number - Off state. If t<t0, switch is in s0 state. At t0 switch is set to s0 state. At t1 switch is set to s1 state, and so on. At t>tn, switch remains in sn state if "Cycle" parameter is set to "No", otherwise states sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
List = 0,0,3,1,4,0,5,1,8,0
```

The following switching diagram is shown for "Cycle" = Yes, "Delay" = 0:



S	Switch		V/S_Switch_File.nl5
Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

File switch. Switching sequence is defined in the text file. "File" parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Switching sequence is defined in the csv ("comma-separated values") format, as follows:

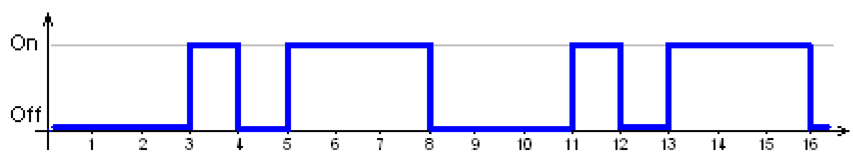
```
<if first line does not start with a number, it is ignored>
t0,s0
t1,s1
.....
tn,sn
```

s0…sn defines switch state: positive number corresponds to On state, zero or negative number - Off state. If t<t0, switch is in s0 state. At t0 switch is set to s0 state. At t1 switch is set to s1 state, and so on. At t>tn, switch remains in sn state if "Cycle" parameter is set to "No", otherwise states sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

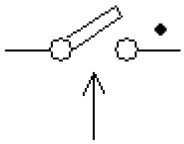
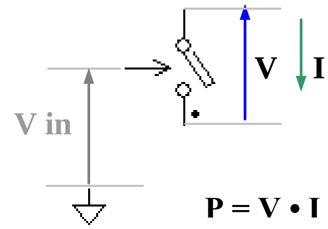
```
0,0
3,1
4,0
5,1
8,0
```

The following switching diagram is shown for "Cycle" = Yes, "Delay" = 0:



S	Switch		S/S_Switch_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

S	Logic controlled switch	Models	Switch Pulse Steps SubCir	Traces	
					

S	Logic controlled switch		S/S_LCS_Switch.nl5
Model	Parameter	Units	Description
Switch	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Logic controlled switch. Switch is set to active or non-active state using following rules:

Vin > logical threshold . . . : active
Vin < logical threshold . . . : non-active

To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point switch is set to the state defined in “IC”.

S	Logic controlled switch		S/S_LCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Active		Active state: Off/On.

One-shot pulse generator. When increasing input voltage Vin crosses logical threshold, switch is set to active state for “Width” time interval. If increasing Vin crosses logical threshold value while switch is in active state, the pulse is restarted.

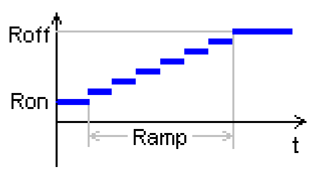
To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

S	Logic controlled switch		S/S_LCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

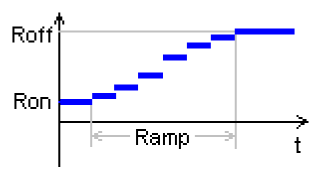
Switch with resistance ramping. When increasing input voltage Vin crosses logical threshold, switch resistance starts ramping from "Roff" to "Ron". When decreasing input voltage Vin crosses logical threshold, switch resistance starts ramping from "Ron" to "Roff".

Resistance is changing during "Ramp" time interval, with number of steps specified by "Steps" parameter. If "Steps" = 0, resistance is changed instantly.

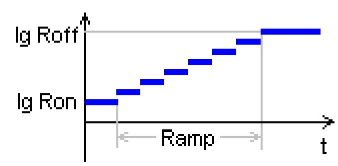
"Slope" parameter specifies how resistance is changing during he ramp. The following slope types are available ("Steps" = 6):



Linear



Cos



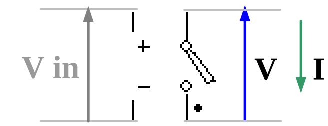
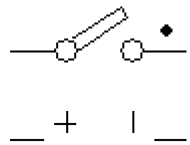
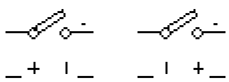
Log

To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point switch is set to the state specified in "IC".

S	Logic controlled switch		S/S_LCS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

S	Voltage controlled switch	Models	Switch Pulse Steps SubCir	Traces	 P = V • I
	 				
Views	 				

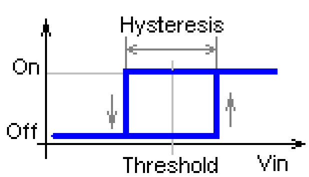
S	Voltage controlled switch		S/S_VCS_Switch.nl5
Model	**Parameter**	**Units**	**Description**
Switch	Threshold	V	Voltage threshold.
	Hysteresis	V	Hysteresis.
	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Voltage controlled switch. Switch is set to active or non-active state using following rules:

Vin > Threshold + Hysteresis/2. . . : active
Vin < Threshold - Hysteresis/2 . . . : non-active
Otherwise . . . . : previous state

When calculating DC operating point switch is set to the state defined in "IC".

The following is switching diagram for "Active" = On:



S	Voltage controlled switch		S/S_VCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	V	Voltage threshold.
	Active		Active state: Off/On.

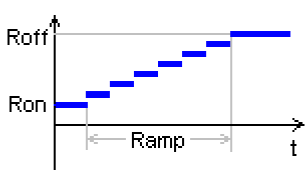
One-shot pulse generator. When increasing input voltage Vin crosses "Threshold" value, switch is set to active state for "Width" time interval. If increasing Vin crosses "Threshold" value while switch is in active state, the pulse is restarted.

S	Voltage controlled switch		S/S_VCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Threshold	V	Voltage threshold.
	Hysteresis	V	Hysteresis.
	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

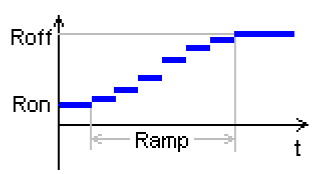
Switch with resistance ramping. When increasing input voltage Vin crosses "Threshold" plus "Hysteresis"/2 value, switch resistance starts ramping from "Roff" to "Ron". When decreasing input voltage Vin crosses "Threshold" minus "Hysteresis"/2 value, switch resistance starts ramping from "Ron" to "Roff".

Resistance is changing during "Ramp" time interval, with number of steps specified by "Steps" parameter. If "Steps" = 0, resistance is changed instantly.

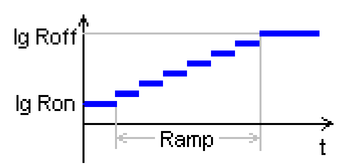
"Slope" parameter specifies how resistance is changing during he ramp. The following slope types are available ("Steps" = 6):



Linear



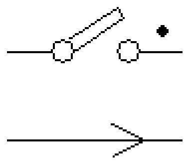
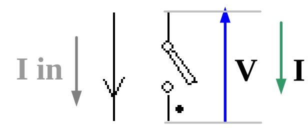
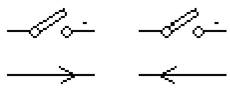
Cos



Log

When calculating DC operating point switch is set to the state specified in "IC".

S	Voltage controlled switch		S/S_VCS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

S	Current controlled switch	Models	Traces
		Switch Pulse Steps SubCir	 P = V • I
Views			

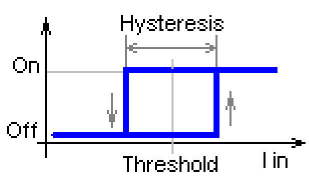
S	Current controlled switch		S/S_CCS_Switch.nl5
Model	**Parameter**	**Units**	**Description**
Switch	Threshold	A	Current threshold.
	Hysteresis	A	Hysteresis.
	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Current controlled switch. Switch is set to active or non-active state using following rules:

Iin > Threshold + Hysteresis/2. . . : active
Iin < Threshold - Hysteresis/2 . . . : non-active
Otherwise . . . . : previous state

When calculating DC operating point switch is set to the state defined in "IC".

The following is switching diagram for "Active" = On:



S	Current controlled switch		S/S_CCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	A	Current threshold.
	Active		Active state: Off/On.

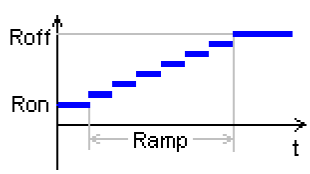
One-shot pulse generator. When increasing input current Iin crosses “Threshold” value, switch is set to active state for “Width” time interval. If increasing Iin crosses “Threshold” value while switch is in active state, the pulse is restarted.

S	Current controlled switch		S/S_CCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Threshold	A	Current threshold.
	Hysteresis	A	Hysteresis.
	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

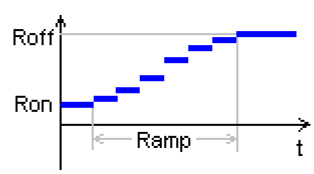
Switch with resistance ramping. When increasing input current Iin crosses “Threshold” plus “Hysteresis”/2 value, switch resistance starts ramping from “Roff” to “Ron”. When decreasing input current Iin crosses “Threshold” minus “Hysteresis”/2 value, switch resistance starts ramping from “Ron” to “Roff”.

Resistance is changing during “Ramp” time interval, with number of steps specified by “Steps” parameter. If “Steps” = 0, resistance is changed instantly.

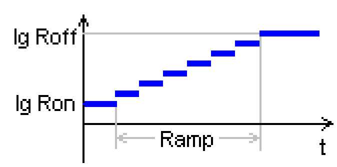
“Slope” parameter specifies how resistance is changing during he ramp. The following slope types are available (“Steps” = 6):



Linear



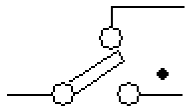
Cos



Log

When calculating DC operating point switch is set to the state specified in “IC”.

S	Current controlled switch		S/S_CCS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

S	SPDT switch	Models	Switch Pulse Step List File SubCir	Traces	
					

S	SPDT switch		S/S_SPDT_Switch_Switch.nl5
Model	Parameter	Units	Description
Switch	Switch		Switch state: Off/On.

SPDT (single pole, double throw) switch.

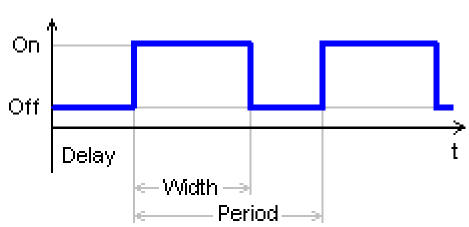
Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

S	SPDT switch		S/S_ SPDT_Switch_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Period	s	Period.
	Width	s	Pulse width.
	Delay	s	Delay before first pulse starts.
	Active		Active switch state: Off/On.

SPDT (single pole, double throw) pulse switch.

Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switching starts after "Delay" time. Switch is in active state during "Width" time. The following switching diagram shows state of "common to dotted pin" path for "Active" = On. "Common to another pin" always has an opposite state.

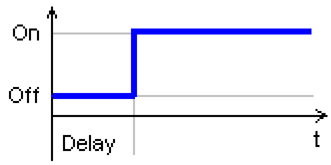


S	SPDT switch		S/S_ SPDT_Switch_Step.nl5
Model	Parameter	Units	Description
Step	Delay	s	Delay before active state.
	Active	s	Active switch state: Off/On.

SPDT (single pole, double throw) step switch.

Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switch is in active state after "Delay" time. The following switching diagram shows state of "common to dotted pin" path for "Active" = On. "Common to another pin" always has an opposite state.



S	SPDT switch		V/S_ SPDT_Switch_List.nl5
Model	Parameter	Units	Description
List	List		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

SPDT (single pole, double throw) list switch.

Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switching sequence is defined in the "List" parameter in the csv ("comma-separated values") format, as follows:

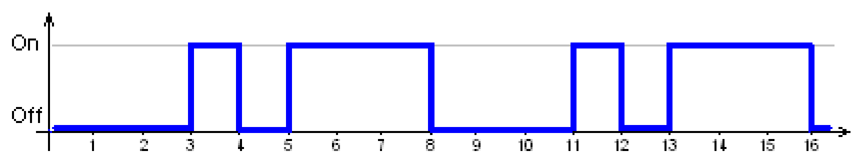
t0,s0,t1,s1,...,tn,sn

s0…sn defines switch state: positive number corresponds to On state, zero or negative number - Off state. If t<t0, switch is in s0 state. At t0 switch is set to s0 state. At t1 switch is set to s1 state, and so on. At t>tn, switch remains in sn state if "Cycle" parameter is set to "No", otherwise states sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
List = 0,0,3,1,4,0,5,1,8,0
```

The following switching diagram shows state of "common to dotted pin" path for "Cycle" = Yes, "Delay" = 0. "Common to another pin" always has an opposite state.



S	SPDT switch		V/S_ SPDT_Switch_File.nl5
Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

SPDT (single pole, double throw) file switch.

Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switching sequence is defined in the text file. "File" parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Switching sequence is defined in the csv ("comma-separated values") format, as follows:

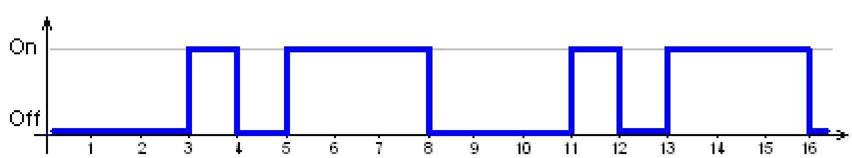
```
<if first line does not start with a number, it is ignored>
t0,s0
t1,s1
.....
tn,sn
```

s0…sn defines switch state: positive number corresponds to On state, zero or negative number - Off state. If t<t0, switch is in s0 state. At t0 switch is set to s0 state. At t1 switch is set to s1 state, and so on. At t>tn, switch remains in sn state if "Cycle" parameter is set to "No", otherwise states sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

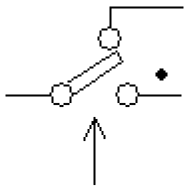
Example:

```
0,0
3,1
4,0
5,1
8,0
```

The following switching diagram shows state of "common to dotted pin" path for "Cycle" = Yes, "Delay" = 0. "Common to another pin" always has an opposite state.



S	SPDT switch		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

S	SPDT logic controlled switch	Models	Switch Pulse Steps SubCir	Traces	
					

S	SPDT logic controlled switch		S/S_SPDT_LCS_Switch.nl5
Model	Parameter	Units	Description
Switch	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Logic controlled switch.

Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switch is set to active or non-active state using following rules:

Vin > logical threshold . . . : active
Vin < logical threshold . . . : non-active

To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point switch is set to the state defined in "IC".

S	SPDT logic controlled switch		S/S _SPDT_LCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Active		Active state: Off/On.
	One-shot pulse generator. Off state: "common to pin with dot" - open, "common to another pin" - closed. On state: "common to pin with dot" - closed, "common to another pin" - open. Open state has infinite resistance, closed state has zero resistance. When increasing input voltage Vin crosses logical threshold, switch is set to active state for "Width" time interval. If increasing Vin crosses logical threshold value while switch is in active state, the pulse is restarted. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.		

S	SPDT logic controlled switch		S/S_SPDT_LCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

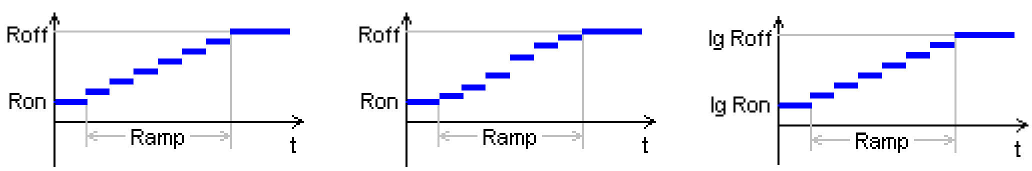
Switch with resistance ramping.

When increasing input voltage Vin crosses logical threshold, resistance of “common to pin with dot” path starts ramping from “Roff” to “Ron”, resistance of “common to another pin” path starts ramping from “Ron” to “Roff”

When decreasing input voltage Vin crosses logical threshold, resistance of “common to pin with dot” path starts ramping from “Ron” to “Roff”, resistance of “common to another pin” path starts ramping from “Roff” to “Ron”

Resistance is changing during “Ramp” time interval, with number of steps specified by “Steps” parameter. If “Steps” = 0, resistance is changed instantly.

“Slope” parameter specifies how resistance is changing during he ramp. The following slope types are available (“Steps” = 6):

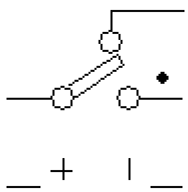


Linear **Cos** **Log**

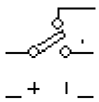
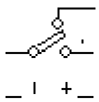
To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point switch is set to the state specified in “IC”.

S	SPDT logic controlled switch		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

S	SPDT voltage controlled switch	Models	Traces
		Switch Pulse Steps SubCir	

Views

S	SPDT voltage controlled switch		S/S_SPDT_VCS_Switch.nl5
Model	Parameter	Units	Description
Switch	Threshold	V	Voltage threshold.
	Hysteresis	V	Hysteresis.
	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Voltage controlled switch.

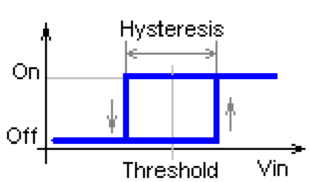
Off state: "common to pin with dot" - open, "common to another pin" - closed.
On state: "common to pin with dot" - closed, "common to another pin" - open.
Open state has infinite resistance, closed state has zero resistance.

Switch is set to active or non-active state using following rules:

Vin > Threshold + Hysteresis/2. . . : active
Vin < Threshold - Hysteresis/2 . . . : non-active
Otherwise . . . . : previous state

When calculating DC operating point switch is set to the state defined in "IC".

The following is switching diagram for "common to pin with dot" path, "Active" = On:
The following switching diagram shows state of "common to dotted pin" path for "Active" = On. "Common to another pin" always has an opposite state.



S	SPDT voltage controlled switch		S/S_SPDT _VCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	V	Voltage threshold.
	Active		Active state: Off/On.

One-shot pulse generator.

Off state: “common to pin with dot” - open, “common to another pin” - closed.
On state: “common to pin with dot” - closed, “common to another pin” - open.
Open state has infinite resistance, closed state has zero resistance.

When increasing input voltage Vin crosses “Threshold” value, switch is set to active state for “Width” time interval. If increasing Vin crosses “Threshold” value while switch is in active state, the pulse is restarted.

S	SPDT voltage controlled switch		S/S_SPDT _VCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Threshold	V	Voltage threshold.
	Hysteresis	V	Hysteresis.
	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

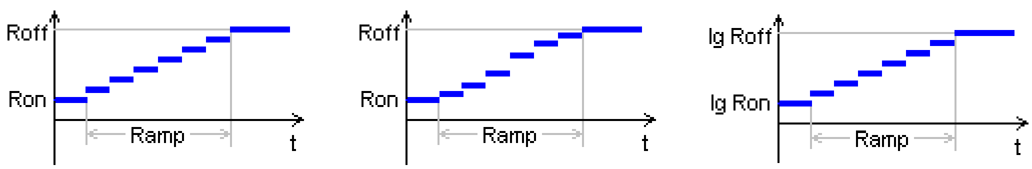
Switch with resistance ramping.

When increasing input voltage Vin crosses "Threshold" plus "Hysteresis"/2 value, resistance of "common to pin with dot" path starts ramping from "Roff" to "Ron", resistance of "common to another pin" path starts ramping from "Ron" to "Roff".

When decreasing input voltage Vin crosses "Threshold" minus "Hysteresis"/2 value, resistance of "common to pin with dot" path starts ramping from "Ron" to "Roff", resistance of "common to another pin" path starts ramping from "Roff" to "Ron"

Resistance is changing during "Ramp" time interval, with number of steps specified by "Steps" parameter. If "Steps" = 0, resistance is changed instantly.

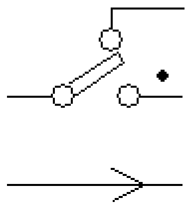
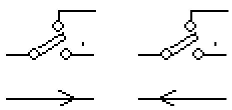
"Slope" parameter specifies how resistance is changing during he ramp. The following slope types are available ("Steps" = 6):



Linear **Cos** **Log**

When calculating DC operating point switch is set to the state specified in "IC".

S	SPDT voltage controlled switch		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

S	SPDT current controlled switch	Models	Switch Pulse Steps SubCir	Traces	
					
Views					

S	SPDT current controlled switch		S/S_SPDT _CCS_Switch.nl5
Model	Parameter	Units	Description
Switch	Threshold	A	Current threshold.
	Hysteresis	A	Hysteresis.
	Active		Active state: Off/On.
	IC		Initial condition: Off/On.

Current controlled switch.

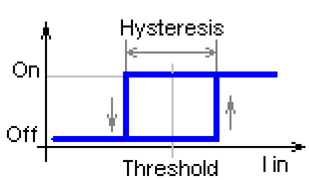
Off state: “common to pin with dot” - open, “common to another pin” - closed.
On state: “common to pin with dot” - closed, “common to another pin” - open.
Open state has infinite resistance, closed state has zero resistance.

Switch is set to active or non-active state using following rules:

Iin > Threshold + Hysteresis/2. . . : active
Iin < Threshold - Hysteresis/2 . . . : non-active
Otherwise . . . . : previous state

When calculating DC operating point switch is set to the state defined in “IC”.

The following is switching diagram for “common to pin with dot” path, “Active” = On:
The following switching diagram shows state of ”common to dotted pin” path for “Active” = On. “Common to another pin” always has an opposite state.



S	SPDT current controlled switch		S/S_SPDT _CCS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	A	Current threshold.
	Active		Active state: Off/On.
	One-shot pulse generator. Off state: "common to pin with dot" - open, "common to another pin" - closed. On state: "common to pin with dot" - closed, "common to another pin" - open. Open state has infinite resistance, closed state has zero resistance. When increasing input current Iin crosses "Threshold" value, switch is set to active state for "Width" time interval. If increasing Iin crosses "Threshold" value while switch is in active state, the pulse is restarted.		

S	SPDT current controlled switch		S/S_SPDT _CCS_Steps.nl5
Model	Parameter	Units	Description
Steps	Threshold	A	Current threshold.
	Hysteresis	A	Hysteresis.
	Roff	Ohm	Off state resistance.
	Ron	Ohm	On state resistance.
	Slope		Type of resistance change: Linear/Cos/Log.
	Ramp	s	Resistance ramp time.
	Steps		Number of resistance steps in the ramp.
	IC		Initial condition: Off/On.

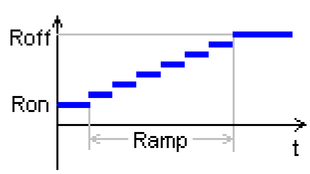
Switch with resistance ramping.

When increasing input current Iin crosses "Threshold" plus "Hysteresis"/2 value, resistance of "common to pin with dot" path starts ramping from "Roff" to "Ron", resistance of "common to another pin" path starts ramping from "Ron" to "Roff".

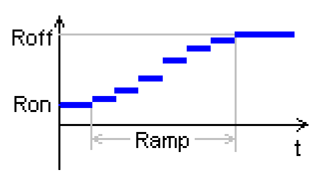
When decreasing input current Iin crosses "Threshold" minus "Hysteresis"/2 value, resistance of "common to pin with dot" path starts ramping from "Ron" to "Roff", resistance of "common to another pin" path starts ramping from "Roff" to "Ron"

Resistance is changing during "Ramp" time interval, with number of steps specified by "Steps" parameter. If "Steps" = 0, resistance is changed instantly.

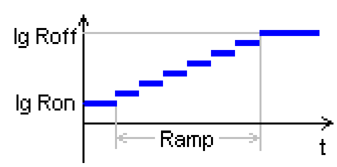
"Slope" parameter specifies how resistance is changing during he ramp. The following slope types are available ("Steps" = 6):



Linear



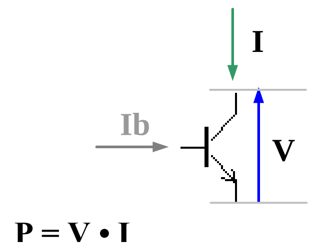
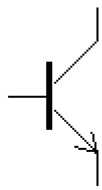
Cos



Log

When calculating DC operating point switch is set to the state specified in "IC".

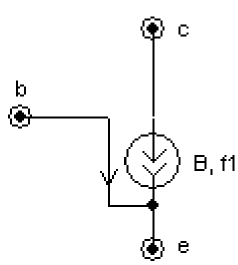
S	SPDT current controlled switch		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

T	NPN transistor	Models	Linear Switch Transistor SubCir	Traces	
					

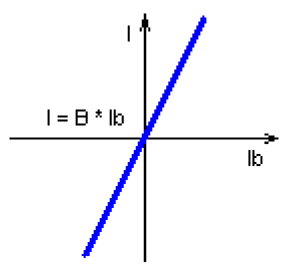
T	NPN transistor		T/T_NPN_Linear.nl5
Model	**Parameter**	**Units**	**Description**
Linear	B	A/A	Gain (beta)
	f1	Hz	Unit gain frequency.
	IC	A	Initial condition: collector current.

Linear BJT transistor. Current controlled current source with specified bandwidth. "B" is open loop gain (beta). Frequency response consists of one pole, "f1" is unit gain frequency. "B" and "f1" can be set to infinity ("inf").

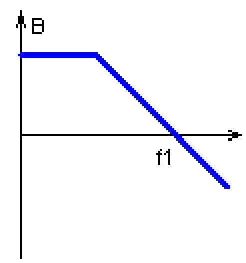
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, collector current is set to specified output current "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



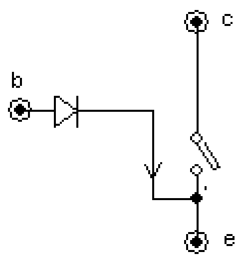
Static characteristic



AC response

<table>
<tr><td>T</td><td colspan="3">NPN transistor</td><td>T/T_NPN_Switch.nl5</td></tr>
<tr><td>Model</td><td>Parameter</td><td>Units</td><td colspan="2">Description</td></tr>
<tr><td rowspan="3">Switch</td><td>Vbe</td><td>V</td><td colspan="2">Forward voltage drop of base-emitter diode.</td></tr>
<tr><td>IC</td><td></td><td colspan="2">Initial condition of base-emitter diode: Off/On.</td></tr>
<tr><td colspan="4">BJT transistor switch. Current controlled switch with a base-emitter diode. Switch is closed if diode current is non-zero.

When calculating DC operating point the diode is set to the state specified in “IC”.



Equivalent schematic</td></tr>
</table>

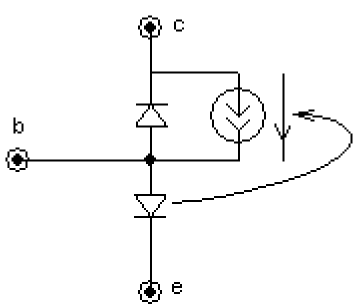
T	NPN transistor		T/T_NPN_Transistor.nl5
Model	Parameter	Units	Description
Transistor	B	A/A	Gain (beta)
	f1	Hz	Unit gain frequency.
	Vbe	V	Forward voltage drop of base-emitter diode.
	Vsat	V	Collector-emitter saturation voltage drop.
	IC	A	Initial condition: collector current.
	ICbe		Initial condition of base-emitter diode: Off/On.
	ICbc		Initial condition of base-collector diode: Off/On.

BJT transistor. Simplified Ebers-Moll BJT transistor model with saturation. It consists of two diodes (base-emitter and base-collector), and current source controlled by current through base-emitter diode with gain "alpha": $\alpha = \dfrac{\beta}{1+\beta}$

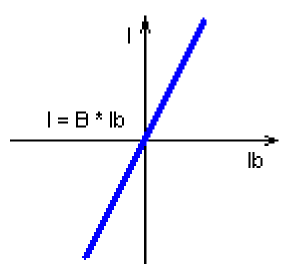
If collector-emitter voltage is higher than "Vsat", base-collector diode is open, transistor is not saturated, and behaves as "Linear" model (current controlled current source with specified bandwidth). "B" is open loop gain (beta). Low signal frequency response consists of one pole, "f1" is unit gain frequency. "B" and "f1" can be set to infinity ("inf").

If collector voltage drops below "Vsat", base-collector diode is closed, and transistor is saturated: collector-emitter voltage is equal to "Vsat".

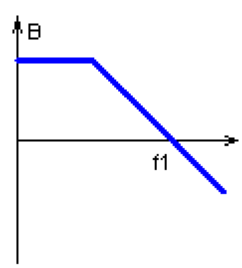
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, collector current is set to specified output current "IC". If "IC" is blank, static characteristic is used. Base-emitter diode is set to the state specified in "ICbe", Base-collector diode is set to the state specified in "ICbc".



Equivalent schematic

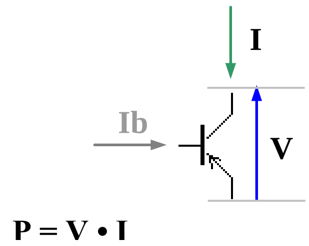
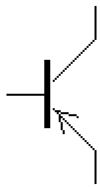


Non-saturated static characteristic



Low signal AC response

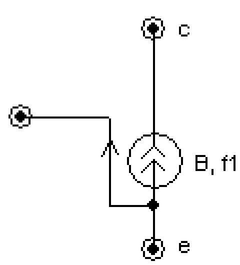
T	NPN transistor		T/T_NPN_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

T	PNP transistor	Models	Linear Switch Transistor SubCir	Traces	
					

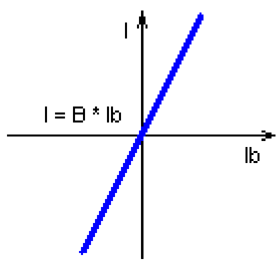
T	PNP transistor		T/T_PNP_Linear.nl5
Model	**Parameter**	**Units**	**Description**
Linear	B	A/A	Gain (beta)
	f1	Hz	Unit gain frequency.
	IC	A	Initial condition: collector current.

Linear BJT transistor. Current controlled current source with specified bandwidth. "B" is open loop gain (beta). Frequency response consists of one pole, "f1" is unit gain frequency. "B" and "f1" can be set to infinity ("inf").

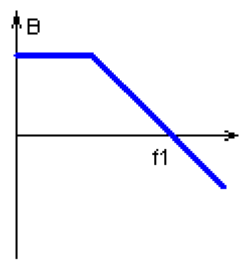
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, collector current is set to specified output current "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



Static characteristic

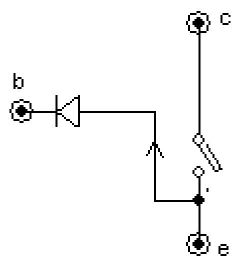


AC response

T	PNP transistor		T/T_PNP_Switch.nl5
Model	Parameter	Units	Description
Switch	Vbe	V	Forward voltage drop of base-emitter diode.
	IC		Initial condition of base-emitter diode: Off/On.

BJT transistor switch. Current controlled switch with a base-emitter diode. Switch is closed if diode current is non-zero.

When calculating DC operating point the diode is set to the state specified in "IC".



Equivalent schematic

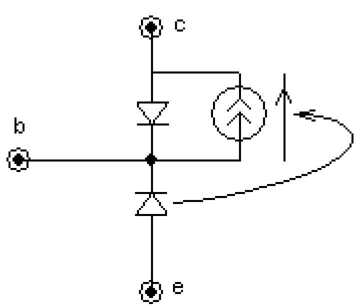
T	PNP transistor		T/T_PNP_Transistor.nl5
Model	Parameter	Units	Description
Transistor	B	A/A	Gain (beta)
	f1	Hz	Unit gain frequency.
	Vbe	V	Forward voltage drop of base-emitter diode.
	Vsat	V	Collector-emitter saturation voltage drop.
	IC	A	Initial condition: collector current.
	ICbe		Initial condition of base-emitter diode: Off/On.
	ICbc		Initial condition of base-collector diode: Off/On.

BJT transistor. Simplified Ebers-Moll BJT transistor model with saturation. It consists of two diodes (base-emitter and base-collector), and current source controlled by current through base-emitter diode with gain "alpha": $\alpha = \frac{\beta}{1+\beta}$

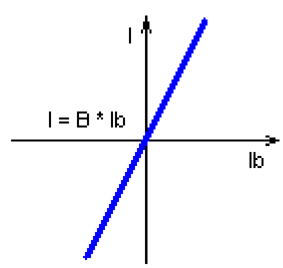
If collector-emitter voltage is negative and less than -"Vsat", base-collector diode is open, transistor is not saturated, and behaves as "Linear" model (current controlled current source with specified bandwidth). "B" is open loop gain (beta). Low signal frequency response consists of one pole, "f1" is unit gain frequency. "B" and "f1" can be set to infinity ("inf").

If collector voltage is higher than -"Vsat", base-collector diode is closed, and transistor is saturated: collector-emitter voltage is equal to -"Vsat".

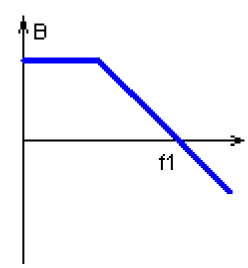
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, collector current is set to specified output current "IC". If "IC" is blank, static characteristic is used. Base-emitter diode is set to the state specified in "ICbe". Base-collector diode is set to the state specified in "ICbc".



Equivalent schematic

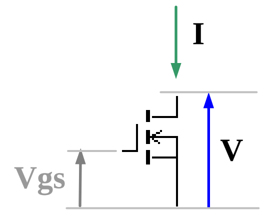
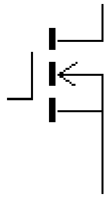


Non-saturated static characteristic



Low signal AC response

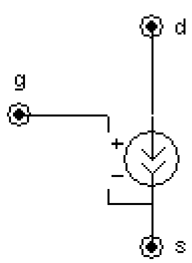
T	PNP transistor		T/T_PNP_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

T	N-FET	Models	Linear Switch FET SubCir	Traces	 P = V • I
					

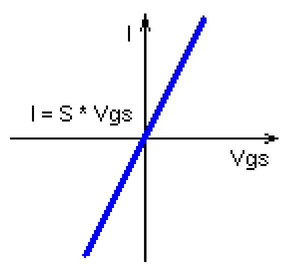
T	N-FET		T/T_NFET_Linear.nl5
Model	Parameter	Units	Description
Linear	S	A/V	Slope
	f1	Hz	Unit gain frequency.
	IC	A	Initial condition: drain current.

Linear FET transistor. Voltage controlled current source with specified bandwidth. "S" is open loop slope. Frequency response consists of one pole, "f1" is unit gain frequency. "S" and "f1" can be set to infinity ("inf").

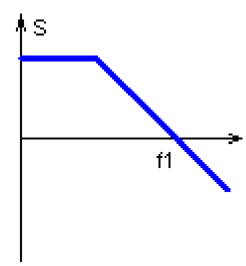
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, drain current is set to specified output current "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



Static characteristic

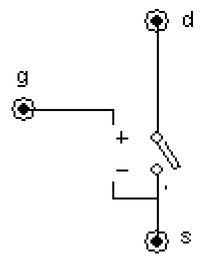


AC response

T	N-FET		T/T_NFET_Switch.nl5
Model	Parameter	Units	Description
Switch	Vth	V	Threshold.
	IC		Initial condition of the switch: Off/On.

FET switch. Voltage controlled switch. Switch is closed if gate-source voltage exceeds threshold "Vth".

When calculating DC operating point switch is set to the state specified in "IC".



Equivalent schematic

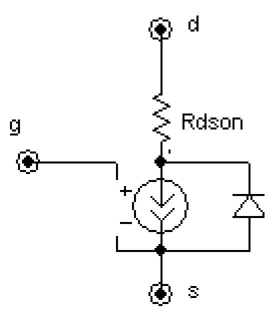
T	N-FET		T/T_NFET_FET.nl5
Model	Parameter	Units	Description
FET	S	A/V	Slope.
	Vth	V	Threshold.
	Rdson	Ohm	Rdson resistance.

FET transistor. The model has 3 modes of operation.

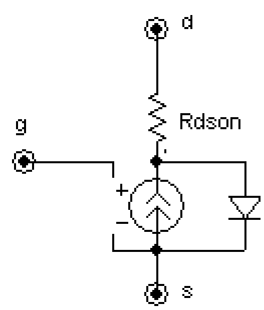
1. Vgs <= Vth: . . . . : I = 0 (open)
2. Vgs > Vth, Vds <= (Vgs - Vth) * S * Rdson . . : V = I * Rdson (resistor)
3. Vgs > Vth, Vds > (Vgs - Vth) * S * Rdson . . : I = (Vgs - Vth) * S (current source)

FET works similar for positive and negative drain-source voltage, current direction changes accordingly.

Please note: the diode in this model is not a body diode. If body diode is needed for accurate simulation, it should be added as an external component.



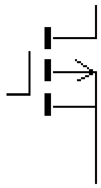
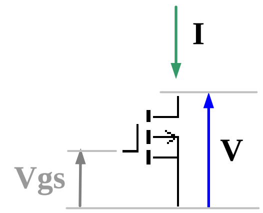
Equivalent schematic: Vds > 0



Equivalent schematic: Vds < 0

T	N-FET		T/T_NFET_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

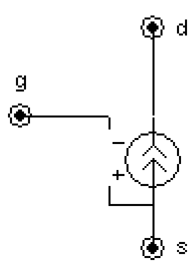
Subcircuit. See *Working with Subcircuits* chapter for details.

T	P-FET	Models	Linear Switch FET SubCir	Traces	
					 P = V • I

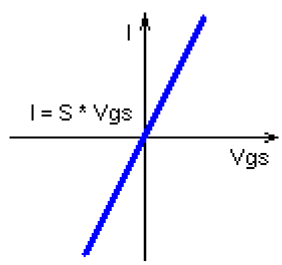
T	P-FET		T/T_PFET_Linear.nl5
Model	**Parameter**	**Units**	**Description**
Linear	S	A/V	Slope
	f1	Hz	Unit gain frequency.
	IC	A	Initial condition: drain current.

Linear FET transistor. Voltage controlled current source with specified bandwidth. "S" is open loop slope. Frequency response consists of one pole, "f1" is unit gain frequency. "S" and "f1" can be set to infinity ("inf").

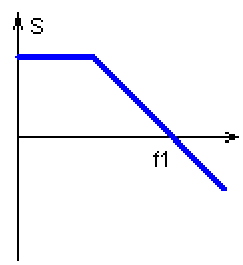
When calculating DC operating point, if "f1" is not infinity and "IC" is defined, drain current is set to specified output current "IC". If "IC" is blank, static characteristic is used.



Equivalent schematic



Static characteristic

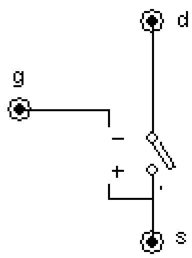


AC response

T	P-FET		T/T_PFET_Switch.nl5
Model	Parameter	Units	Description
Switch	Vth	V	Threshold.
	IC		Initial condition of the switch: Off/On.

FET switch. Voltage controlled switch. Switch is closed if gate-source voltage is less than threshold "Vth".

When calculating DC operating point switch is set to the state specified in "IC".



Equivalent schematic

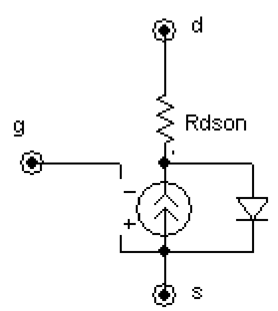
T	P-FET		T/T_PFET_FET.nl5
Model	Parameter	Units	Description
FET	S	A/V	Slope.
	Vth	V	Threshold.
	Rdson	Ohm	Rdson resistance.

FET transistor. The model has 3 modes of operation.

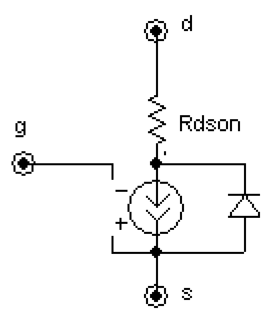
1. Vgs >= Vth: . . . . : I = 0 (open)
2. Vgs < Vth, Vds >= (Vgs - Vth) * S * Rdson . . : V = I * Rdson (resistor)
3. Vgs < Vth, Vds < (Vgs - Vth) * S * Rdson . . . : I = (Vgs - Vth) * S (current source)

FET works similar for positive and negative drain-source voltage, current direction changes accordingly.

Please note: the diode in this model is not a body diode. If body diode is needed for accurate simulation, it should be added as an external component.



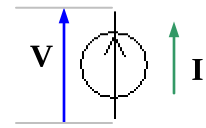
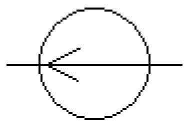
Equivalent schematic: Vds < 0



Equivalent schematic: Vds > 0

T	P-FET		T/T_PFET_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

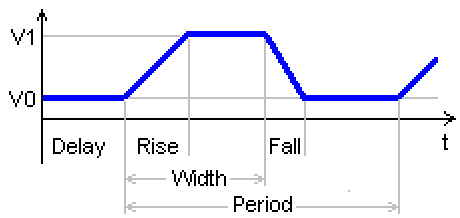
Subcircuit. See *Working with Subcircuits* chapter for details.

<table>
<tr><td>V</td><td>Voltage source</td><td rowspan="2">Models</td><td rowspan="2">V
Pulse
Step
Sin
PWL
Function
File
SubCir</td><td rowspan="2">Traces</td><td rowspan="2">
P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
</table>

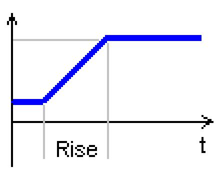
V	Voltage source		V/V_VS_V.nl5
Model	Parameter	Units	Description
V	V	V	Voltage.
	Constant voltage source. Voltage = "V".		

V	Voltage source		V/V_VS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	Period	s	Period.
	Width	s	Pulse width.
	Slope		Slope type: Linear/Cos/Exp
	Rise	s	Pulse rise length.
	Fall	s	Pulse fall length.
	Delay	s	Delay before first pulse starts.

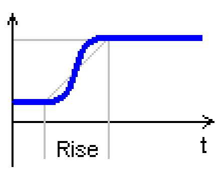
Pulse voltage source. Pulses start after "Delay" time. "Rise" time is included into "Width", "Fall" time is not included into "Width". Almost every parameter can be set to zero and infinity ("inf"), otherwise the error message will be displayed.



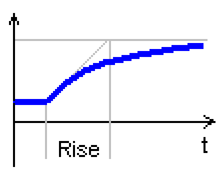
Slope type applies both to pulse rise and fall. The following slope types are available:



Linear



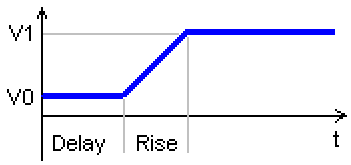
Cos (cosine)



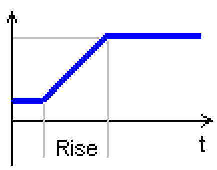
Exp (exponential)

V	Voltage source		V/V_VS_Step.nl5
Model	Parameter	Units	Description
Step	V1	V	Step On voltage.
	V0	V	Step Off voltage.
	Slope	s	Slope type: Linear/Cos/Exp
	Rise	s	Step rise length.
	Delay	s	Delay before step starts.

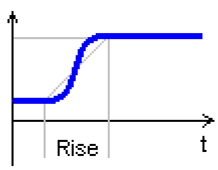
Step voltage source. Step starts after "Delay" time.



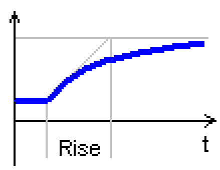
The following slope types are available:



Linear



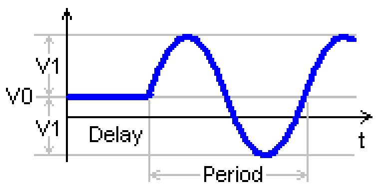
Cos (cosine)



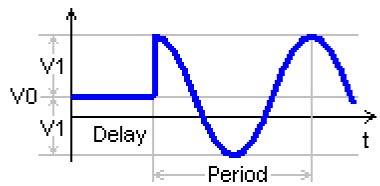
Exp (exponential)

V	Voltage source		V/V_VS_Sin.nl5
Model	Parameter	Units	Description
Sin	V1	V	Voltage amplitude.
	V0	V	Voltage baseline.
	Period	s	Period.
	Phase	deg	Phase.
	Delay	s	Delay before sine signal starts.

Sine voltage source. Sine signal starts after "Delay" time. "Phase" is sine phase in degrees at the moment when signal starts:

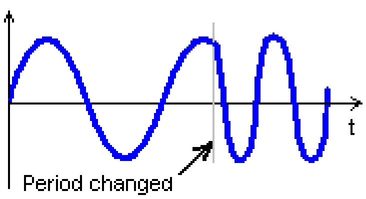


Phase = 0



Phase = 90

If transient is paused, sine period changed, then transient is continued, the phase of the signal remains continuous, providing smooth sine signal of variable frequency:



V	Voltage source		V/V_VS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Piece-wise linear voltage source. Signal is defined by "pwl" parameter in the csv ("comma-separated values") format, as follows:

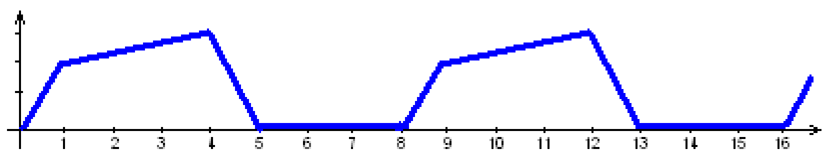
t0,V0,t1,V1,...,tn,Vn

where all t and V can be numerical values or expressions. If t<t0, signal is V0. If t0<t<t1, signal value is linearly interpolated between V0 and V1, etc. If t>tn, then signal value is Vn if "Cycle" parameter is set to "No", otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

```
pwl = 0,0,1,2,4,3,5,0,8,0
```

If "Cycle" = Yes, "Delay" = 0, the following voltage will be generated:



V	Voltage source		V/V_VS_Function.nl5
Model	Parameter	Units	Description
Function	F(t)	V	Function

Arbitrary function. F(t) defines voltage as a function of the following variables:

t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(t) is blank, voltage is zero.

Example:
F(t) = sin(t) * (1+cos(t*.01))
F(t) = V(R1) * I(R1)

Please note that **V**, **I**, and **P** variables are taken at previous calculation step. This may affect stability of the schematic with closed loop.

V	Voltage source		V/V_VS_File.nl5
Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Voltage source defined in the text file. "File" parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Signal is defined in following format:

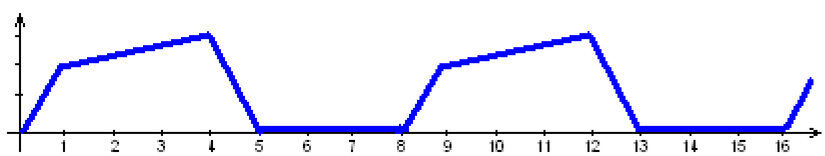
```
<if first line does not start with a number, it is ignored>
t0,V0
t1,V1
.....
tn,Vn
```

where all t and V can be numerical values or expressions. If t<t0, signal is V0. If t0<t<t1, signal value is linearly interpolated between V0 and V1, etc. If t>tn, then signal value is Vn if "Cycle" parameter is set to "No", otherwise signal defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

Example:

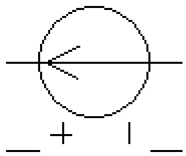
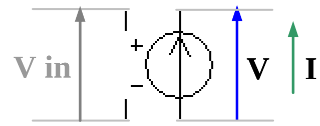
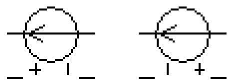
```
0,0
1,2
4,3
5,0
8,0
```

If "Cycle" = Yes, "Delay" = 0, the following voltage will be generated:



V	Voltage source		V/V_VS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

V	Voltage controlled voltage source	Models		Traces	
			Linear Function PWL VCO Pulse PWM SubCir		 P = V • I
Views					

V	Voltage controlled voltage source		V/V_VCVS_Linear.nl5
Model	Parameter	Units	Description
Linear	K	V/V	Gain
	Linear voltage controlled voltage source. V = K * Vin.		

V	Voltage controlled voltage source		V/V_VCVS_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.

Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – input voltage Vin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input voltage ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage “IC”.

V	Voltage controlled voltage source		V/V_VCVS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Vin)
	Piece-wise linear voltage controlled voltage source. Source gain K is defined by "pwl" string as a function of input voltage Vin. See *Working with PWL* chapter for details.		

V	Voltage controlled voltage source		V/V_VCVS_VCO.nl5
Model	Parameter	Units	Description
VCO	V1	V	Voltage amplitude (Sin), or Pulse On voltage (Pulse).
	V0	V	Voltage baseline (Sin), or Pulse Off voltage (Pulse).
	dFdV	Hz/V	Gain.
	Phase	deg	Phase.
	Type		Signal type: Sin/Square/Triangle/Sawtooth.
	Voltage controlled oscillator. Output voltage is a signal with frequency equal to: f(Hz) = dFdV * Vin. For Sine signal, "V0" is baseline, and "V1" is amplitude. For Square, Triangle, and Sawtooth signals, "V0" is Off level, "V1" is On level. "Phase" is additional phase of the signal, in degrees.		

V	Voltage controlled voltage source		V/V_VCVS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	V	Voltage threshold.
	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	One-shot pulse generator. When increasing input voltage Vin crosses "Threshold" value, voltage pulse of "Width" duration is generated. "V0" is pulse Off level, "V1" is pulse On level. If increasing Vin crosses "Threshold" value while pulse is generated, the pulse is restarted.		

V	Voltage controlled voltage source		I/I_VCVS_PWM.nl5
Model	Parameter	Units	Description
PWM	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	F	Hz	Frequency.
	Phase	deg	Phase.
	Vmax	V	Input voltage corresponding to 100% duty.

Voltage controlled Pulse-Width Modulator. Output voltage is a pulse signal of frequency "F" shifted by "Phase". Input voltage Vin is sampled at the beginning of each cycle of the signal, and width of the output pulse during this cycle is calculated according to the equation:

width = 1/F * (Vin / Vmax)

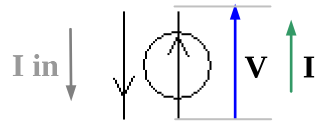
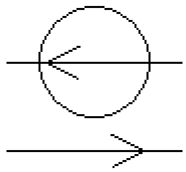
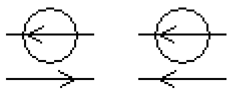
or

duty = 100% * (Vin / Vmax);

If the width is equal or less than zero, a short "On" pulse with the width equal to the minimum calculation step at that moment will be generated. If the width is equal or greater than period of frequency "F", a short "Off" pulse at the end of the period will be generated. Due to that, the frequency of the output signal is always "F". Such a signal can be, for instance, divided by D-trigger to create a signal with duty cycle less than 50%.

V	Voltage controlled voltage source		V/V_VCVS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

<table>
<tr><td>V</td><td>Current controlled voltage source</td><td rowspan="2">Models</td><td rowspan="2">Linear
Function
PWL
CCO
Pulse
PWM
SubCir</td><td rowspan="2">Traces</td><td rowspan="2">
P = V • I</td></tr>
<tr><td colspan="2"></td></tr>
<tr><td>Views</td><td colspan="5"></td></tr>
</table>

<table>
<tr><td>V</td><td colspan="3">Current controlled voltage source</td><td>V/V_CCVS_Linear.nl5</td></tr>
<tr><td>Model</td><td>Parameter</td><td>Units</td><td colspan="2">Description</td></tr>
<tr><td rowspan="2">Linear</td><td>K</td><td>V/A</td><td colspan="2">Gain</td></tr>
<tr><td colspan="4">Linear current controlled voltage source. V = K * Iin.</td></tr>
</table>

V	Current controlled voltage source		V/V_CCVS_Function.nl5
Model	Parameter	Units	Description
Function	F(x)	V	Output as function of the input.
	F(s)		AC transfer function in ***s*** domain.
	IC	V	Initial condition: output voltage.

Arbitrary function.

Transient analysis. F(x) defines output voltage as a function of the following variables:

x – input current Iin
t - current time
V(*name*) - voltage on the component ***name***
I(*name*) - current through the component ***name***
P(*name*) – power on the component ***name***

where ***name*** is the name of any component in the schematic. If F(x) is blank, output is zero. F(s) is ignored.

Example:
F(x) = x^3
F(x) = x * sin(t)
F(x) = P(r1)+P(r2)

Please note that input current ***x***, and variables **V**, **I**, and **P** are taken at previous calculation step. This may affect stability of the schematic with closed loop.

AC analysis. F(s) defines transfer function in ***s*** domain. Only operators and functions that support complex numbers can be used in this function. The following variables can be used in the function:
f – current AC frequency, Hz
w – angular AC frequency, ***w = 2πf*** .
s or ***p*** – Laplace parameter, ***s = p = j*2πf***.

Example:
F(s) = 1/(1+s)
F(s) = exp(-1mk*s)

F(s) is calculated at each frequency. If F(s) is blank, it is assumed to be 1. Also, if F(x) is not blank, it is linearized at DC operating point, and F(s) is multiplied by linearized gain.

When calculating DC operating point for transient or AC analysis, output is set to specified output voltage "IC".

V	Current controlled voltage source		V/V_CCVS_PWL.nl5
Model	Parameter	Units	Description
PWL	pwl		Comma-separated string, K(Iin)
	Piece-wise linear current controlled voltage source. Source gain K is defined by "pwl" string as a function of input current Iin. See *Working with PWL* chapter for details.		

V	Current controlled voltage source		V/V_CCVS_CCO.nl5
Model	Parameter	Units	Description
CCO	V1	V	Voltage amplitude (Sin), or Pulse On voltage (Pulse).
	V0	V	Voltage baseline (Sin), or Pulse Off voltage (Pulse).
	dFdI	Hz/A	Gain.
	Phase	deg	Phase.
	Type		Signal type: Sin/Square/Triangle/Sawtooth.
	Current controlled oscillator. Output voltage is a signal with frequency equal to: f(Hz) = dFdI * Iin. For Sine signal, "V0" is baseline, and "V1" is amplitude. For Square, Triangle, and Sawtooth signals, "V0" is Off level, "V1" is On level. "Phase" is additional phase of the signal, in degrees.		

V	Current controlled voltage source		V/V_CCVS_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Width	s	Pulse width.
	Threshold	A	Current threshold.
	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	One-shot pulse generator. When increasing input current Iin crosses "Threshold" value, voltage pulse of "Width" duration is generated. "V0" is pulse Off level, "V1" is pulse On level. If increasing Iin crosses "Threshold" value while pulse is generated, the pulse is restarted.		

V	Current controlled voltage source		I/I_CCVS_PWM.nl5
Model	Parameter	Units	Description
PWM	V1	V	Pulse On voltage.
	V0	V	Pulse Off voltage.
	F	Hz	Frequency.
	Phase	deg	Phase.
	Imax	A	Input current corresponding to 100% duty.

Current controlled Pulse-Width Modulator. Output voltage is a pulse signal of frequency "F" shifted by "Phase". Input current Iin is sampled at the beginning of each cycle of the signal, and width of the output pulse during this cycle is calculated according to the equation:

width = 1/F * (Iin / Imax)

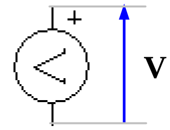
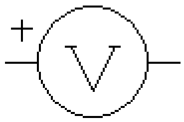
or

duty = 100% * (Iin / Imax);

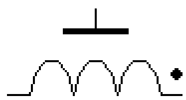
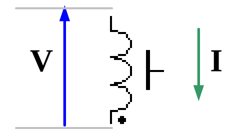
If the width is equal or less than zero, a short "On" pulse with the width equal to the minimum calculation step at that moment will be generated. If the width is equal or greater than period of frequency "F", a short "Off" pulse at the end of the period will be generated. Due to that, the frequency of the output signal is always "F". Such a signal can be, for instance, divided by D-trigger to create a signal with duty cycle less than 50%.

V	Current controlled voltage source		V/V_CCVS_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

V	Voltmeter	Models	Voltmeter	Traces	
					

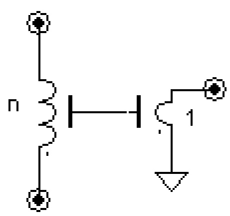
V	Voltmeter	V/V_Voltmeter_Voltmeter.nl5
Model	No parameters	
Voltmeter	**Voltmeter.** I = 0 (open circuit).	

W	Winding	Models	Winding	Traces	
					 P = V • I

W	Winding	W/W_Winding_Winding.nl5

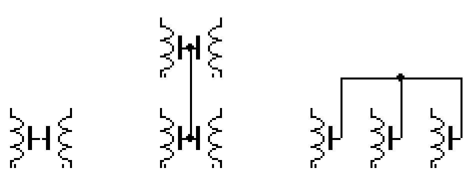
Model	Parameter	Units	Description
Winding	n	turns	Number of turns.

Winding. The Winding is actually an ideal transformer, with 1 turn second winding, one end of each is grounded, and another end is shown as a "core" pin of the winding:

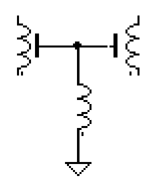


Equivalent schematic

To make an ideal transformer, connect cores of two or more windings by wire. Core magnetizing can be modeled by setting linear or non-linear inductor from core to ground:

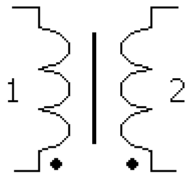


Ideal transformers

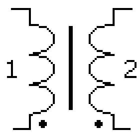


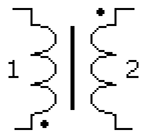
Transformer with magnetizing inductor

W	Transformer	Models	Transformer SubCir	Traces	



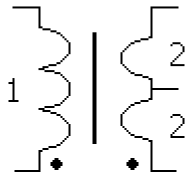
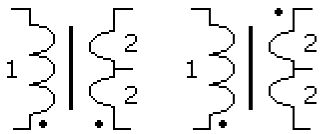
Views





W	Transformer		W/W_Transformer_Transformer.nl5
Model	**Parameter**	**Units**	**Description**
Transformer	n1	turns	Number of turns in the first winding.
	n2	turns	Number of turns in the second winding.
	Ideal transformer with 2 windings. Coupling coefficient = 1.		

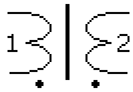
W	Transformer		
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

W	Differential transformer	Models	Transformer SubCir	Traces	
					
Views					

W	Differential transformer		W/W_DifTransformer_Transformer.nl5
Model	Parameter	Units	Description
Transformer	n1	turns	Number of turns in the first winding.
	n2	turns	Number of turns in the second and the third winding.
	Ideal differential transformer with 3 windings. Coupling coefficient = 1. Second and third windings have the same number of turns "n2", and connected to form a differential transformer.		

W	Differential transformer		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Pin5		Name of subcircuit label connected to pin 5
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

W	Custom transformer	Models	Transformer SubCir	Traces	



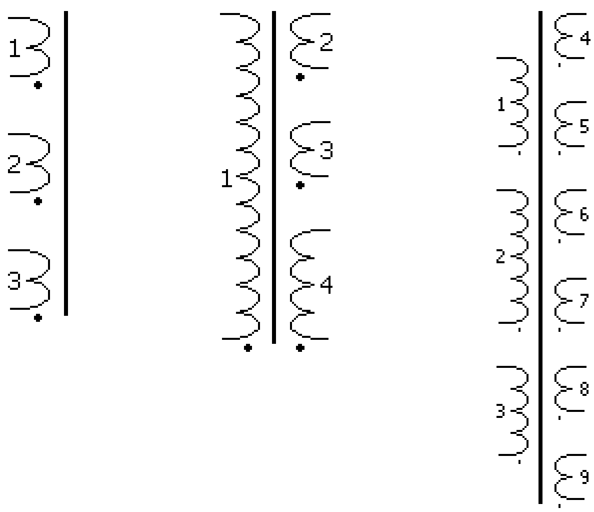
Views

This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:

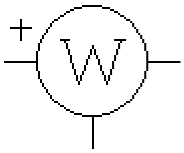
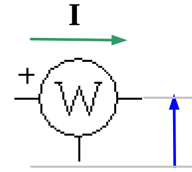
- height from 2 to 32,
- up to 9 windings (total) on both sides,
- arbitrary length of a winding.

Examples of Custom transformer component:



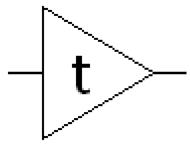
W	Custom transformer		W/W_CustomTransformer_Transformer.nl5
Model	Parameter	Units	Description
Transformer	n1	turns	Number of turns in the first winding.
	…	…	…
	nN	turns	Number of turns in the N^{th} winding.
	Ideal transformer with N windings. Coupling coefficient = 1.		

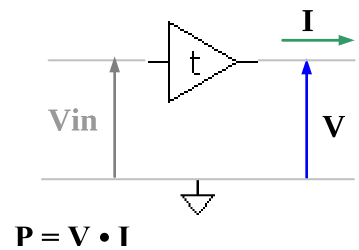
W	Custom transformer		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	…		…
	PinN		Name of subcircuit label connected to pin N
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

W	Wattmeter	Models	Wattmeter	Traces	
					 $P = V \bullet I$

W	Wattmeter	W/W_Wattmeter_Wattmeter.nl5
Model	No parameters	
Wattmeter	**Wattmeter.** Short circuit between current ports, open circuit between voltage ports. Can be used to measure power in grounded or non-grounded load.	

X	Delay	Models	Delay SubCir	Traces	





X	Delay		X/X_Delay_Delay.nl5
Model	**Parameter**	**Units**	**Description**
Delay	t0	s	Delay.
	IC	V	Initial condition: output voltage.

Delay. Output voltage is equal to input voltage, delayed by delay time "t0":

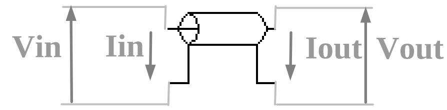
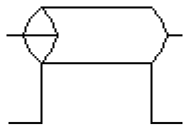
V(t) = Vin(t – t0), where t is current time.

When calculating DC operating point, output is set to specified output voltage "IC", or, if "IC" is blank, to input voltage. Then output voltage is not changing until delay time "t0".

The model allocates memory for storing delayed data only when needed, and frees it immediately when possible. At transient start, an approximate amount of needed memory is estimated based on calculation step, and, if it exceeds a limit specified in preferences (Transient page), the warning message is displayed.

X	Delay		
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string

Subcircuit. See *Working with Subcircuits* chapter for details.

X	Transmission line	Models	Line Lossy	Traces	
					

X	Transmission line		X/X_Line_Line.nl5
Model	**Parameter**	**Units**	**Description**
Line	t0	s	Delay.
	z0	Ohm	Characteristic impedance.
	VIC	V	Initial condition: voltage.
	IIC	A	Initial condition: current.

Lossless transmission line. The voltage and current in the line are represented as a superposition of forward and reflected waves, with V/I ratio in each wave equal to line characteristic impedance "z0". V and I values of each wave are calculated based on boundary (input and output) conditions. The line functionality can also be described by the following equations:

$$Vin(t) = z0 * (Iin(t) - Iout(t - t0))$$
$$Vout(t) = z0 * (Iout(t) - Iin(t - t0))$$

where t is current time.

Input and output are galvanically isolated: no current is flowing between input and output, and any voltage difference between input and output may exist.

When calculating DC operating point initial forward and reflected voltage and current are calculated based on the following conditions:

if "VIC" and "IIC" are blank . . . . : Vin = Vout, Iin = -Iout.
if "VIC" is specified and "IIC" is blank . . : Vin = Vout = "VIC".
if "VIC" is blank and "IIC" is specified . . : Iin = "IIC", Iout = -"IIC".
if "VIC" and "IIC" are specified . . . . : Vin = Vout = "VIC", Iin = "IIC", Iout = -"IIC".

The model allocates memory for storing forward and reflected wave data only when needed, and frees it immediately when possible. At transient start, an approximate amount of needed memory is estimated based on calculation step, and, if it exceeds a limit specified in preferences (Transient page), the warning message is displayed.

X	Transmission line		X/X_Line_Lossy.nl5
Model	Parameter	Units	Description
Lossy	t0	s	Delay.
	z0	Ohm	Characteristic impedance.
	R	Ohm/ns	Series resistance per ns.
	fr	MHz	Skin losses cutoff (3 dB) frequency.
	G	1/Ohm/ns	Shunt conductance per ns.
	fG	MHz	Dielectric losses cutoff (3 dB) frequency.
	VIC	V	Initial condition: voltage.
	IIC	A	Initial condition: current.

Lossy transmission line. Lossy line modeling is similar to lossless transmission line, with addition of losses due to series resistance, skin effect, shunt conductance, and dielectric losses.

Constant series resistance is defined by "r" parameter. Skin losses are modeled by a number of RL chains, providing series impedance increase as a square root of frequency. The number of chains is automatically optimized based on calculation step value; however, the maximum impedance increase due to skin effect is limited to 40 dB (100 times). "fr" parameter defines a frequency where effective series impedance is approximately 3 dB higher than "r". Skin losses are calculated only if "r" > 0 and "fr is not infinite.

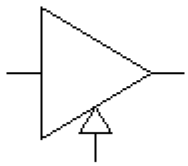
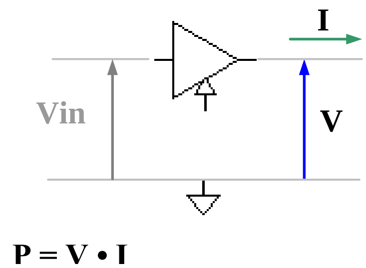
Constant shunt conductance is defined by "G" parameter. Dielectric losses are modeled by a shunt capacitance, providing shunt admittance increase proportional to frequency. "fG" parameter defines a frequency where effective shunt admittance is approximately 3 dB higher than "G". Dielectric losses are calculated only if "G" > 0 and "fG is not infinite.

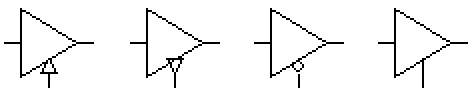
Input and output are galvanically isolated: no current is flowing between input and output, and any voltage difference between input and output may exist.

When calculating DC operating point initial forward and reflected voltage and current are calculated based on the following conditions:

if "VIC" and "IIC" are blank . . . . : Vin = Vout, Iin = -Iout.
if "VIC" is specified and "IIC" is blank . . : Vin = Vout = "VIC".
if "VIC" is blank and "IIC" is specified . . : Iin = "IIC", Iout = -"IIC".
if "VIC" and "IIC" are specified . . . . : Vin = Vout = "VIC", Iin = "IIC", Iout = -"IIC".

The model allocates all the required memory immediately at transient start. The amount of memory is proportional to line delay and inverse proportional to calculation step. If the memory required exceeds a limit specified in preferences (Transient page), the warning message is displayed.

X	Sample/Hold	Models	Traces
		SH SubCir	

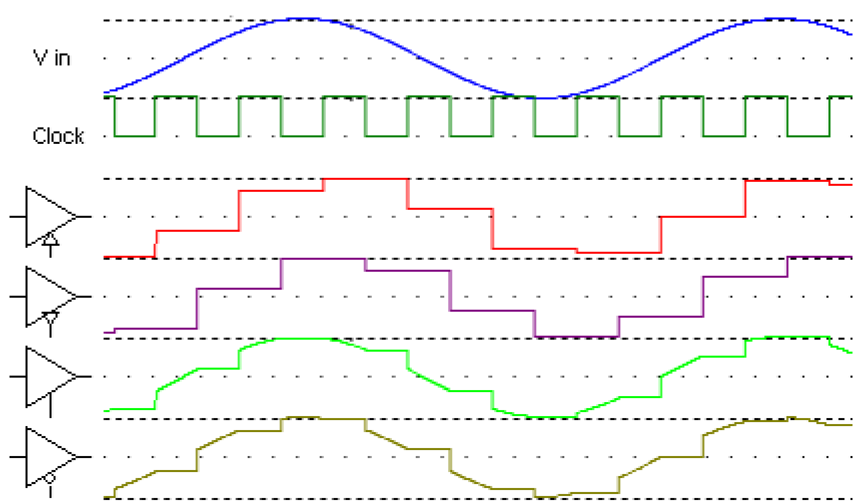
Views	
	Views are used to specify type and polarity of control signal.

X	Sample/Hold		X/X_SampleHold_SH.nl5
Model	Parameter	Units	Description
SH	IC	V	Initial condition: output voltage.

Sample/hold, track/hold. Depending on view, the model is functioning as a sample/hold, or as a track/hold. In sample/hold mode, input voltage is sampled at rising or falling edge of a logical clock signal. In track/hold mode, output voltage tracks input voltage while clock signal is above the logical threshold, and holds it while clock signal is below the logical threshold (clock signal can be inverted).

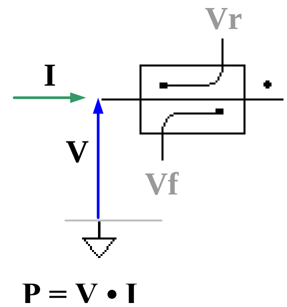
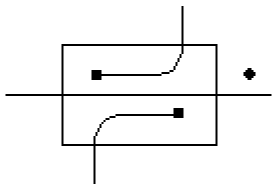
To see and set logical threshold go to **Transient | Settings**, or **AC | Settings** then click **Advanced** button.

A waveforms example for different modes:



When calculating DC operating point output is set to specified output voltage “IC”.

X	Sample/Hold		
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Directional coupler	Models	Coupler	Traces	
					

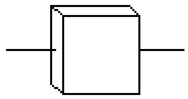
X	Directional coupler		X/X_DirCoupler_Coupler.nl5
Model	**Parameter**	**Units**	**Description**
Coupler	z0	Ohm	Characteristic impedance
	CF	dB	Coupling factor

Directional coupler is a short circuit (no insertion loss) with two output ports: forward (**Vf**) and reflected (**Vr**). Output ports are voltage sources with zero output impedance and coupling factor **CF**. The output voltages are calculated as follows:

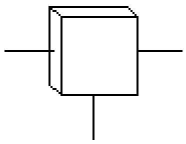
Vf = K * (V + I*z0) / 2
Vr = K * (V – I*z0) / 2

where K = $10^{-CF/20}$.

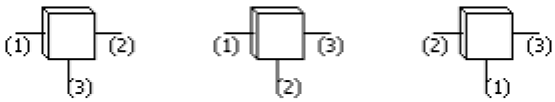
All voltages are referenced to ground.

X	Block-2	Models	SubCir	Traces	
					

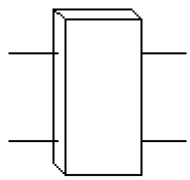
X	Block-2		X/X_Block-2_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Block-3	Models	SubCir	Traces	
					

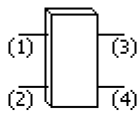
Views

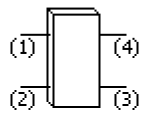


X	Block-3		X/X_Block-3_SubCir.nl5
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Block-4	Models	SubCir	Traces	
					

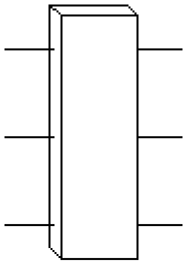
Views



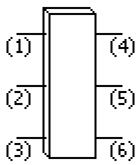


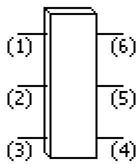
X	Block-4		X/X_Block-4_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Block-6	Models	SubCir	Traces	



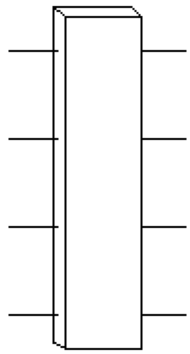
Views



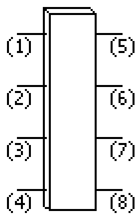


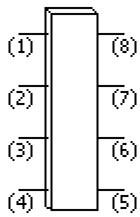
X	Block-6		X/X_Block-6_SubCir.nl5
Model	Parameter	Units	Description
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Pin5		Name of subcircuit label connected to pin 5
	Pin6		Name of subcircuit label connected to pin 6
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Block-8	Models	SubCir	Traces



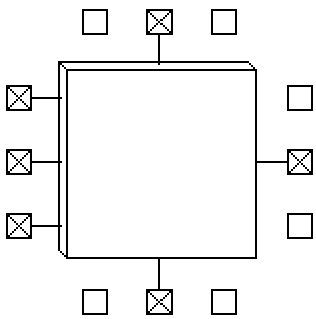
Views





X	Block-8		X/X_Block-8_SubCir.nl5
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	Pin2		Name of subcircuit label connected to pin 2
	Pin3		Name of subcircuit label connected to pin 3
	Pin4		Name of subcircuit label connected to pin 4
	Pin5		Name of subcircuit label connected to pin 5
	Pin6		Name of subcircuit label connected to pin 6
	Pin7		Name of subcircuit label connected to pin 7
	Pin8		Name of subcircuit label connected to pin 8
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

X	Custom block	Models	SubCir	Traces	

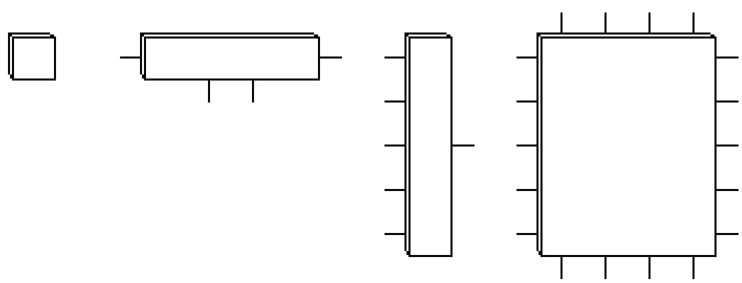


This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:

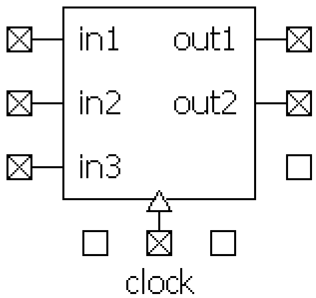
- arbitrary size up to 32(width) X 32(height),
- up to 32 pins on each side

Examples of Custom block component:



X	Custom block		X/X_CustomBlock_SubCir.nl5
Model	**Parameter**	**Units**	**Description**
SubCir	File		File name of subcircuit schematic.
	Pin1		Name of subcircuit label connected to pin 1
	…		…
	PinN		Name of subcircuit label connected to pin N
	Cmd		Subcircuit start-up command string
	IC		Subcircuit Initial conditions string
	Subcircuit. See *Working with Subcircuits* chapter for details.		

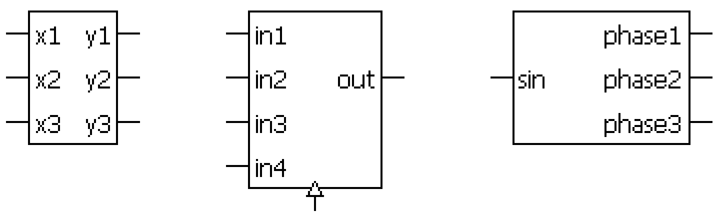
X	Code	Models	C DLL	Traces	



This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:
- arbitrary size up to 32(width) X 32(height),
- up to 32 inputs on the left side,
- up to 32 outputs on the right side,
- one or no clock pins on the bottom side.
- custom input and output names.

Examples of Code component:



X	Code	X/X_Code_C_Ex1.nl5 X/X_Code_C_Ex2.nl5 X/X_Code_C_Ex3.nl5

Model	Parameter	Units	Description
C	Init		Initialization code.
	Main		Main code.
	IC		Initial conditions.

C-code block. The model contains code written on simplified C language. The code will be interpreted by NL5 during transient simulation.

Initialization code "Init" is executed once at the beginning of simulation at t=0. Initialization code is optional. Leave "Init" parameter blank if initialization code is not used.

"Main" is the main code. If ***clock*** pin does not exist, the code is executed on every calculation step. If ***clock*** pin exists, the code is executed only on rising edge of logical clock signal.

"IC" may contain the code assigning initial values to output variables and global variables defined in the initialization code. If not empty, "IC" code will be executed after initialization code.

See *Creating C-code* chapter for details of the model functionality and instructions on creating the code.

X	Code	X/X_Code_DLL_Ex1.nl5 X/X_Code_DLL_Ex2.nl5 X/X_Code_DLL_Ex3.nl5

Model	Parameter	Units	Description
DLL	DLL		DLL file name
	Init		Initialization function name.
	Main		Main function name.

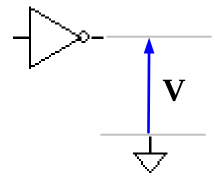
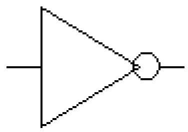
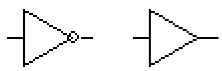
DLL block. Component's code is written in C, compiled, and placed in the DLL file. DLL functions will be called by NL5 during transient simulation.

"DLL" parameter is a DLL file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. File extension "dll" can be omitted.

"Init" is the name of initialization function. Initialization function is executed once at the beginning of simulation at t=0. Initialization function is optional. Leave "Init" parameter blank if initialization function is not used.

"Main" is the name of main function. If ***clock*** pin does not exist, the function is executed on every calculation step. If ***clock*** pin exists, the function is executed only on rising edge of logical clock signal.

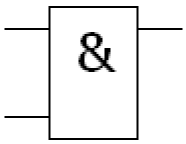
See *Creating DLL code* chapter for details of the model functionality and instructions on creating code and DLL.

Y	Logic-1	Models	Logic Delay	Traces	
					
Views		Views are used to specify inverted or non-inverted output.			

Y	Logic-1		Y/Y_Logic-1_Logic.nl5
Model	Parameter	Units	Description
Logic	IC		Initial condition: Low/High.
	Logical component with one input. Output type (inverted or non-inverted) depends on selected view. Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button. When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.		

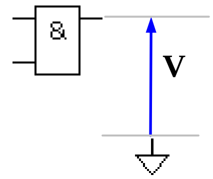
Y	Logic-1		Y/Y_Logic-1_Delay.nl5
Model	Parameter	Units	Description
Delay	Delay	s	Output delay.
	IC		Initial condition: Low/High.
	Logical component with one input and delay. Output type (inverted or non-inverted) depends on selected view. Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button. The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output. When calculating DC operating point output is set to specified level "IC".		

Y Logic-2

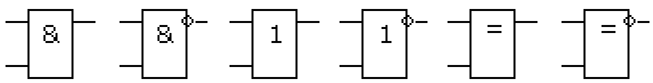


Models: Logic, Delay

Traces:



Views:



Views are used to specify logical function and inverted or non-inverted output.

Y	Logic-2		Y/Y_Logic-2_Logic.nl5
Model	**Parameter**	**Units**	**Description**
Logic	IC		Initial condition: Low/High.

Logical component with two inputs. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.

Y	Logic-2		Y/Y_Logic-2_Delay.nl5
Model	**Parameter**	**Units**	**Description**
Delay	Delay	s	Output delay.
	IC		Initial condition: Low/High.

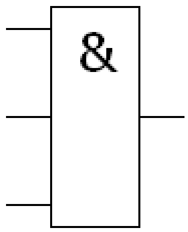
Logical component with two inputs and delay. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output.

When calculating DC operating point output is set to specified level "IC".

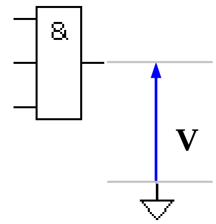
Y Logic-3



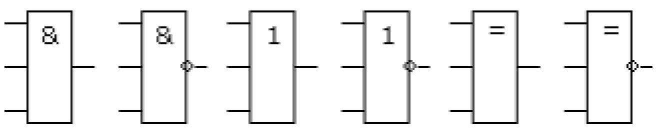
Models

Logic
Delay

Traces



Views



Views are used to specify logical function and inverted or non-inverted output.

Y	Logic-3		Y/Y_Logic-3_Logic.nl5
Model	Parameter	Units	Description
Logic	IC		Initial condition: Low/High.

Logical component with three inputs. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.

Y	Logic-3		Y/Y_Logic-3_Delay.nl5
Model	Parameter	Units	Description
Delay	Delay	s	Output delay.
	IC		Initial condition: Low/High.

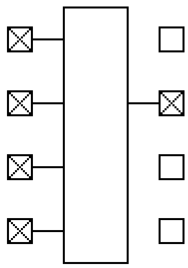
Logical component with three inputs and delay. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output.

When calculating DC operating point output is set to specified level "IC".

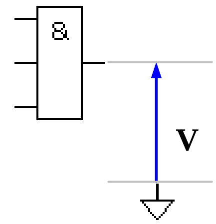
Y Custom logic



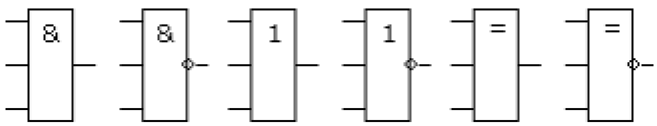
Models

Logic
Delay

Traces



Views



Views are used to specify logical function and inverted or non-inverted output.

This is a customized component. A component can be edited in the **Edit Component** dialog box. See *Editing customized component* chapter for instructions on editing a component.

This component may have:

- arbitrary size up to 32(width) X 32(height),
- up to 32 inputs on the left side,
- one output on the right side.

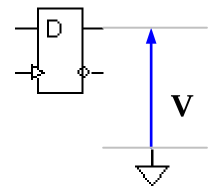
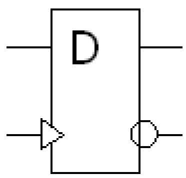
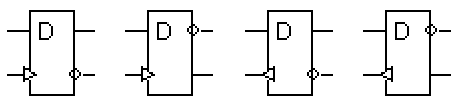
Y	Custom logic		Y/Y_Custom_Logic_Logic.nl5
Model	Parameter	Units	Description
Logic	IC		Initial condition: Low/High.

Custom logical component. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

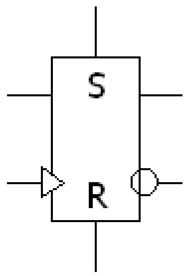
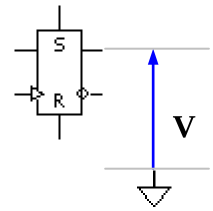
When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.

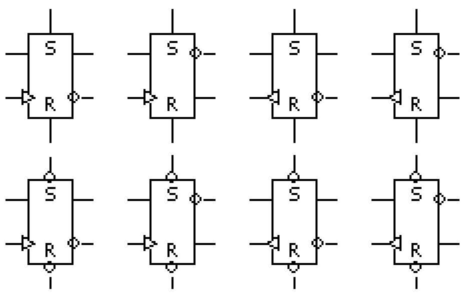
Y	Custom logic		Y/Y_Custom_Logic_Delay.nl5
Model	Parameter	Units	Description
Delay	Delay	s	Output delay.
	IC		Initial condition: Low/High.
	Custom logical component with delay. Logic function (AND, OR, XOR) and output type (inverted or non-inverted) depend on selected view. Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button. The output is delayed by "Delay" time. Input pulses shorter than "Delay" will not pass through and will not affect output. When calculating DC operating point output is set to specified level "IC".		

Y	D-trigger	Models	Logic Delay	Traces	
					
Views		Views are used to specify clock polarity and location of output pins.			

Y	D-trigger		Y/Y_D_Trigger_Logic.nl5
Model	Parameter	Units	Description
Logic	IC		Initial condition: Low/High.
	D-trigger. Clock polarity (rising or falling edge) depends on selected view. Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button. When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.		

Y	D-trigger		Y/Y_D_Trigger_Delay.nl5
Model	Parameter	Units	Description
Delay	Delay	s	Output delay
	IC		Initial condition: Low/High.
	D-trigger with delay. Clock polarity (rising or falling edge) depends on selected view. Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button. Trigger outputs are delayed by "Delay" time. Output pulses with duration shorter than "Delay" will not show up. When calculating DC operating point output is set to specified level "IC".		

Y	RS-trigger	Models	Traces
		Logic Delay	

Views	
	Views are used to specify clock and S/R inputs polarity and location of output pins.

Y	RS-trigger		Y/Y_RS_Trigger_Logic.nl5
Model	Parameter	Units	Description
Logic	IC		Initial condition: Low/High.

RS-trigger. Clock polarity (rising or falling edge) and R/S polarity (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.

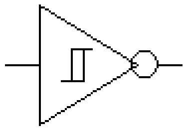
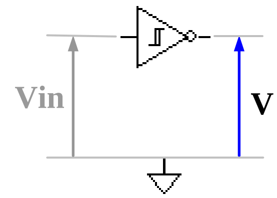
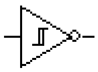
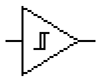
Y	RS-trigger		Y/Y_RS_Trigger_Delay.nl5
Model	Parameter	Units	Description
Delay	Delay	s	Output delay.
	IC		Initial condition: Low/High.

RS-trigger with delay. Clock polarity (rising or falling edge) and R/S polarity (inverted or non-inverted) depend on selected view.

Output voltage may have only logical levels (Low/High). Input voltage is considered Low if it is below logical threshold, or High if it is above logical threshold. To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Trigger outputs are delayed by "Delay" time. Output pulses with duration shorter than "Delay" will not show up.

When calculating DC operating point output is set to specified level "IC".

Y	Schmitt trigger	Models	Traces
		Logic Delay	
Views	 	Views are used to specify inverted or non-inverted output.	

Y	Schmitt trigger		Y/Y_Schmitt_Trigger_Logic.nl5
Model	Parameter	Units	Description
Logic	Hysteresis	V	Hysteresis.
	IC		Initial condition: Low/High.

Schmitt trigger. Output type (inverted or non-inverted) depends on selected view.

Output voltage may have only logical levels (Low/High). To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Output is set to Low or High level following rules (inverted output):

Vin > Threshold + Hysteresis/2 . . . : V = Low
Vin < Threshold - Hysteresis/2 . . . .: V = High
Otherwise . . . .: V = previous state

When calculating DC operating point output is set to specified level "IC". When calculating transient, output voltage is always delayed by one calculation step.

Y	Schmitt trigger		Y/Y_Schmitt_Trigger_Delay.nl5
Model	Parameter	Units	Description
Delay	Hysteresis	V	Hysteresis.
	Delay	s	Output delay
	IC		Initial condition: Low/High.

Schmitt trigger with delay. Output type (inverted or non-inverted) depends on selected view.

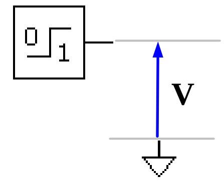
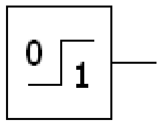
Output voltage may have only logical levels (Low/High). To see and set logical levels and threshold go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Output is set to Low or High level following rules (inverted output):

Vin > Threshold + Hysteresis/2 . . . : V = Low
Vin < Threshold - Hysteresis/2 . . . .: V = High
Otherwise . . . .: V = previous state

Trigger outputs are delayed by “Delay” time. Output pulses with duration shorter than “Delay” will not show up.

When calculating DC operating point output is set to specified level “IC”.

Y	Logic generator	Models	V Pulse Step List File	Traces	
					

Y	Logic generator		Y/Y_Logic_Generator_V.nl5
Model	Parameter	Units	Description
V	Out		Logical output: Low/High.

Logical output. Generates constant Low or High logical output.

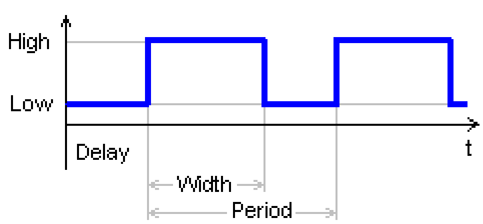
To see and set logical levels go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Y	Logic generator		Y/Y_Logic_Generator_Pulse.nl5
Model	Parameter	Units	Description
Pulse	Period	s	Period.
	Width	s	Pulse width.
	Delay	s	Delay before first pulse starts.
	Active		Active output state: Low/High.

Logical pulses. Pulses start at "Delay" time. Output level is "Active" during "Width" time.

To see and set logical levels go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

The following pulses will be generated if "Active" = On:



Y	Logic generator		Y/Y_Logic_Generator_Step.nl5
Model	Parameter	Units	Description
Step	Delay	s	Delay before active state.
	Active		Active output state: Low/High.

Logical step. Output level is non-"Active" before "Delay" time, turns to "Active" level after "Delay" time.

To see and set logical levels go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Y	Logic generator		Y/Y_Logic_Generator_List.nl5
Model	Parameter	Units	Description
List	List		Comma-separated string.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Logic list. Logical output sequence is defined in the "List" parameter in the csv ("comma-separated values") format, as follows:

t0,s0,t1,s1,...,tn,sn

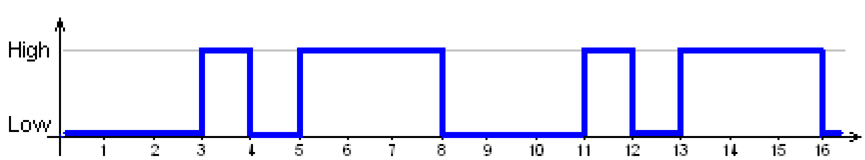
where all t and s can be numerical values or expressions. s0…sn defines output logical level: positive number corresponds to High, zero or negative number - Low. If t<t0, output level is s0. At t0 output level is s0. At t1 output level is s1, and so on. At t>tn, output remains at sn level if "Cycle" parameter is set to "No", otherwise the sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

To see and set logical levels go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Example:

```
List = 0,0,3,1,4,0,5,1,8,0
```

The following logical output will be generated if "Cycle" = Yes, "Delay" = 0:



Y	Logic generator		Y/Y_Logic_Generator_File.nl5
Model	Parameter	Units	Description
File	File		File name.
	Cycle		Cycling (repeat): No/Yes.
	Delay	s	Delay.

Logic file. Logical output sequence is defined in the text file. "File" parameter is a file name, with full path to the file. If the file is located in the same directory as schematic file, the path can be omitted. Logical output sequence is defined in the following format:

```
<if first line does not start with a number, it is ignored>
t0,s0
t1,s1
.....
tn,sn
```

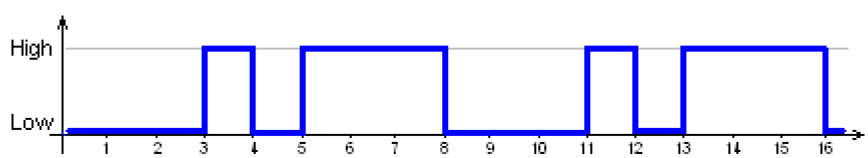
where all t and s can be numerical values or expressions. s0…sn defines output logical level: positive number corresponds to High, zero or negative number - Low. If t<t0, output level is s0. At t0 output level is s0. At t1 output level is s1, and so on. At t>tn, output remains at sn level if "Cycle" parameter is set to "No", otherwise the sequence defined in t0…tn interval is repeated continuously. In addition, the whole signal is delayed by "Delay" time.

To see and set logical levels go to **Transient | Settings**, or **AC | Settings**, then click **Advanced** button.

Example:

```
0,0
3,1
4,0
5,1
8,0
```

The following logical output will be generated if "Cycle" = Yes, "Delay" = 0:



2. Operators

P column shows operator precedence: 1- least, 8 - most.
C column indicates if operator supports complex numbers.

Operator	P	C	Description	Example
x=y	1	√	Assignment	
x+=y	1	√	Assignment by addition, x=x+y	
x-=y	1	√	Assignment by subtraction, x=x-y	
x*=y	1	√	Assignment by multiplication, x=x*y	
x/=y	1	√	Assignment by division, x=x/y	
a?x:y	2	√	x if a > 0, y otherwise Condition a is calculated first, then only x or y is calculated, according to condition.	3>2?1:0=1 3==2?1:0=0
x&&y	3		1 if x>0 and y>0, 0 otherwise	5&&8=1 5&&0=0
x||y	3		1 if x>0 or y>0, 0 otherwise	5||(-5)=1 (-5)||(-8)=0
x<y	4		1 if x<y, 0 otherwise	5<8=1 8<5=0
x<=y	4		1 if x<=y, 0 otherwise	5<=5=1 5<=2=0
x>y	4		1 if x>y, 0 otherwise	5>8=0 8>5=1
x>=y	4		1 if x>=y, 0 otherwise	5>=5=1 5>=8=0
x==y	4	√	1 if x equal y, 0 otherwise	5==8=0 5+j1==5+j1=1
x!=y	4	√	1 if x non-equal y, 0 otherwise	5!=8=1 5+j1!=5+j1=0
x+y	5	√	x+y	2+3=5 1+1j+2+2j=3+3j
x-y	5	√	x-y	3-2=1 3+3j-2-2j=1+1j
x*y	6	√	x*y	2*3=6 (1+2j)*(1-2j)=5

Operator	P	C	Description	Example
x/y	6	√	x/y	4/2=2 (2+2j)/(1-1j)=0+2j
x%y	6		x modulo y, the remainder of x/y	5%2=1
x^y	7	√	x^y : x to the power of y	2^3=8 (-4)^0.5=0+2j 1j^3=0-1j
++x	8	√	Prefix increment: x=x+1 before use	
x++	8	√	Postfix increment: x=x+1 after use	
--x	8	√	Prefix decrement: x=x-1 before use	
x--	8	√	Postfix decrement: x=x-1 after use	

3. Functions

C column indicates if function supports complex numbers.

Function	C	Description	Example
sin(x)		sin(x), x in degrees.	sin(90)=1
cos(x)		cos(x), x in degrees.	cos(45)= 707.106e-3
tan(x) tg(x)		tan(x), x in degrees.	tan(45)=1
sqrt(x)	√	$\sqrt{x}$: square root	sqrt(4)=2 sqrt(-4)=0+2j sqrt(2j)=1+1j
sqr(x)		"Signed" square root: $\sqrt{x}$ if x>=0, $-\sqrt{(-x)}$ if x<0	sqr(4)=2 sqr(-4)=-2
sq(x)	√	x^2 : square	sq(2)=4 sq(1+1j)=0+2j
pow(x,y)	√	x^y : x to the power of y	pow(10,2)=100 pow(-4,0.5)=0+2j pow(1j,3)=0-1j
pwr(x,y)		"Signed" power: x^y if x>=0, $-(-x)^y$ if x<0	pwr(10,2)=100 pwr(-10,2)=-100
exp(x)	√	e^x : exponent	exp(3)=20.0855 exp(PI*0.5j)=0+1j
ln(x) log(x)	√	ln(x) : logarithm x to base *e*	ln(100)=4.60517 ln(-1)=0+3.14159j ln(-1j)=0-1.57079j
lg(x) log10(x)	√	$\log_{10}(x)$: logarithm x to base 10	log10(100)=2 lg(-100)=2+1.36437j lg(1j)=0+682.188e-3j
lb(x) log2(x)	√	$\log_2(x)$: logarithm x to base 2 ("binary" logarithm)	log2(8)=3 lb(-8)=3+4.53236j lb(1j)=0+2.26618j
log(x,y)	√	$\log_y(x)$: logarithm x to base y. Complex base y not allowed.	log(PI,PI)=1 log(-10,10)=1+1.36437j log(1j,10)=0+682.1e-3j
asin(x)		arcsin(x), -90…+90 degrees	asin(1)=90
acos(x)		arccos(x), 0…+180 degrees	acos(.5)=60
atan(x)		arctan(x), -90…+90 degrees	atan(1)=45

Function	C	Description	Example
atan2(x,y)		arctan(x/y), -180…180 degrees	atan2(1,-1)=135 atan2(1,1)=45
abs(x) mag(x)		Absolute value (magnitude)	abs(1)=1 mag(-10)=10
abs(x,y,…) mag(x,y,…)		$\sqrt{x^2+y^2+\dots}$, number of arguments not limited	abs(3,4)=5 mag(1,1,1,1)=2
sign(x)		Sign x : 1 if x>0, 0 if x=0, -1 if x<0	sign(-2)=-1 sign(0)=0 sign(100)=1
db(x)		20*$\log_{10}$(abs(x)) : x in decibel	db(100)=40
db(x,y)		20*$\log_{10}$(abs(x/y)) : ratio x/y in decibel	db(1,10)=-20
min(x,y, …)		Minimum, number of arguments not limited	min(5,9,10,2)=2
max(x,y, …)		Maximum, number of arguments not limited	max(5,9,10,2)=10
int(x) round(x)		Round x to the nearest integer	int(1.4449)=1 int(1.5)=2
int(x,y) round(x,y)		Round x to the nearest multiple of y, x if y<=0.	int(123,10)=120 int(3.1415,0.1)=3.1
abs(**c**) mag(**c**)	√	Absolute value (magnitude) of complex number **c**	abs(3+4j)=5 mag(-3j)=3
phase(**c**)	√	Phase of complex number **c**, -180…180 degrees	phase(1+1j)=45
re(**c**)	√	Real part of complex number **c**	re(2+3j)=2
im(**c**)	√	Imaginary part of complex number **c**	im(2-3j)=-3
par(x,y)	√	(x*y)/(x+y) : x in parallel with y	par(2,3)=1.2 par(10j,10)=5+5j
par(x,y,…)	√	x,y,… in parallel, number of arguments not limited	par(1,1,1,1)=0.25
random(x) rand(x)		Random number in the range 0…x	rand(1)=0.2937463
limit(x,min,max) lim(x,min,max)		Limit x: min if x<min, max if x>max x if min<=x<=max	lim(0,-1,1)=0 lim(-2,-1,1)=-1 lim(10,-1,2)=2
islow(x)		1 if x < logical threshold 0 otherwise	islow(1.0)=1 islow(55)=0

Function	C	Description	Example
ishigh(x)		1 if x >= logical threshold 0 otherwise	ishigh(1.0)=0 ishigh(55)=1

4. C language syntax

Comments. Use // to comment text until the end of the line, or delimiters /* and */ to comment block of the text. Delimiters /* and */ can be nested.

```
for( i=0; i<10; ++i ) { // this is a comment
   /* This block is commented out
   x=i*2;
   y=i/10;
   */
   x=i;
}
```

Variables. Only **double** type is supported. To declare a new variable:

- use keyword **double**,
- or assign some value to a new name,
- or use a new name in the **for** statement**.**

```
double x, y, z;
double x=1.0;
i=5;
for( n=0; n<10; ++n );
```

Arrays. Only one-dimensional arrays are supported. Index is zero-based. Use keyword **double** to declare a new array.

```
double x[100];
double y[] = { 1, 2, 3, 4, 5 };
```

if…else. Conditional statement.

```
if(i<=0) R1=1.0;
else if(i==1) R1=2.0;
else {
   R1=3.0;
   C1=1n;
}
```

for. Loop operator.

```
for( i=0; i<10; ++i ) {
   x[i]=2^i;
   y+=x[i];
}
```

for. "Foreach" loop operator. The code is executed for all values from the comma-separated list.

```
for( i=1,5,10,50,100 ) {
   y*=i;
}
```

while. Loop operator.

```
i=0;
while( i<10 ) {
   x[i]=2^i;
   ++i;
}
```

do…while. Loop operator.

```
i=0;
do {
   x[i]=2^i;
   ++i;
}
while( i<10 );
```

switch. Selective structure.

```
switch(i) {
   case 1: x=1; break;
   case 2: x=2; break;
   default: x=3; break;
}
```

continue. Skip the rest of the code in the current loop.

```
for( i=0; i<10; ++i ) {
   x[i]=2^i;
   if(i==5) continue;
   y+=x[i];
}
```

break. Leave current loop or **switch** statement.

```
for( i=0; i<10; ++i ) {
   x[i]=2^i;
   if(i==5) break;
   y+=x[i];
}
```

return. Stop execution of the code immediately and exit.

```
for( i=0; i<10; ++i ) {
   x[i]=2^i;
   if(x[i]==0) return;
   y/=x[i];
}
```

5. Script commands

In alphabetical order.

Command	Description

`ac [`*from*`[,`*to* `[,`*points* `[,`*scale*`]]]]`	Set AC analysis parameters and start AC analysis.
	    from = start frequency
	    to = stop frequency
	    points = number of points
	    scale =log or lin - logarithmic or linear frequency scale.
	If called from the script, command will not return until AC analysis is completed. If called from console or HTTP link, returns immediately. Use "ready" command to check for analysis completion.
	
	Example:
	    ac;
	    ac 1M;
	    ac 1M, 100M;
	    ac 1M, 100M, 500;
	    ac 1M, 100M, 500, lin;

clear	Clear storage. Example: clear;
close	Close active document. Example: close;

`cont [`*screen*`[,`*step*`]]`	Continue transient.
	   screen = screen size
	   step = calculation step
	If called from the script, command will not return until transient is completed. If called from console or HTTP link, returns immediately. Use "ready" command to check for transient completion.
	
	Example:
	    cont;
	    cont 1m;
	    cont 1m, 10n;

`cursors` *left*`,`*right*	Show cursors (transient or AC) and set to specified positions. Example: cursors 1.5, 2.5;

Command	Description
cursors off	Hide cursors (transient or AC). Example: cursors off;
display off display on	Hide or show transient and AC windows. Example: display off;
exit	Close all documents and exit NL5. Can not be called from console command line. Example: exit;
export [*filename*[,*from*[,to [,*step*]]]]	Export traces into csv file. If *filename* is omitted, name of the file to export is the same as script file name, with "csv" extension. If file path is not specified, export in the script file directory. Extension "csv" can be omitted. Number of points can not exceed "Max number of points" value defined on the Preferences Transient page. If *step* is omitted, export 101 points. from = start of the data interval to = end of the data interval step = step Example: export; export rctraces; export rctraces,0,100; export rctraces,0,1,1m;
export [*filename* [*from*[,*to* [,*points*[,*scale*]]]]]	Export AC traces into csv file. If *filename* is omitted, name of the file to export is the same as script file name, with "csv" extension. If file path is not specified, export in the script file directory. Extension "csv" can be omitted. from = start frequency to = stop frequency points = number of points scale =log or lin - logarithmic or linear frequency scale Example: export; export actraces; export actraces,1m,1k; export actraces,1m,1k,100; export actraces,1m,1k,100,lin;

Command	Description

open *filename*	Open schematic file *filename*. Extension "nl5" can be omitted. If file path is not specified, search in the script file directory.
	
	Example:
	    open "c:Project files/nl5/rc.nl5";
	    open rc;

pause	Pause transient. Command can be called from console command line only. Example: pause;
ready	Check if transient or AC analysis is completed. Returns "0" if analysis is running, returns "1" if completed. Example: ready;

rununtil [*expression*]	Set up "run until" transient mode.
	If parameter is omitted, turn off "run until" mode and clear "run until" expression. Otherwise turn on "run until" mode and use parameter as "run until" expression.
	
	Example:
	    rununtil;
	    rununtil V(C1)<0;

save [*filename*]	Save schematic to file *filename*. Extension "nl5" can be omitted.
	If *filename* is omitted, save into the same file. If file path is not specified, save in the script file directory.
	
	Example:
	    save;
	    save rcnew;

savedata [*filename*]	Save traces into "nlt" data file. Extension "nlt" can be omitted.
	If *filename* is omitted, name of the file to save data is the same as script file name, with "nlt" extension. If file path is not specified, save in the script file directory.
	
	Example:
	    savedata;
	    savedata rctraces;

Command	Description
saveic	Save Initial Conditions. Example: saveic;
sleep *time*	Pause script execution for *time* ms. Example: sleep 1000;
stop	Stop transient. This command can be used to free memory allocated for transient analysis. Transient can not be continued after this command. Example: stop;

store [*expr*]	Move run into storage. The parameter is evaluated as an expression, and the result is used as a storage name. If parameter is omitted, a default storage name "RunN" is used.
	
	Example:
	    store;
	    store R1*C1;

storetext [*text*]	Move run into storage with parameter as a storage name. If parameter is omitted, a default storage name "RunN" is used.
	
	Example:
	    storetext;
	    storetext This is first run;

tracename [*from*[,to [,*step*]]]	Request transient trace data as a comma-separated string. Trace *tracename* should be specified in the Transient Data, however it does not need to be displayed on the graph or in the table. Number of points can not exceed "Max number of points" value defined on the Preferences Transient page.
	   from = start of the data interval
	   to = end of the data interval
	   step = step
	
	Example: // Returns:
	    V(R1); // 101 points of the entire trace
	    V(R1) 1.23; // trace value at t=1.23
	    V(R1) 0,100; // 101 points in the specified
	                    // interval
	    V(R1) 0,10,0.1; // specified interval and step

Command	Description

tracename [*from*[,*to*
[,*points*[,*scale*]]]]
	Request AC trace data as a comma-separated string. Trace *tracename* should be specified in the AC Data, however it does not need to be displayed on the graph or in the table.
	from = start frequency
	to = stop frequency
	points = number of points
	scale =log or lin - logarithmic or linear frequency scale
	
	Example: // Returns:
	V(R1); // all calculated AC points
	V(R1) 1.23; // trace value at t=1.23
	V(R1) 1,100; // specified interval
	V(R1) 1,100,10; // specified interval and number
	// of points
	V(R1) 1,100,10,lin; // specified interval, number
	// of points and frequency scale

tran [*start*[,*screen*[,*step*]]]	Set transient parameters and start transient start = start of transient display screen = screen size step = calculation step If called from the script, command will not return until transient is completed. If called from console or HTTP link, returns immediately. Use "ready" command to check for transient completion. Example: tran; tran 0, 10m; tran 0, 10m, 1mk;

6. END USER LICENSE AGREEMENT

This End-User License Agreement ("EULA", "Agreement") is a legal agreement between you ("you", either an individual or a single entity) and Sidelinesoft, LLC ("Sidelinesoft") for the NL5 Circuit Simulator software ("the Software", "the Software Product"), NL5 License ("the Software License"), and accompanying documentation.

Ownership
The Software, any accompanying documentation, and all intellectual property rights therein are owned by Sidelinesoft. The Software is licensed, not sold. The Software is protected by copyright laws and treaties, as well as laws and treaties related to other forms of intellectual property. The Licensee's license to download, use, copy, or change the Software Product is subject to these rights and to all the terms and conditions of this Agreement.

Acceptance
YOU ACCEPT AND AGREE TO BE BOUND BY THE TERMS OF THIS AGREEMENT BY DOWNLOADING THE SOFTWARE PRODUCT OR BY INSTALLING, USING, OR COPYING THE SOFTWARE PRODUCT. YOU MUST AGREE TO ALL OF THE TERMS OF THIS AGREEMENT BEFORE YOU WILL BE ALLOWED TO DOWNLOAD THE SOFTWARE PRODUCT. IF YOU DO NOT AGREE TO ALL OF THE TERMS OF THIS AGREEMENT, YOU MUST NOT INSTALL, USE, OR COPY THE SOFTWARE PRODUCT.

License Grant
Sidelinesoft grants you a right to download, install, and use unlimited copies of the Software Product. Without a Software License, the Software operates as a Demo version, with limited number of components in the schematic, and possibly some functional and performance limitations. Several types of Full-Function Software Licenses can be obtained at Software Product website (nl5.sidelinesoft.com). Terms and conditions of each type of Full-Function Software License are available at the website and are subject to change without notice.

Restrictions on Reverse Engineering, Decompilation, and Disassembly.
You may not decompile, reverse-engineer, disassemble, or otherwise attempt to derive the source code for the Software Product.

Restrictions on Alteration
You may not modify the Software Product or create any derivative work of the Software Product or its accompanying documentation without obtaining permission of Sidelinesoft. Derivative works include but are not limited to translations. You may not alter any files or libraries in any portion of the Software Product.

Consent to Use of Data
Sidelinesoft may ask for your permission to collect and use technical information gathered as part of the product support services provided to you, if any, related to the Software. Sidelinesoft may use this information solely to improve the Software or to provide customized services to you and will not disclose this information in a form that personally identifies you.

Disclaimer of Warranties and Limitation of Liability
UNLESS OTHERWISE EXPLICITLY AGREED TO IN WRITING BY SIDELINESOFT, SIDELINESOFT MAKES NO OTHER WARRANTIES, EXPRESS OR IMPLIED, IN FACT OR IN

LAW, INCLUDING, BUT NOT LIMITED TO, ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE OTHER THAN AS SET FORTH IN THIS AGREEMENT.
Sidelinesoft makes no warranty that the Software Product will meet your requirements or operate under your specific conditions of use. Sidelinesoft makes no warranty that operation of the Software Product will be secure, error free, or free from interruption. YOU MUST DETERMINE WHETHER THE SOFTWARE PRODUCT SUFFICIENTLY MEETS YOUR REQUIREMENTS FOR SECURITY AND UNINTERRUPTABILITY. YOU BEAR SOLE RESPONSIBILITY AND ALL LIABILITY FOR ANY LOSS INCURRED DUE TO FAILURE OF THE SOFTWARE PRODUCT TO MEET YOUR REQUIREMENTS. UNDER NO CIRCUMSTANCES SHALL SIDELINESOFT BE LIABLE TO YOU OR ANY OTHER PARTY FOR INDIRECT, CONSEQUENTIAL, SPECIAL, INCIDENTAL, PUNITIVE, OR EXEMPLARY DAMAGES OF ANY KIND (INCLUDING LOST REVENUES OR PROFITS OR LOSS OF BUSINESS) RESULTING FROM THIS AGREEMENT, OR FROM THE PERFORMANCE, INSTALLATION, USE OR INABILITY TO USE THE SOFTWARE PRODUCT, WHETHER DUE TO A BREACH OF CONTRACT, BREACH OF WARRANTY, OR THE NEGLIGENCE OF SIDELINESOFT OR ANY OTHER PARTY, EVEN IF SIDELINESOFT IS ADVISED BEFOREHAND OF THE POSSIBILITY OF SUCH DAMAGES. TO THE EXTENT THAT THE APPLICABLE JURISDICTION LIMITS SIDELINESOFT'S ABILITY TO DISCLAIM ANY IMPLIED WARRANTIES, THIS DISCLAIMER SHALL BE EFFECTIVE TO THE MAXIMUM EXTENT PERMITTED.

Limitation of Remedies and Damages
Your remedy for a breach of this Agreement or of any warranty included in this Agreement is the correction or replacement of the Software Product. Selection of whether to correct or replace shall be solely at the discretion of Sidelinesoft. Any claim must be made within the applicable warranty period. All warranties cover only defects arising under normal use and do not include malfunctions or failure resulting from misuse, abuse, neglect, alteration, improper installation, or a virus. All limited warranties on the Software Product are granted only to you and are non-transferable. You agree to indemnify and hold Sidelinesoft harmless from all claims, judgments, liabilities, expenses, or costs arising from your breach of this Agreement and/or acts or omissions.

Severability
If any provision of this Agreement shall be held to be invalid or unenforceable, the remainder of this Agreement shall remain in full force and effect. To the extent any express or implied restrictions are not permitted by applicable laws, these express or implied restrictions shall remain in force and effect to the maximum extent permitted by such applicable laws.

Termination
This Agreement is effective until terminated. Without prejudice to any other rights, Sidelinesoft may terminate this Agreement if you fail to comply with the terms and conditions of this Agreement. In such event, you must destroy all copies of the Software License.

Governing Law, Dispute Resolution
This Agreement is governed by the laws of the State of Colorado, U.S.A., without regard to its choice of law principles to the contrary.

Contact Information.
Any inquiries regarding this Agreement or the Software may be addressed to Sidelinesoft at the Software Product website (nl5.sidelinesoft.com).

The end