



# NL Circuit Simulator User's Manual

Ver.1

## VERSION

NL Circuit Simulator User's Manual version 1, 04/10/2025

The latest versions of NL documents can be found at [sidelinesoft.com/nl5](http://sidelinesoft.com/nl5).

## LIMITED LIABILITY

NL, together with 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 NL.

## COPYRIGHTS

© 2025, A.Smirnov. The program and User's Manual are copyrighted. No portion of this Manual can be translated or reproduced for commercial purpose without the express written permission from the copyright holder. On publication of results obtained from use of NL citation is appreciated.



---

Microsoft, Windows, and Microsoft Visual C++ are registered trademarks of Microsoft Corporation.

# Table of Contents

<b>I. Introduction .....</b>	<b>4</b>
What is NL.....	5
Install and run NL .....	5
NL file types.....	6
NL Help.....	6
NL preferences .....	7
Create and Simulate Your First Schematic.....	8
<b>II. User Interface .....</b>	<b>11</b>
Application.....	12
Navigation bar .....	12
Settings window.....	13
Context menus.....	14
Shortcuts .....	14
Mouse .....	14
Data format.....	15
Numbers .....	15
Names .....	17
Operators.....	18
Functions.....	18
Expressions .....	19
<b>III. Schematic .....</b>	<b>20</b>
<b>Edit schematic .....</b>	<b>21</b>
Cursor.....	21
Wire.....	21
Ground .....	21
Connection .....	22
Component.....	22
Label .....	23
Zoom and scrolling.....	23
Selection.....	24
General commands and shortcuts .....	24
<b>Components.....</b>	<b>25</b>
<b>Working with special component types and models.....</b>	<b>28</b>
PWL model .....	28
List model (voltage and current source) .....	31
List model (switch and logic generator) .....	32

Table model .....	33
<b>Schematic variables .....</b>	<b>34</b>
<b>Schematic settings .....</b>	<b>35</b>
<b>Check schematic.....</b>	<b>36</b>
<b>IV. Transient .....</b>	<b>37</b>
<b>Algorithm.....</b>	<b>38</b>
<b>Running transient .....</b>	<b>40</b>
<b>Transient window.....</b>	<b>41</b>
<b>Transient settings .....</b>	<b>44</b>
Simulation .....	44
Add traces .....	45
Format traces.....	46
Screen.....	47
<b>V. AC analysis .....</b>	<b>48</b>
<b>Algorithm.....</b>	<b>49</b>
<b>Running AC analysis .....</b>	<b>50</b>
<b>AC window .....</b>	<b>51</b>
<b>AC settings.....</b>	<b>53</b>
Simulation .....	53
Add traces .....	54
Format traces.....	55
Screen.....	56

# I. Introduction

## What is NL

NL is a **free** introductory version of **NL5 Circuit Simulator**. It uses the same simulation algorithm as **NL5**, same component types and models, and it can perform simulation with unlimited number of components. However, its user interface is very simple and limited, and it does not support most of the features, tools, and unique functionality available in **NL5**. It also supports just a small number of component types and models offered by **NL5**.

NL can be used to learn how schematic is entered and edited, how component parameters are specified; then you can perform simulation and see results in a very simple format. You can continue using NL for simple projects that do not require special components, extensive analysis and tools: this should be quite adequate tool for beginners, students, and hobbyists.

For professionals who need special components and features, extensive analysis capabilities and tools, switching to full-functional **NL5** is highly recommended. After starting with simple NL, such a transition can be made easily.

NL and **NL5** schematic files have the same format, so that any \*.nl5 schematic file can be opened and simulated by both NL and **NL5**. However, due to limited functionality NL may remove or replace schematic settings which are not supported. For example, common changes in **NL5** schematic made by NL will be:

- Components of unsupported types removed.
- Unsupported models replaced by another supported model.
- Formulas in component parameters replaced by values.
- Advanced formatting, drawings and text in schematic/transient/AC windows removed.
- Traces of non-supported types removed or replaced by **V** trace.
- Digital and Bus transient trace modes replaced by **Analog** mode.
- Transient and AC data included into the file removed.
- All Transient and AC Tools configuration removed.
- And more...

Please note that all those changes (except component/model removal and modification) will not affect simulation results.

## Install and run NL

The only file required for NL operation is executable `nl.exe`. You can place `nl5.exe` into any directory. There could be several copies of `nl.exe` in different directories on one computer.

It is recommended to run `nl.exe` with administrator privileges, to have full access to the **Registry**. To set up administrator mode in Windows 10 and later, right-click on the NL icon, select **Properties**, select **Compatibility** tab, check **Run this program as an administrator**.

## NL file types

The following file extensions are used by NL:

Extension	Description
n15	Schematic
n15~	Schematic backup
nlp	Preferences

Information about NL file types and icons is stored in the registry. Please run NL as administrator to be able to modify that information.

Please note that NL and NL5 schematic files are compatible and can be used by both tools. However, if you open NL5 file with NL and then save it, some NL5-specific information can be lost.

## NL Help

NL is using the same help file as **NL5**: `n15.chm` . The help file is not required, but if used, it should be placed in the same directory as `n1.exe`. The file contains only reference information, such as description of operators, functions, commands, components, and models. For detailed information refer to the NL User's Manual and NL5 User's Reference.

Please note that some information in the Help file is for NL5 only and does not apply to NL.

For context Help, press **F1**, or click **Help** button , which is available in some windows.

If you cannot see content of Help file, most likely the file is blocked. To unblock:

- Locate `n15.chm` file in the NL directory.
- Right-click the file, then click **Properties**.
- Select **General** tab.
- Click **Unblock**.
- If **Unblock** button is not visible, delete `n15.chm` file from the directory, copy it to NL directory again from `n1.zip` download package, and repeat this procedure.

## NL preferences

**Preferences**  are stored in the file `n1.nlp`, typically located in the same directory as `n1.exe`. NL reads preferences from the file at start-up and saves into the file every time **Apply** or **OK** button in the **Preferences** window, and on exit.

Please note that if `n1.exe` is located anywhere inside `C:\Program Files` directory, it may not be able to save preferences file `n1.nlp` due to Windows writing restrictions. In this case, the preferences file will be saved in the `C:\Users\user_name\Documents\n15` directory (or similar directory, depending on specific OS).

To open **Preferences** window click  in the Main menu, or in some pop-up context menus.

## Create and Simulate Your First Schematic

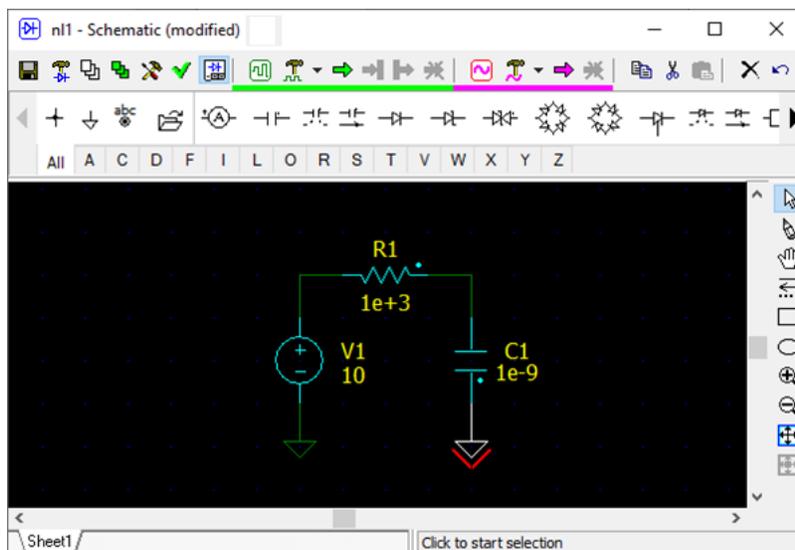
To create a new schematic, click **New** toolbar button  in the main NL window.

### Enter schematic.

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

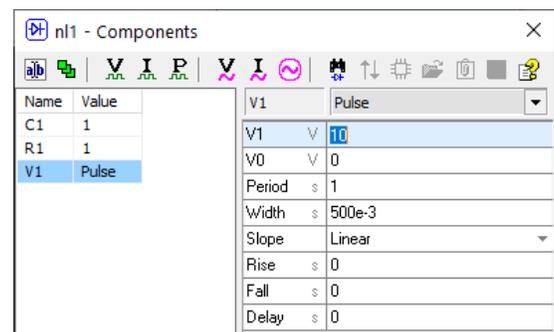
A red cursor is located in the middle of the screen and is pointing to the right.

- 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.



### Edit component parameters.

- Double click on the voltage source **V1** on the schematic. A **Components** window will show up.
- Click on **V1** in the components list. Click  right to the model name (top-right), select **Pulse**.
- Click on the resistor **R1**, enter 1 in the parameter field **R**.
- Click on the capacitor **C1**, enter 1 in the parameter field **C**.

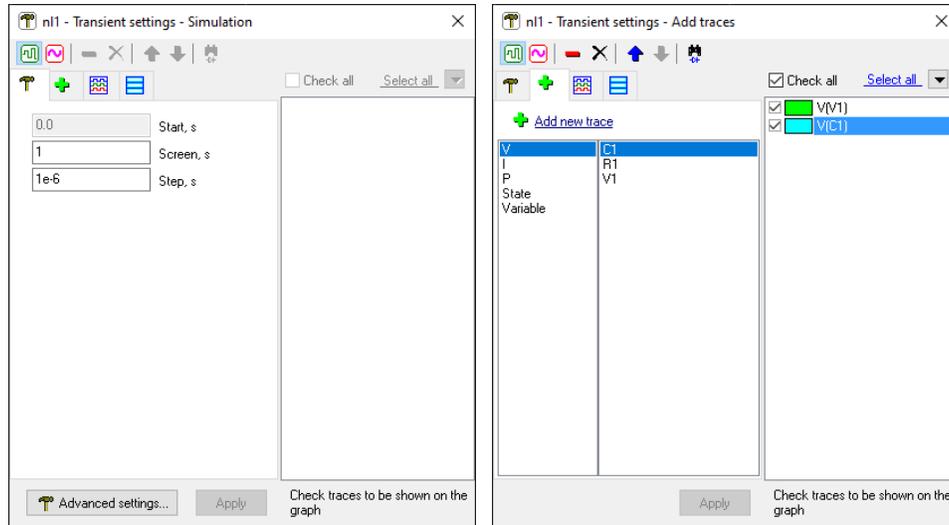


## View transient settings.

Click **Transient settings**  in the schematic window toolbar, select **Simulation**  to go to **Settings** window, **Simulation** tab. You do not need to change anything here, but you can, if you wish.

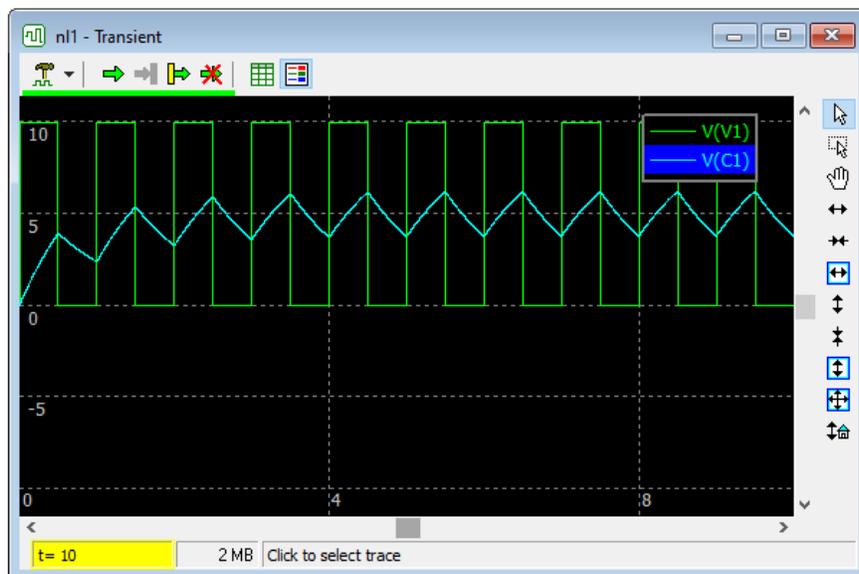
## Add transient traces.

Select **Add traces** tab , select **V** in the left window, then double-click on **V1** and **C1** in the components list. Voltage traces will be added to the trace list.



## Run transient.

Click **Start transient**  to run simulation:

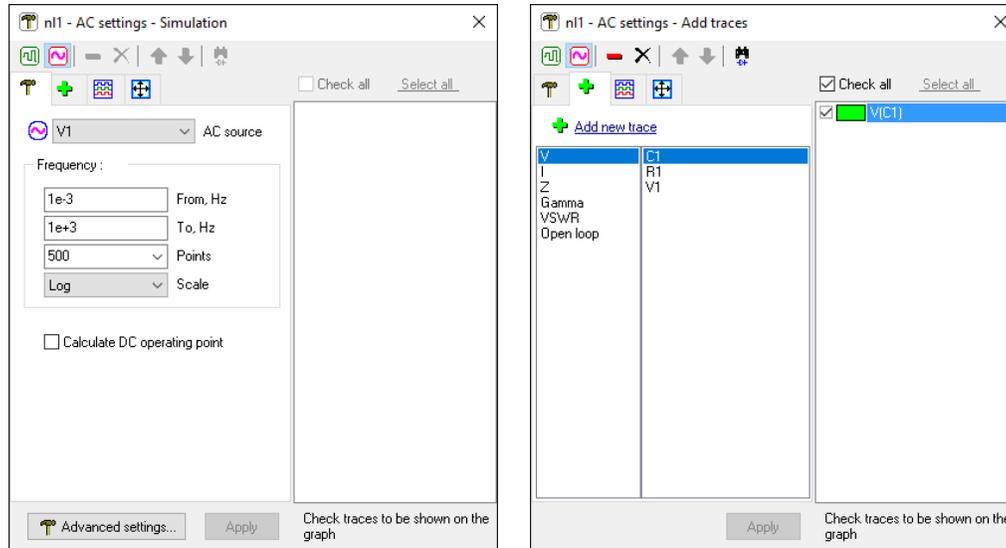


## View and edit AC settings.

Click **AC settings**  in the schematic window toolbar, select **Simulation**  to go to **Settings** window, **Simulation** tab. Click on the **AC source** drop-down list, select **V1**.

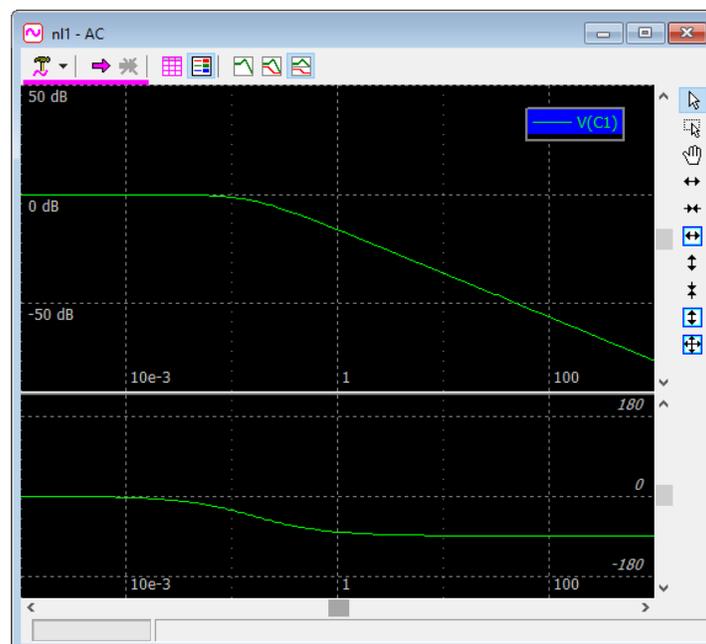
## Add AC traces.

Select **Add traces** tab , select **V** in the left window, then double-click on **C1** in the components list. AC voltage trace will be added to the traces list.



## Run AC.

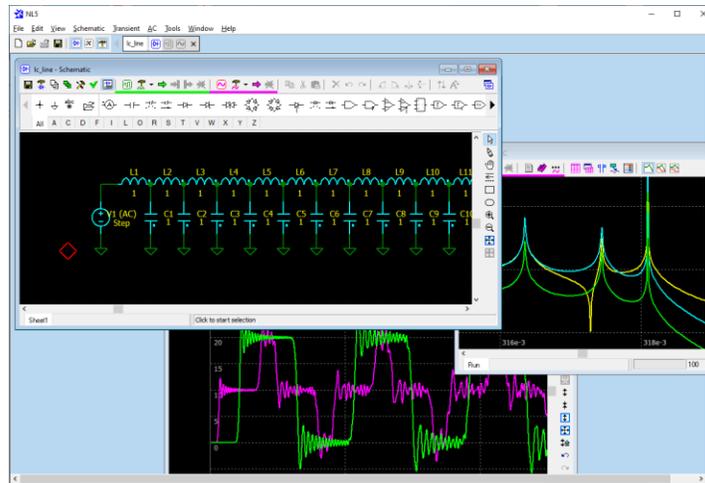
Click **Start AC**  to run AC simulation:



## II. User Interface

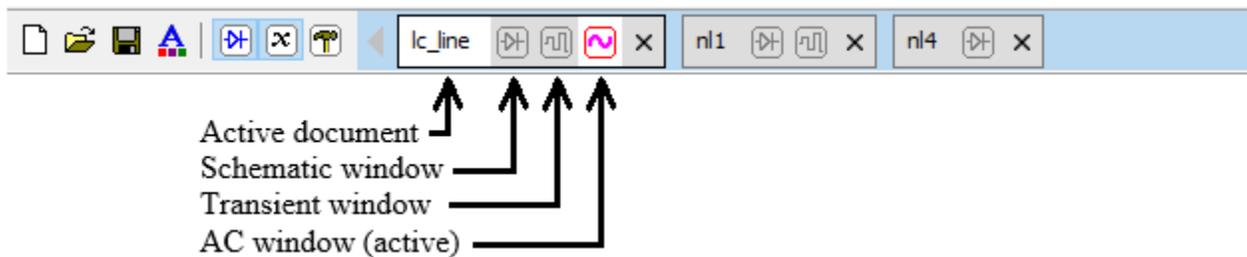
## Application

NL uses standard Windows multiple-document interface (**MDI**). All schematic-related windows (Schematic, Transient, AC) are created in “MDI child” mode, and are located inside the “main” window:



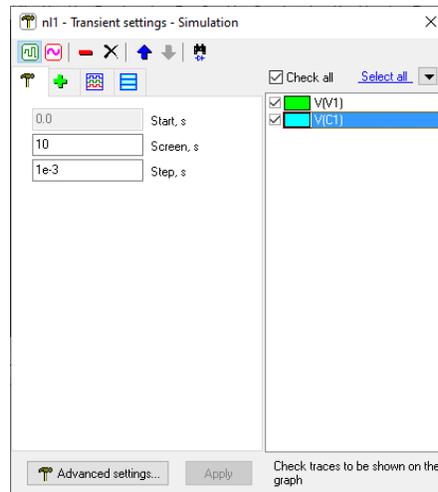
## Navigation bar

**Navigation bar** shows currently opened documents (schematics), with highlighted active document and window:



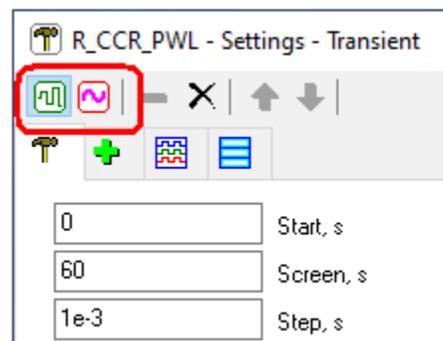
- Click on the document name to activate the document.
- Click on the window icon to activate document and window.
- Click  to close schematic.

## Settings window

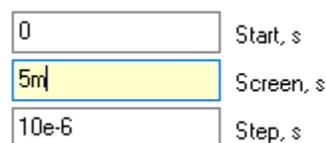


**Settings window** shows settings for **Transient** and **AC** simulation and window window. **Settings window** always shows information for active document (schematic). The name of active schematic is shown in the header of **Settings window**.

When **Transient** or **AC** window is activated, **Setting window** will automatically show settings for that window. You can also select desired settings clicking Transient or AC settings button (top left):



If some parameters in **Settings window** are modified, their background color will change to light-yellow:



Press **Enter** or click **Apply** to accept changes.

**Settings window** can be always opened.

## Context menus

In some places, mouse right-click will open context pop-up menu window. The content of the context menu depends on current window, selection, position of mouse pointer, etc.

## Shortcuts

- F1** – Help (context help on a specific window, selected component, C function, etc.).
- F2** – show Schematic window.
- F3** – show Components window.
- F4** – show Variables window.
- F5** – show Transient window.
- F6** – start transient.
- F7** – continue transient.
- F8** – show AC window.
- F9** – start AC analysis.
- F10** – show Settings window.

## Mouse

Most of mouse and mouse wheel operations are similar in **Schematic**, **Transient**, and **AC** windows.

Some options of mouse wheel operation can be changed in **Preferences/Document/Schematic/Mouse**, and **Preferences/Document/Graphs/Mouse**.

## Data format

**Data format** used in NL mostly complies with common engineering and scientific practice.

### Numbers

**Boolean** number can be entered as `false` or `true` (case-insensitive):

```
bool i = true;
bool retvalue = FALSE;
```

When converted to other types, `true` is considered as 1, `false` as 0. When other types are converted to `bool`, non-zero value is considered as `true`, zero value as `false`.

**Integer** (`int`, `int64`) number can be entered in decimal, binary, octal, or hexadecimal formats.

**Binary:** use `'0b'` or `'0B'` prefix, then use digits 0 and 1:

```
0b1111111111, 0B10101010, 0b10
```

**Octal:** start number with prefix 0 (zero), then use digits 0...7:

```
0377, 0123456
```

**Hexadecimal:** use `'0x'` or `'0X'` prefix. Then use 0...9, and capital or low-case A, B, C, D, E, F:

```
0xFF, 0X10aa, 0x10000
```

If a value of a number exceeds 32-bit range, it will be automatically converted to `int64` type. Use `i64` suffix to explicitly define 64-bit integer:

```
0i64, 0xfffffffffi64
```

**Floating point** (`float`, `double`) number can use exponential multipliers **E** or **e**, or **case-sensitive** letter multipliers:

Letter	Multiplier
T	$10^{12}$
G	$10^9$
M	$10^6$
k, K	$10^3$
m	$10^{-3}$
u	$10^{-6}$
n	$10^{-9}$
p	$10^{-12}$
f	$10^{-15}$

For example:

```
1.3e+3, 47E-9, 100k, 0.33u
```

Letter multiplier can be used instead of a decimal point. Zero before decimal point or letter multiplier can be omitted:

```
1k3, .47, n47
```

Infinite value is denoted by:

```
inf
```

**Complex** number consists of real and imaginary parts of floating point type. 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:

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

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.

All numerical component parameters, and most of other parameters in NL5 are floating point (`double`). Those parameters can be entered in any format, however after that they are automatically converted and stored in the floating point format.

When floating point 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
```

To access component's parameter in the function, use component name followed by dot '.' and parameter name:

```
R1.R
V2.slope
C123.IC
```

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

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

If current component model does not have parameters, a component model name will be used:

```
A1 = Amperemeter
```

**Schematic variable.** Schematic variable (a variable defined in the **Variables window**) name has the same format as a component, except it does not have parameters. For example:

```
Freq
X1.var
```

**Trace.** The basic name of transient or AC trace 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)
```

## Operators

NL supports the following arithmetic and logical operators:

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

and type-casting operators:

(bool) (int) (int64) (float) (double) (complex)

See **NL5 User's Reference** for details.

## Functions

NL offers many standard and NL5-specific functions. The functions can be used in **Function** model of some components.

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/she is more comfortable with. The following functions are available:

sin	sqrt	mag, abs	par	sum
cos	sqr	phase	random, rand	mean
tan, tg	sq	re	gauss	max
asin	pow	im	limit, lim	min
acos	pwr	sign	islow	bool
atan	exp	round	ishigh	int
atan2	ln, log	floor	db	int64
	lg, log10	ceil		float
	lb, log2			double
				complex

Please note that argument of functions **sin**, **cos**, **tan**, **tg** is in radians, not degrees. Similarly, functions **asin**, **acos**, **atan**, **phase** return radians, not degrees.

See **NL5 User's Reference** for details.

## Expressions

Expression may consist of:

- Numbers.
- Predefined constants.
- Names of components, parameters and variables.
- Operators.
- Functions.
- Parentheses with unlimited nesting level.

For example:

```
2*2
2<<3
sin(2*PI*f) // "f" is schematic variable.
max(R1,R2,R3)
1/((R1+R2)*C1)
```

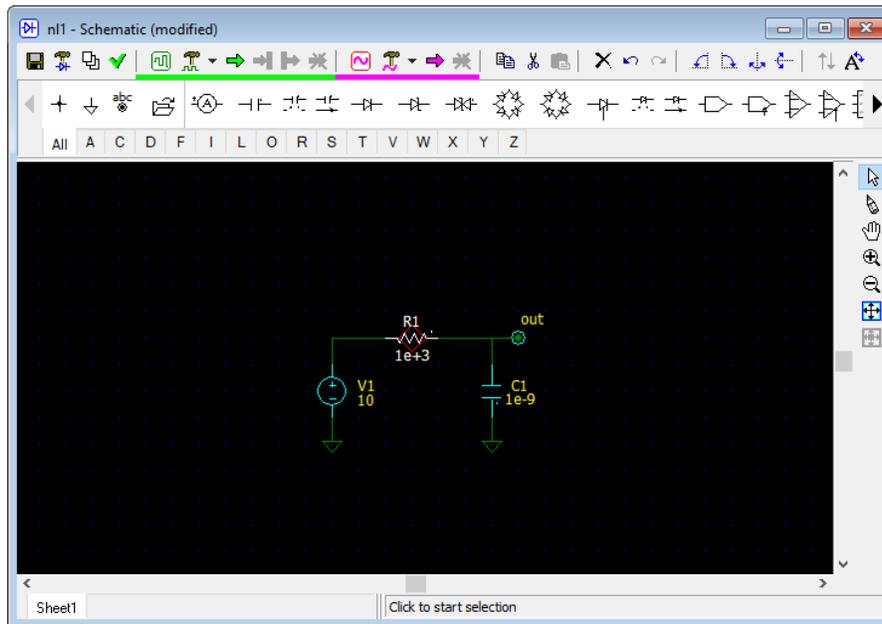
Expression can be used instead of number in most entry fields in the dialog boxes, and for some component parameters. When **Enter** key is pressed, or **OK** or **Apply** button (if exists) is clicked, the expression is immediately evaluated and replaced with the numerical value.

Please note that division of two integer values will produce double value, and a result will not be rounded. This is similar to how calculations are performed in calculators, Excel spreadsheets, etc. For example:

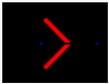
```
1/100 = 0.01
5/2 = 2.5
```

# III. Schematic

## Edit schematic



### Cursor



- **Double-click** - set cursor at that point and center it on the screen.
- **Left, Up, Right, Down** – change cursor direction, move cursor.
- **Click** on the cursor close to the cursor corner to change cursor direction.
- **Home** - center cursor on the screen.

### Wire



- Press **Space** or click **Wire**  to switch to **Wire** mode.
- **Click** and drag to draw wire.
- **Left, Up, Right, Down** – change cursor direction, draw wire.
- Press and hold **Ctrl** to draw diagonal wire.
- Press **Space** or **ESC** to switch back to **Selection** mode.

### Ground



- Press **G** or click  on the **Components bar**.

## Connection



- Press '.' (dot) or click  on the **Components bar**.

Three wires coming to one point are connected automatically: a connection point will be automatically added during schematic check. Two crossing wires are not connected by default and should be connected manually. All unnecessary connection points will be automatically removed during schematic check. **Warning:** diagonal wire cannot be connected to the wires it is crossing.

## Component



- Select tab with component letter on the **Components bar**, click on the component.  
**OR**
- Press component letter key (for example 'R') one or more time to select required component.
- Move component by keyboard/mouse, rotate, mirror, flip, select view if needed.
- **Right-click** on the component to see context menu, or use toolbar buttons or shortcuts to do the following:
  -  or **Ctrl-L** – rotate component left.
  -  or **Ctrl-R** – rotate component right.
  -  or **Ctrl-M** – mirror component.
  -  or **Ctrl-F** – flip component.
  -  or +/- key – show next component view (if applicable).
- Press **Enter** to place component or press **Del** or **ESC** to cancel.

When component is placed, click on the component to select it, then use the following ways to access component editing commands:

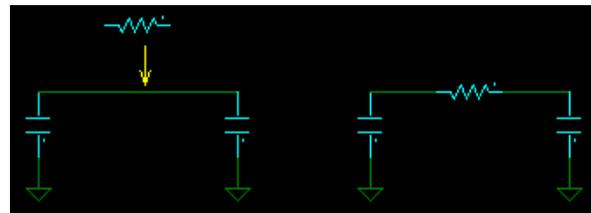
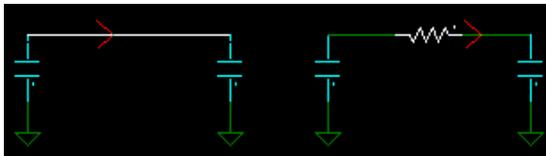
- toolbar buttons,
- keyboard shortcuts,
- **Main menu/Edit, Main menu/Edit/Selection,**
- **Right-click** on the component to see pop-up context menu.

The following component editing commands are available:

-  – set component at 45 degrees (available for some R, L, C, S, and D components).
-  or **Ctrl-T** – rotate attribute of selected component.
-  or +/- key – show next component view (if applicable).

-  or **Ctrl-L** – rotate component left.
-  or **Ctrl-R** – rotate component right.
-  or **Ctrl-M** – mirror component.
-  or **Ctrl-F** – flip component.
- **Double-click** on the component to edit component parameters in the **Components window**. Then, if you finish editing parameters by pressing **Enter** or **Esc**, you will switch back to the schematic.

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. Similarly, when component is moved/copied above existing wire, a piece of the wire underneath the component will be automatically removed:



## Label



- Press **Enter** or click  on the **Components bar**.
- For new label, enter label **Name** and **Description** (optional), click **OK**.
- For existing label, enter label **Name** or select from the list of existing labels, click **OK**.
- **Double-click** to edit label parameters in the **Components window**.

## Zoom and scrolling

- Click  to switch to **Scrolling** mode, then click and drag to scroll the screen.  
**OR**
- Press and hold **Shift**, then click and drag to scroll the screen.
- **Shift-Left, Shift-Up, Shift-Right, Shift-Down** - scroll the screen.
- **Ctrl-mouse wheel** - scroll horizontally.
- **Shift- mouse wheel** - scroll vertically.
-  or **PgUp** - zoom-in.
-  or **PgDo** - zoom out.
-  or **Ctrl-Home** - fit all schematic to the screen.
-  or **Shift-Home** - fit selection to the screen.
- **Mouse-wheel** – zoom in/out.

## Selection

- **Click** on the schematic element to select.
- Point on empty space, **click** and drag to select a block (rectangle).
- Press and hold **Ctrl** while selecting – add to existing selection.
- **Ctrl-A** – select all schematic.
- **Right-click/Select net** - selects all wires connected to the selection either directly, or through labels (including other sheets).
- **Click** on empty space or press **Esc** to unselect (press **Esc** twice to unselect a block).
  
- **Left, Up, Right, Down** – move selected block.
- **Click** and drag to move element or selection.
- **Click** and drag to move attribute of selected component.
- **Shift + Click** and drag to move element or selection with **rubber bands**.
- **Ctrl + Click** and drag to copy element or selection.
- **Ctrl + Shift + Click** and drag to copy element or selection with **rubber bands**.
- **Right-click/Disable** or **Ctrl-D** to disable selection.
- **Right-click/Enable** or **Ctrl-E** to enable selection.

Disabled schematic elements are shown in “disabled” color and are not used for simulation. Disabling elements allows temporarily exclude elements from simulation without deleting.

-  or **Ctrl-C** – copy selection.
-  or **Ctrl-X** – cut selection.
-  or **Ctrl-V** – paste selection.
-  or **Del** - delete selection.
-  or **Ctrl-L** – rotate selection left.
-  or **Ctrl-R** – rotate selection right.
-  or **Ctrl-M** – mirror selection.
-  or **Ctrl-F** – flip selection.
-  - next view of selected component.

Press +/- to go through all applicable combinations of **View** , **Mirror** , and **Flip** .

## General commands and shortcuts

- Press **Tab** to change attributes display for all components.
- **Right-click** to open context pop-up menu.
-  or **Ctrl-Z** – Undo.
-  or **Ctrl-Y** – Redo.

## Components

All available components are shown in the **Components bar**:



Each **component** has a letter associated with it. When a new component is placed, its **name** consists of a letter and number.

**Label** is a special type of component: there can be many labels with the same name in the schematic. All labels with the same name are electrically connected.

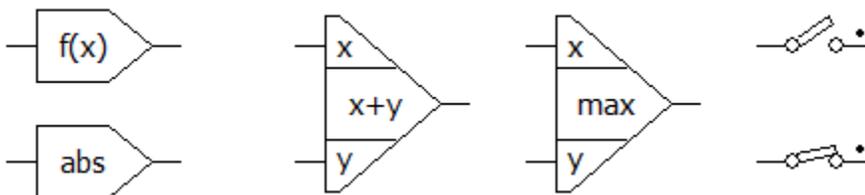
Labels can be used:

- To connect different points of the schematic without wire.
- As a simulation probe (**V trace**).
- As a voltage source.

**Symbol.** Most components have just one symbol associated with it, which cannot be changed. Some components may have several **views**, when the symbol is modified depending on different pin locations (for example, location of positive/negative input pins), and different functionality (for example, inversion of logical signals). For logical gates, changing **view** may also be used to change gate logical function (AND, OR, XOR).

Select component and click  to change a **view**, or press '+'/'-' keys to go through all applicable combinations of **view** , **mirror** , and **flip** .

Also, some components may have different symbols for different component **model** or component parameters. For example, symbol of the **Function** component indicates functionality of a selected model, and symbol of the **Switch** component indicates switch position in non-active state:



**Model.** Component **model** defines specific component functionality. For example, voltage source models include **Pulse**, **Sin**, **Step**, **File**, and more. Select component model in the drop-down list above parameters.

**Parameters.** Component parameters may be of different type: floating point, text, etc. Floating point parameter can be entered as an expression, for example:

$$R1.R*2.5$$

Such an expression will be immediately calculated and replaced by a resulting value.

**Function.** Some component parameters represent a **function**. Function is an expression which is recalculated at every transient or AC calculation step. Function can use the following variables:

$t$  – current transient time, s.

$f$  – current AC frequency, Hz.

$w$  – angular AC frequency,  $w = 2\pi f$ .

$s$  or  $p$  – Laplace parameter,  $s = p = j*2\pi f$ .

$x, y$  – 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.

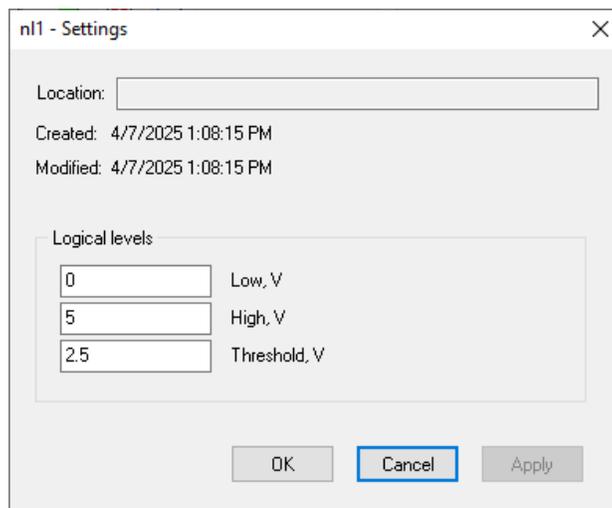
$P(name)$  – power on the component  $name$ . P trace should be available for the component.

For example:

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

**Blank parameter.** Some parameters of floating point type may be **blank** (empty), which means the value is not defined. For example, blank **IC** (initial condition) parameter of capacitor means that its initial voltage at  $t=0$  is not defined and will be determined during DC operating point calculation.

**Logical levels and threshold** for all components are defined in the **Schematic settings**  window:



**Attributes.** There are 2 **attributes** of the component that can be displayed on the schematic: **Name** and **Value**.

**Name** is automatically assigned to the component when it is placed.

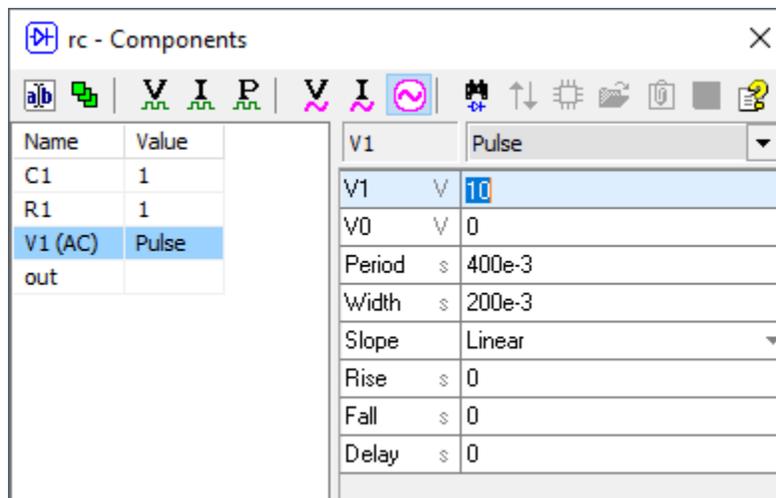
**Value** content depends on the component type. By default, it is the first parameter of the model, model name, or empty.

Press **Tab** to toggle attributes display mode for all components in the schematic:

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

To move attributes, select component first, then click and drag the attribute.

Component model and parameters can be selected and edited in the **Components windows**:



**Double-click** on the component or press **F3** to open or switch to the **Components window**. If **double-click** on the component in the schematic window was used, edit parameter and press **Enter** to switch back to schematic window.

Operations available for selected component are available on the toolbar.

## Working with special component types and models

### PWL model

**PWL** (PieceWise Linear) model describes non-linear characteristic of the component with piecewise linear approximation.

Please note that **pwl** parameter of the **PWL** model typically specifies PWC (PieceWise Constant) function, which represents sensitivity (derivative) of PWL function. For instance, **pwl** parameter of a resistor specifies  $R(V)$ , which is PWC function, while resistor  $I(V)$  characteristic is PWL function. The parameter is still called **pwl** (not **pwc**) for historical reasons.

The following table shows PWC (**pwl** parameter) and corresponding PWL functions for NL5 components:

Component	PWC	PWL
Resistor, diode, zener	$R(V)$	$I(V)$
Resistor	$R(I)$	$V(I)$
Capacitor	$C(V)$	$Q(V)$
Inductor	$L(I)$	$H(I)$
Voltage controlled voltage source, OpAmp	$K(V)$	$V(V)$
Current controlled voltage source	$K(I)$	$V(I)$
Voltage controlled current source	$K(V)$	$I(V)$
Current controlled current source	$K(I)$	$I(I)$

For voltage/current controlled linear components (R, C, L, and amplifier) **pwl** parameter is also PWC function, describing change of component main parameter (R, C, L, K) with control signal:

Component	PWC
Voltage controlled resistor	$R(V_{in})$
Current controlled resistor	$R(I_{in})$
Voltage controlled capacitor	$C(V_{in})$
Current controlled capacitor	$C(I_{in})$
Voltage controlled inductor	$L(V_{in})$
Current controlled inductor	$L(I_{in})$
Voltage controlled amplifier	$K(V_{in})$
Current controlled amplifier	$K(I_{in})$

**PWL** model of a resistor is described here as an example.

**pwl** parameter is a comma-separated string, describing PWC function:

$R_0, V_1, R_1, V_2, R_2, \dots, V_N, R_N$

where:

$R_0$  is resistance while voltage across the resistor is less than  $V_1$ .

$R_1$  is resistance while voltage across the resistor is between  $V_1$  and  $V_2$ .

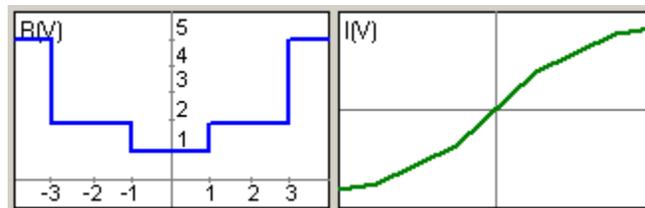
...

$R_N$  is resistance while voltage across the resistor is greater than  $V_N$ .

Values  $V_1 \dots V_N$  should be given in ascending order. Resulting PWL characteristic is calculated automatically, and always goes through the origin (0,0). Please note that only numbers can be used in **pwl** parameter: formulas are not allowed.

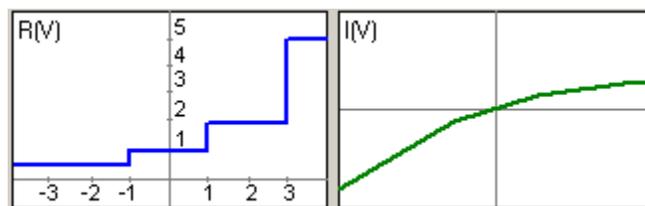
PWC function can be symmetrical or non-symmetrical. **Symmetrical** function is defined only in the interval from zero to plus infinity; the negative part of PWC function is symmetrical to positive one:

**pwl** = 1, 1, 2, 3, 5

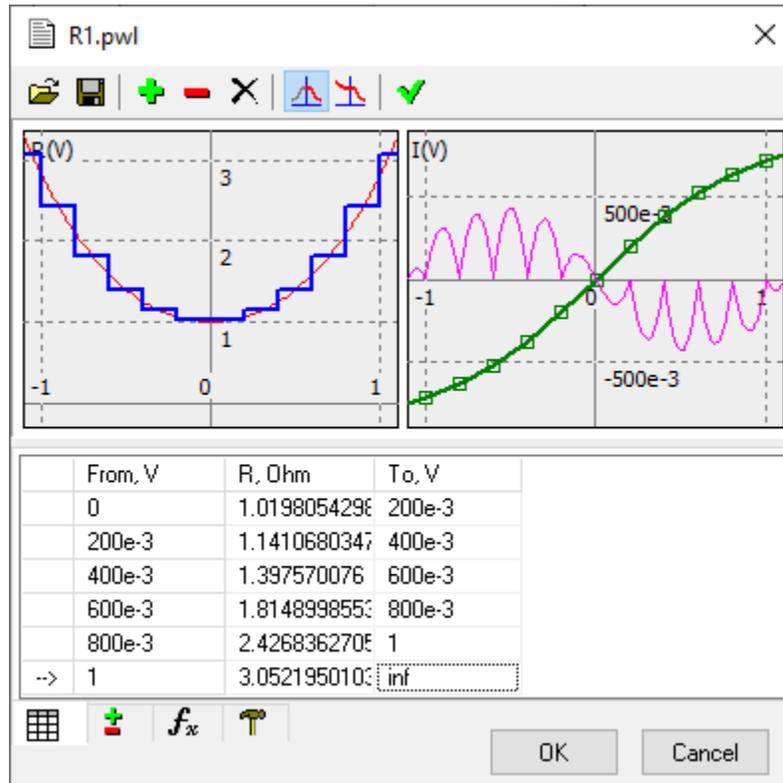


**Non-symmetrical** function is defined from minus to plus infinity. One of the argument points must be **zero**: it serves as an indicator of non-symmetrical characteristic:

**pwl** = .5, -1, 1, 0, 1, 1, 2, 3, 5



To edit **pwl** parameter, select the parameter and click  to open **PWL** window:



**Right-click** on the graphs area to see context menu with relevant commands.

The following tabs are available:

**Table** . Edit PWL data in the table.

 in the first column indicates selected row.

Select cell and edit the number: **From** value is updated automatically.

 - symmetrical PWC. The first **From** value will be **zero**.

 - non-symmetrical PWC. The first **From** value will be **-inf**.

 - split selected row.

 - remove selected row.

 - clear all data.

 or **Enter** - refresh graphs.

## List model (voltage and current source)

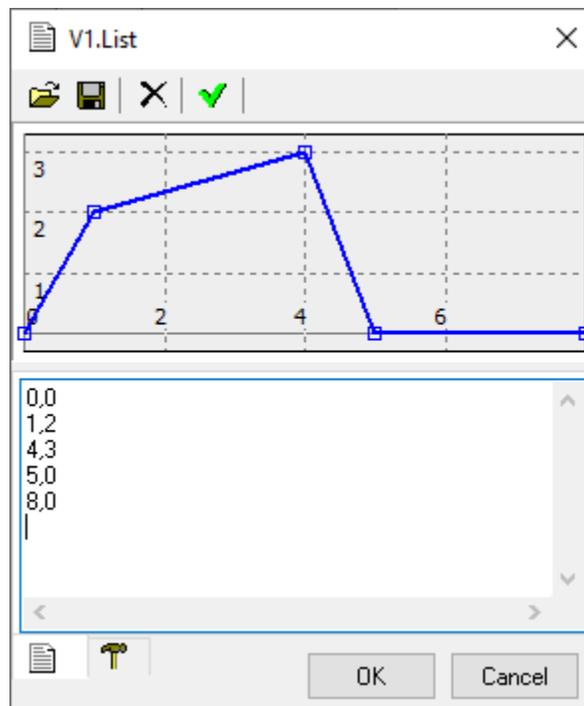
**List** model describes piecewise linear voltage or current source.

**List** parameter is a comma-separated string with time/value pairs:

$$T_1, V_1, T_2, V_2, \dots, T_N, V_N$$

Signal value between specified points is linearly interpolated. Signal value before  $T_1$  is  $V_1$ , signal value after  $T_N$  is  $V_N$ . Values  $T_1 \dots T_N$  should be given in ascending order. Although the signal is defined on the interval  $T_1 \dots T_N$ , it can be repeated continuously, or delayed by setting component parameters **Delay** and **Cycle**. Please note that only numbers can be used in **List** parameter: formulas are not allowed.

Edit **List** parameter manually, or click  to open **List** window:



Enter data in the table as comma-separated time/value pairs. The data will be automatically sorted in ascending order when refreshed.

## List model (switch and logic generator)

**List** model describes switching sequence of **Switch** component, and logical signal of **Logic Generator** component.

**List** parameter is a comma-separated string with time/value pairs:

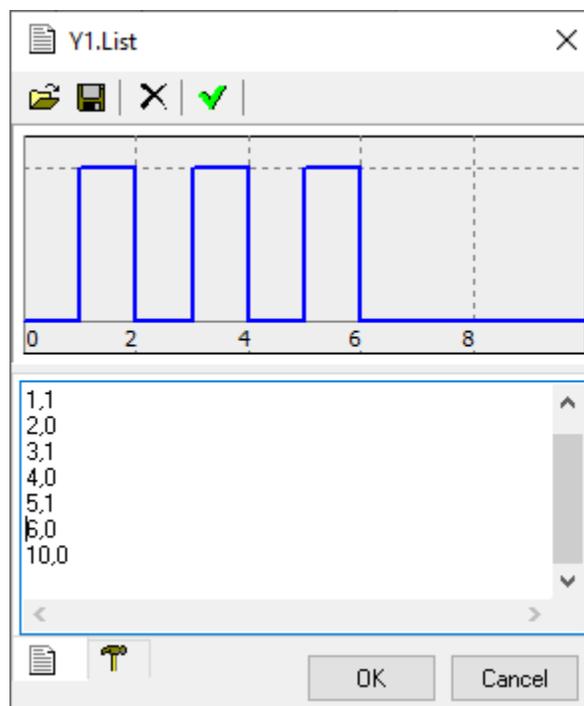
$$T_1, S_1, T_2, S_2, \dots, T_N, S_N$$

where  $T_i, S_i$  pair defines state of the signal at specified time:

- Positive value corresponds to **On** state of the switch, or **High** state of the logic generator.
- Zero or negative value corresponds to **Off** state of the switch, or **Low** state of the logic generator.

Signal state before  $T_1$  is  $S_1$ , signal state after  $T_N$  is  $S_N$ . Values  $T_1 \dots T_N$  should be given in ascending order. Although the signal is defined on the interval  $T_1 \dots T_N$ , it can be repeated continuously, or delayed by setting component parameters **Delay** and **Cycle**. Please note that only numbers can be used in **List** parameter: formulas are not allowed.

Edit **List** parameter manually, or click  to open **List** window:



Enter data in the table as comma-separated time/value pairs. The data will be automatically sorted in ascending order when refreshed.

## Table model

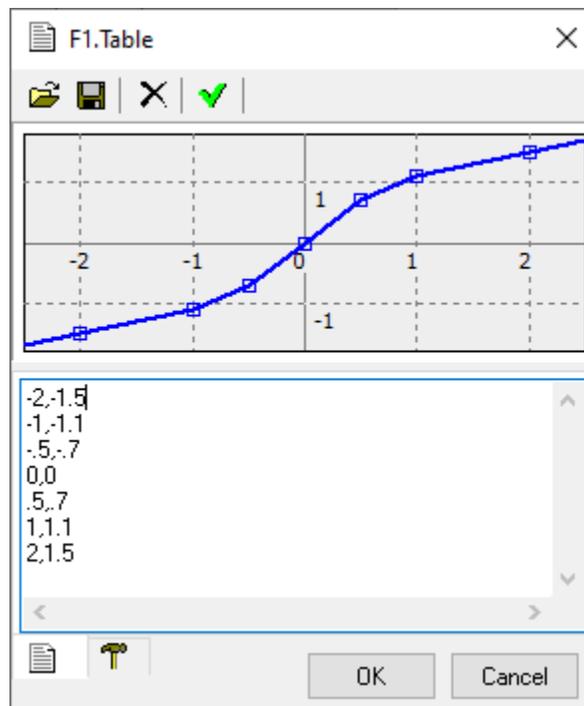
**Table** model describes look-up table of the **Function** component.

**Table** parameter is a comma-separated string with input/output pairs:

$$X_1, Y_1, X_2, Y_2, \dots, X_N, Y_N$$

where  $x_i, y_i$  pair defines input value  $x$  and output value  $y$ . Output value between specified points is linearly interpolated. Output value below  $x_1$  is linearly extrapolated using  $x_1 \dots x_2$  interval data, output value above  $x_N$  is linearly extrapolated using  $x_{(N-1)} \dots x_N$  interval data. Values  $x_1 \dots x_N$  should be given in ascending order. Please note that only numbers can be used in **Table** parameter: formulas are not allowed.

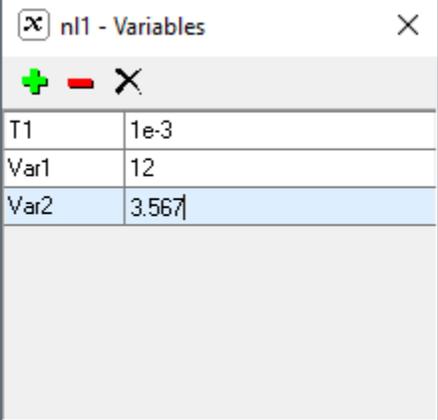
To edit **Table** parameter, select the parameter and click  to open **Table** window:



Enter data in the table as comma-separated input/output pairs. The data will be automatically sorted in ascending order when refreshed.

## Schematic variables

Schematic variable is a floating point variable which belongs to the schematic. The variable and its value can be defined in the **Variables window**:



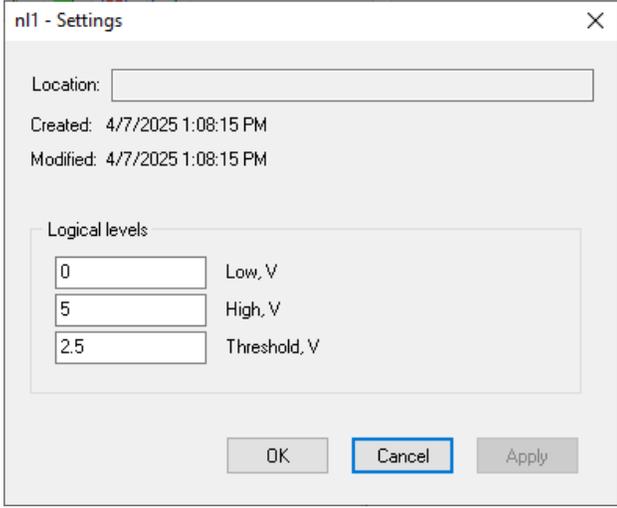
nl1 - Variables	
T1	1e-3
Var1	12
Var2	3.567

Variables can be used in expressions, for example when defining component parameters:

$$R1.R = \text{Var1} * 2$$

## Schematic settings

Schematic settings  window:



n1 - Settings

Location:

Created: 4/7/2025 1:08:15 PM  
Modified: 4/7/2025 1:08:15 PM

Logical levels

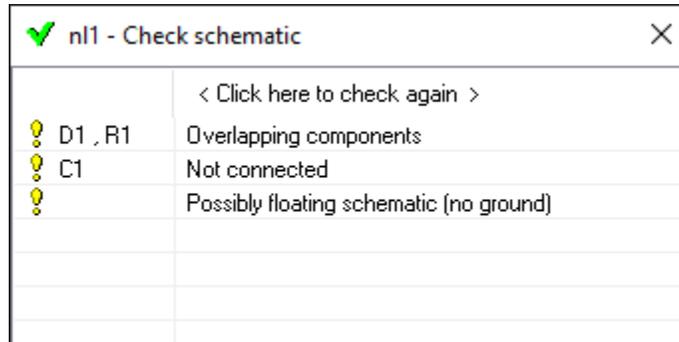
<input type="text" value="0"/>	Low, V
<input type="text" value="5"/>	High, V
<input type="text" value="2.5"/>	Threshold, V

OK Cancel Apply

Specify logical levels and threshold for all components in the schematic.

## Check schematic

**Check schematic**  command performs extensive check of the schematic for potential problems, and reports results in the window:



Also, the check function removes unnecessary connection points and adds connection points where needed (three wires).

Click on the row to select (highlight) a schematic element(s) for which the problem was detected.

Go to **Preferences**  , **Warnings** tab to select/unselect reported issues.

**Right-click** to access check-related commands in the pop-up context menu.

Schematic check is automatically performed at **Transient** and **AC** simulation start. The problems reported by the function do not prevent running simulation and typically do not affect simulation results. However, they could indicate not accurate schematic design and should be fixed if possible.

# IV. Transient

## Algorithm

NL and NL5 perform simulation with ideal and piecewise linear (PWL) components. For instance, an ideal diode is either short circuit (zero resistance), or open circuit (infinite resistance), so that its PWL representation consists of just two linear segments. As long as all of the components are staying within their current segment, the circuit is described by the system of linear differential equations. The system is modified only at the moments when at least one component changes its linear segment.

**Start.** When simulation starts at  $t=0$ , a **Direct Current (DC) operating point** is calculated first. Some of the components may have specified Initial Condition (IC): these ICs will be used during DC operating point calculation. If circuit has more than one steady state, it can be set to a desired state by defining proper ICs. After DC operating point is found, simulation continues as a sequence of **linear segment** simulations, with **switching** between them.

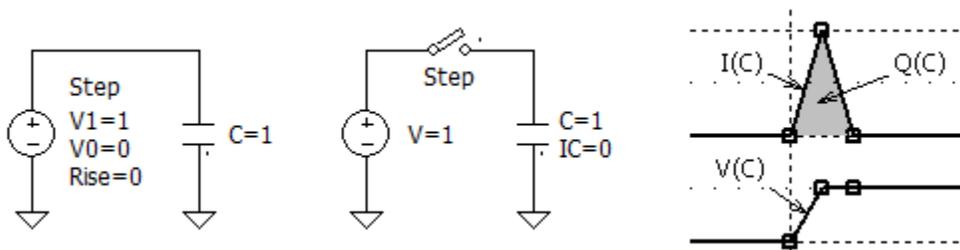
**Linear segment.** In the linear segment, the circuit is described by the system of linear differential equations, which is solved by Trapezoidal integration method. The method provides sufficient accuracy with good robustness and calculation speed. During linear segment simulation, the algorithm is performing “switching point detection”: checking conditions on all components that may change their state or linear segment.

**Switching.** When switching point is detected, the current linear range ends, **switching** algorithm performs instantaneous switching, and a new linear segment starts. Switching algorithm can be optimized for specific circuit by selecting from several options at **Transient settings/Advanced settings**.

Instantaneous switching of ideal components may produce infinite voltage or current pulses. For example, when capacitor is connected to voltage source through ideal switch, an infinitely short current pulse with infinite amplitude may occur. The area (integral over time) of such pulse is limited and is equal to the total charge coming to or out of the capacitor during switching. Similar situations may occur when current through the inductor is discontinued, which results in an infinite voltage pulse across the inductor. The integral of the voltage over time corresponds to a magnetic flux in the inductor. Such a pulse is known as Dirac pulse, or delta-function.

Infinite current or voltage pulses may also occur if **Rise** or **Fall** parameter of voltage or current sources is set to zero, producing similar infinitely short slope of voltage or current. Such a slope is also shown as a “jump” of voltage or current during minimal calculation step used at that moment.

In NL, 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.



**Simulation step.** Unlike many analog simulators, NL does not perform automatic step control. Selecting simulation step is user's responsibility. This gives user full control on simulation, although it requires certain experience and understanding of the process. The rule of thumb is keeping simulation step below smallest time constant in the circuit, otherwise the integration method may get unstable, and produce "numerical oscillations" when signal is "jumping" up and down at each simulation step.

Although simulation step is specified by user, NL 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 of the source component, interpolating the signal between two points, contains at least 4 steps.
- Delay time of transmission line and "delay" component contains at least 2 steps.

Also, simulation step can be reduced to detect switching point and perform accurate switching.

**Traces.** During simulation, NL only stores data for specified **traces**. Pause or stop simulation to add/remove traces, then continue simulation or start simulation again.

**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 all the data, some of the blocks currently storing the very beginning of the trace will be reassigned to a new data, and thus some traces may be "truncated" at the beginning. When this happens the first time, the warning message will be displayed in the status bar of **Transient window**.

By default, NL saves all calculated data for all traces. To reduce memory consumption, a simple data compression method can be used: if several data points in a row have the same amplitude, only first and last point of such sequence are being stored and shown.

## Running transient



- Go to **Settings window** (select desired Transient settings tab).



- **Start** transient (available on the **Schematic** and **Transient window** toolbars). When simulation is running, schematic cannot be edited, and component parameters/model cannot be changed. **Stop** or **Pause** simulation to do any changes.



- **Pause** transient (or press **Space**).



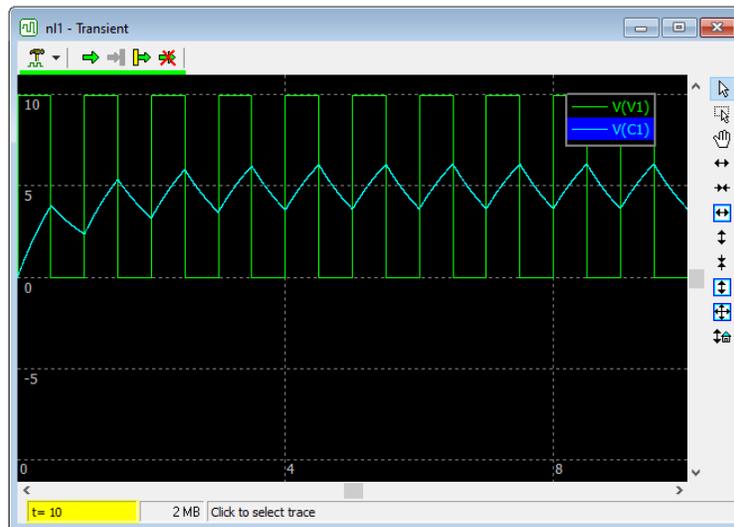
- **Continue** transient (or press **Space**).



- **Stop** transient. Clear extra memory allocated for simulation. Simulation cannot be continued.

Simulation is always paused at the right edge of the screen. Please note that simulation step is not adjusted to stop exactly at the screen edge, so the last calculated point could have time greater than screen edge. Press **Space** or click **Continue**  to continue simulation.

## Transient window



To add transient traces, go to **Add traces**  tab of **Settings window** .

Go to **Format traces**  tab of **Settings window**  to select trace type, color, scales, etc.

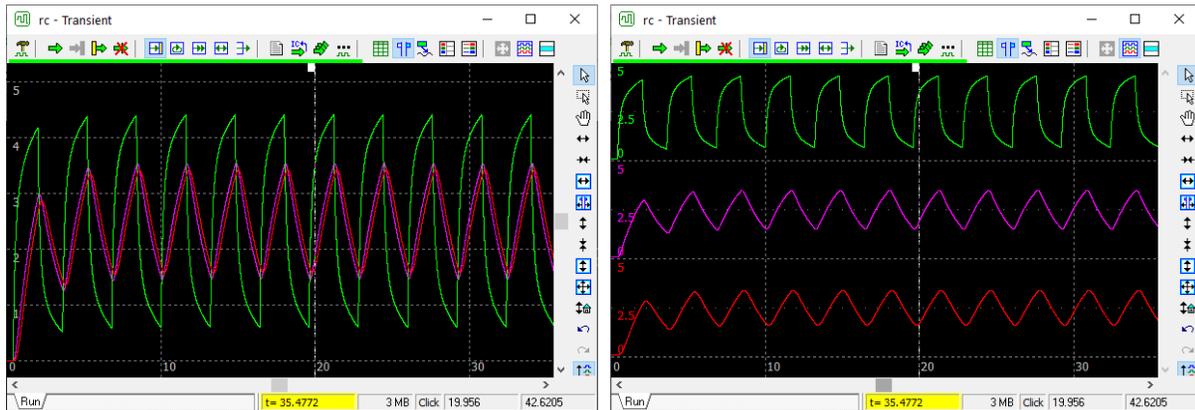
Go to **Screen**  tab of **Settings window**  to configure **Transient window** horizontal scale.

**Vertical scale.** Each trace is shown with its individual **Scale** and **Mid** values, specified at **Format traces**  tab of **Settings window** . Vertical scale and gridlines are shown for one selected trace, or several selected traces if their **Scale** and **Mid** values are the same. If several traces with different **Scale** or **Mid** values are selected, vertical scale is not shown.

All vertical zoom/scroll/reset operations (buttons, keys, mouse, mouse wheel) will change scales and offsets of all traces accordingly.

Click **Reset vertical scale**  to set **Scale** and **Mid** of all traces to default values.

Click **Separate traces**  button or press **Tab** to show each trace in its own smaller area separated by horizontal lines:



Separate traces Off

Separate traces On

**Mouse.** There are 3 modes of **mouse operation** in analog sections:

**Cursors**  mode. In this mode, **double-click** to center the screen to the mouse pointer position.

**Zoom**  mode (press and hold **Ctrl** key to switch to **zoom** mode temporarily). Click and drag to select horizontal, vertical, or rectangle area to be zoomed-in:



Move mouse pointer horizontally to select horizontal area.



Move mouse pointer vertically to select vertical area.



Move mouse pointer diagonally to select rectangle area.

**Scrolling**  mode (press and hold **Shift** key to switch to **scrolling** mode temporarily). Click and drag to scroll the screen.

**Mouse wheel:**

**Mouse wheel** – horizontal zoom (relative to mouse pointer position).

**Shift-mouse wheel** – horizontal scroll.

**Ctrl-mouse wheel** – vertical zoom (relative to mouse pointer position).

**Ctrl-Shift-mouse wheel** – vertical scroll.

**Status bar** shows current transient time, simulation status (running, paused), and amount of memory allocated for calculation and data.

**Legend**  shows list of traces currently displayed on the graph.

- **Click** on the border of the legend and drag to move the legend window.
- **Click** on the trace in the legend to select one trace.
- **Ctrl-click** to select more than one trace or un-select the trace.
- Go to **Preferences** , **Legend** tab to select legend style (colors, font).

### Keys and shortcuts:

**F5** – show transient window .

**F6** - start transient .

**F7** - stop transient .

**Space** – pause/continue transient.

**Up, Down** – vertical scroll.

**PgUp, PgDn** – vertical zoom , .

**Home** – vertical fit the screen, same as .

**Left, Right** – horizontal scroll.

**End** – center beginning of traces (set to the middle of the screen).

**Ctrl-End** – center end of traces.

**Shift-End** – center middle of the traces.

**Ctrl-PgUp, Ctrl-PgDn** – horizontal zoom , .

**Ctrl-Home** – horizontal fit the screen .

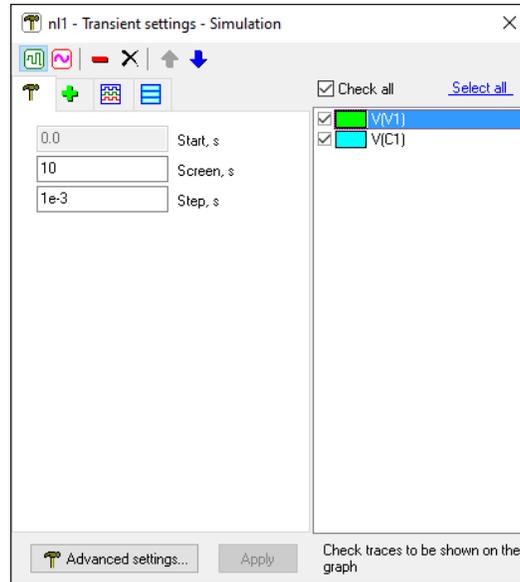
**Shift-PgUp, Shift-PgDn** – vertical and horizontal zoom.

**Shift-Home** – fit the screen .

**Tab** – separate traces .

## Transient settings

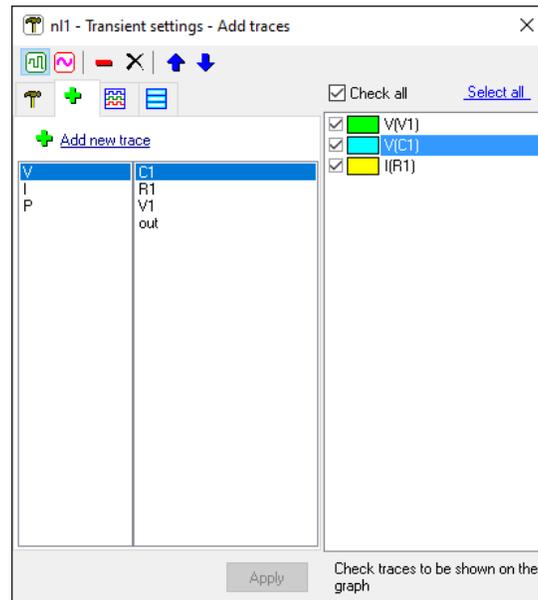
### Simulation



**Screen.** Screen size settings at simulation start (simulation always starts at  $t=0$ ).

**Step.** Maximum calculation step. Actual step may be reduced by the algorithm if needed.

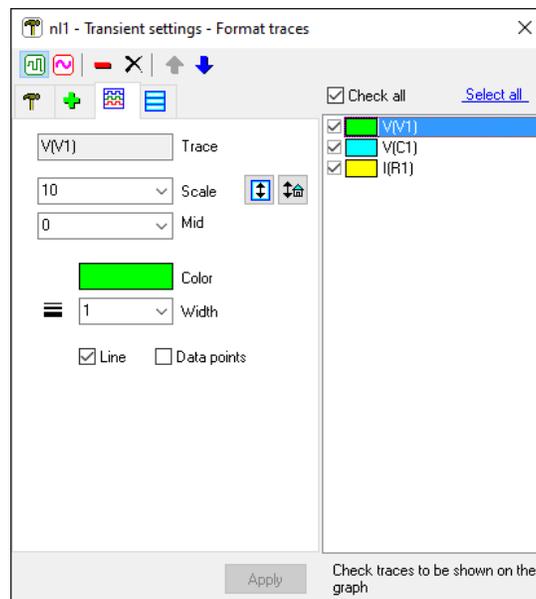
## Add traces



Select trace type to add: **V**, **I**, **P**.

Select one or more components in the list, then click  **Add new trace**, or **double-click** on the component in the list.

## Format traces



**Check** traces to be shown on the transient graph.

**Select** one or more traces (using **Ctrl** and **Shift** keys) to perform operations on selected traces.

Click **Select All** to select all traces. Traces can also be selected in the **Legend** area of the Transient window.

The top toolbar provides various operations on selected traces and trace data.

If more than one trace is selected, only parameters of the same value for **all** selected traces will be shown, otherwise parameter field will be blank. If colors are different, the color will be displayed like

this:  **Color**. If a new value is entered in the empty field, that value will be assigned to all selected traces.

**Scale, Mid** – scale values for analog trace. Click  to select from previously used values. Click

 to auto-scale, or  to reset vertical scale of selected traces.

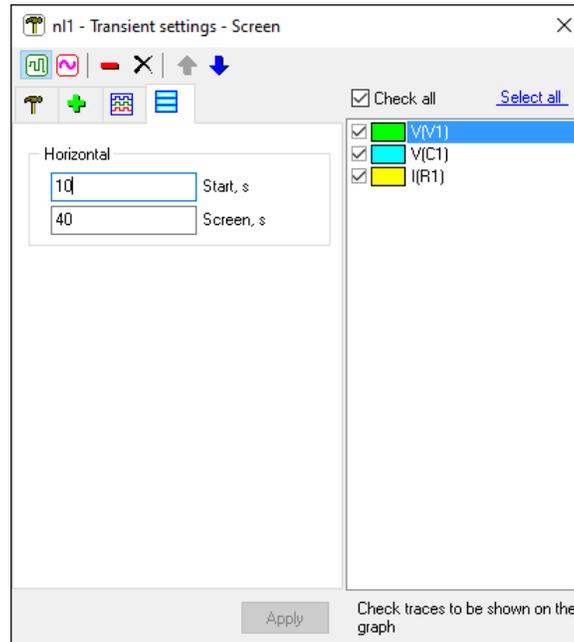
**Color** - click colored rectangle to select trace color.

**Width** - enter or select from drop-down list trace width in pixels.

**Line** – show lines.

**Data points** – show simulation data points.

## Screen



View and modify screen horizontal scale.

# V. AC analysis

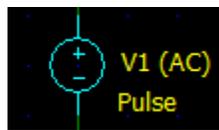
## Algorithm

There is only one method of AC analysis in NL: **Linearize schematic**.

**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, states of all components should be known. It can be done manually by setting Initial Conditions (IC) for 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**). DC operating point is calculated exactly as in transient analysis.

**AC source** . Any voltage source, current source, or label can be used as AC source. Component selected as AC source will be marked with “(AC)” text on the schematic and in the **Components window**:



R2	1
R4	1
V1 (AC)	Pulse

The AC source component may have any model (except **Label, and IC**): the model will be ignored for **Linearized schematic** method. 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.

## Running AC analysis



- Go to **Settings window** (select desired AC settings tab).

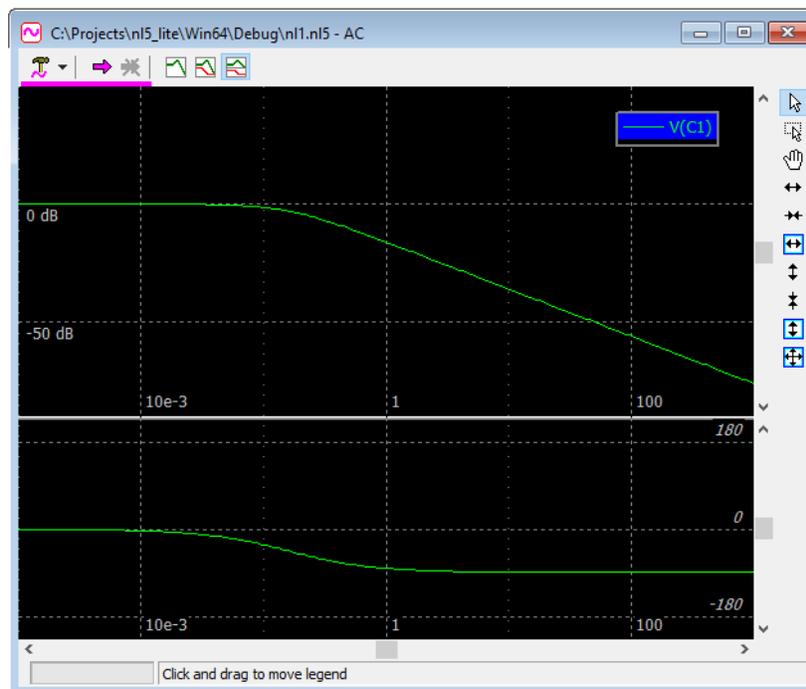


- **Start AC** (available on the **Schematic** and **AC window** toolbars). When simulation is running, schematic cannot be edited, and component parameters/model cannot be changed.



- **Stop AC**.

## AC window



To add AC traces and specify AC source, go to **Add traces**  tab of **Settings window** . Go to **Format traces**  tab of **Settings window**  to select trace color, scales, etc. Go to **Screen**  tab of **Settings window**  to configure **AC window** scales.

**Scales.** All traces are shown with the same scales, which can be linear or logarithmic.

**Phase display method selection:**

- **Phase off**  – do not show phase.
- **Phase On**  - show phase in the same window with other traces.
- **Phase separate**  - show phase in the select **Phase** section.

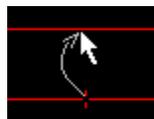
**Mouse.** There are 3 modes of **mouse operation**:

**Cursors**  mode: **double-click** to center the screen to mouse pointer position (magnitude only).

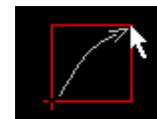
**Zoom**  mode (press and hold **Ctrl** key to switch to **zoom** mode temporarily). Click and drag to select horizontal, vertical, or rectangle area to be zoomed-in:



Move mouse pointer horizontally to select horizontal area.



Move mouse pointer vertically to select vertical area.



Move mouse pointer diagonally to select rectangle area.

**Scrolling**  mode (press and hold **Shift** key to switch to **scrolling** mode temporarily). Click and drag to scroll the screen.

### Mouse wheel:

**Mouse wheel** – horizontal zoom (relative to mouse pointer position).

**Shift-mouse wheel** – horizontal scroll.

**Ctrl-mouse wheel** – vertical zoom (relative to mouse pointer position).

**Ctrl-Shift-mouse wheel** – vertical scroll.

### Keys and shortcuts:

**F8** – show AC window  .

**F9** - start AC  .

**Tab** – toggle **Phase** display mode  .

**Up, Down** – vertical scroll.

**PgUp, PgDn** – vertical zoom  .

**Home** – vertical fit the screen  .

**Left, Right** – horizontal scroll.

**Ctrl-PgUp, Ctrl-PgDn** – horizontal zoom  .

**Ctrl-Home** – horizontal fit the screen  .

**Shift-PgUp, Shift-PgDn** – vertical and horizontal zoom.

**Shift-Home** – fit the screen  .

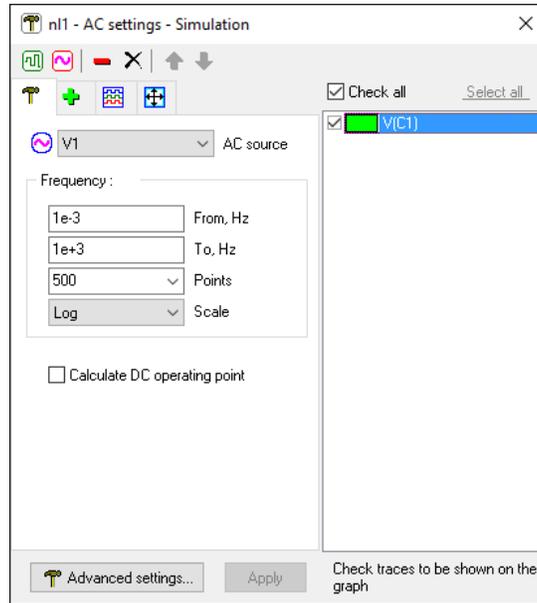
**Status bar** shows current frequency of AC simulation (progress).

**Legend**  shows list of traces currently displayed on the graph.

- **Click** on the border of the legend and drag to move the legend window.
- **Click** on the trace in the legend to select one trace.
- **Ctrl-click** to select more than one trace or un-select the trace.
- Go to **Preferences**  , **Legend** tab to select legend style (colors, font).

## AC settings

### Simulation



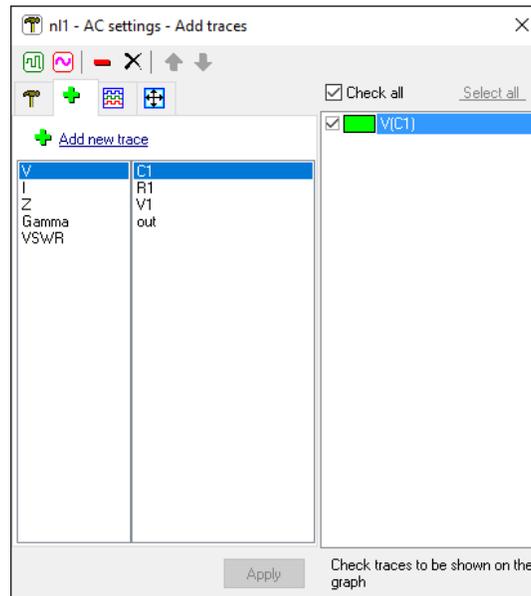
**AC source** – enter the name of AC source component or select the name from drop-down list.

Component selected as AC source will be marked with **(AC)** text on the schematic and in the **Components window**.

**Frequency** - select frequency, number of points, and scale type.

**Calculate DC operating point** – use this option for non-linear schematic if needed.

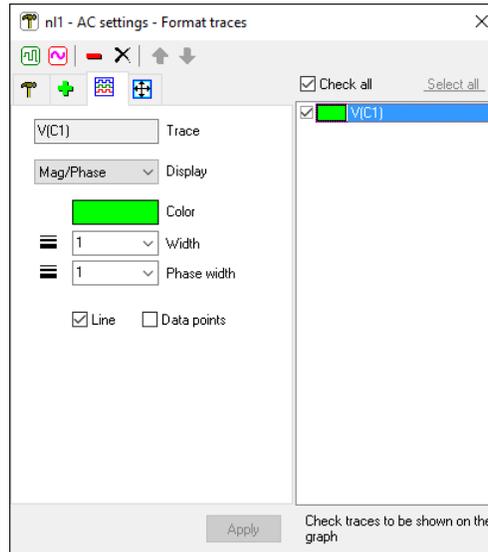
## Add traces



Select trace type to add: **V, I**.

Select one or more components in the list, then click  **Add new trace**, or **double-click** on the component in the list.

## Format traces



**Check** traces to be shown on the AC graph.

**Select** one or more traces (using **Ctrl** and **Shift** keys) to perform operations on selected traces. Click **Select All** to select all traces. Traces can also be selected in the **Legend** area of the AC window.

The top toolbar provides various operations on selected traces and trace data.

If more than one trace is selected, only parameters of the same value for **all** selected traces will be shown, otherwise parameter field will be blank. If colors are different, the color will be displayed like

this: . If a new value is entered in the empty field, that value will be assigned to all selected traces.

**Display** defines how the trace will be displayed:

- **Mag/Phase** - magnitude and phase.
- **Re** – real part only.
- **Im** – imaginary part only.

**Color** - click colored rectangle to select trace color.

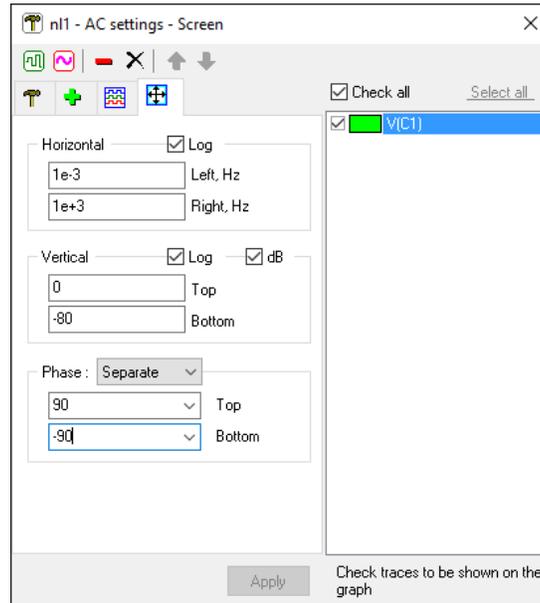
**Width** - enter or select from drop-down list trace width in pixels.

**Phase width** - enter or select from drop-down list phase width in pixels.

**Line** – show lines.

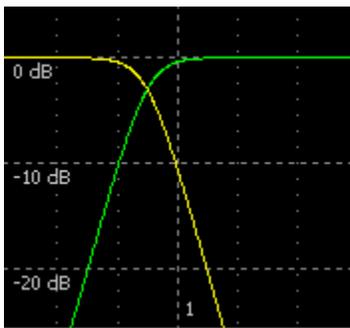
**Data points** – show simulation data points.

## Screen

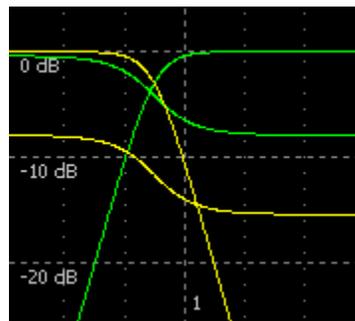


**Phase** - select phase display mode:

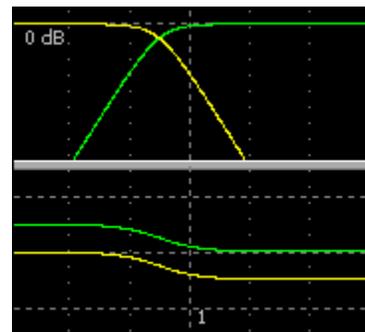
- **Off** - do not show phase.
- **On** - show magnitude and phase in the same area of the graph.
- **Separate** - show magnitude and phase in separate areas of the graph.



**Phase Off**



**Phase On**



**Phase separate**

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