

How to Build Macro-Models in Tina SPICE

Part 1: Text Macro-Model Creation

Text SubCircuit Creation

Symbol Creation

Macro-Model Creation

Part 2: Schematic Macro-Model Creation

Schematic SubCircuit Creation

Symbol Creation

Macro-Model Creation

Tim Green May 4, 2006

Part 1: Text Macro-Model Creation

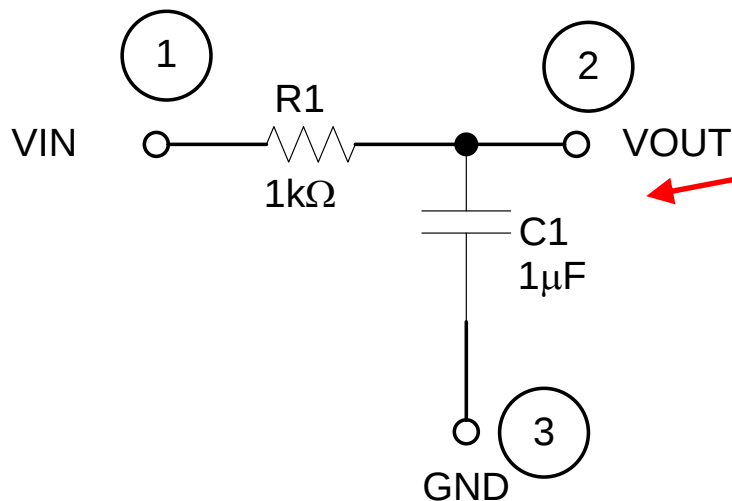
Text SubCircuit Creation

Symbol Creation

Macro-Model Creation

Part 1: Text Macro-Model Creation

Text SubCircuit Creation



Equivalent Schematic of Text SubCircuit
Circled numbers are Node numbers

* TG Simple 159Hz LPF

*Connections..... VIN VOUT GND

.SUBCKT LPF 1 2 3

R1 1 2 1k

C1 2 3 1u

.ENDS LPF

* Indicates Comment

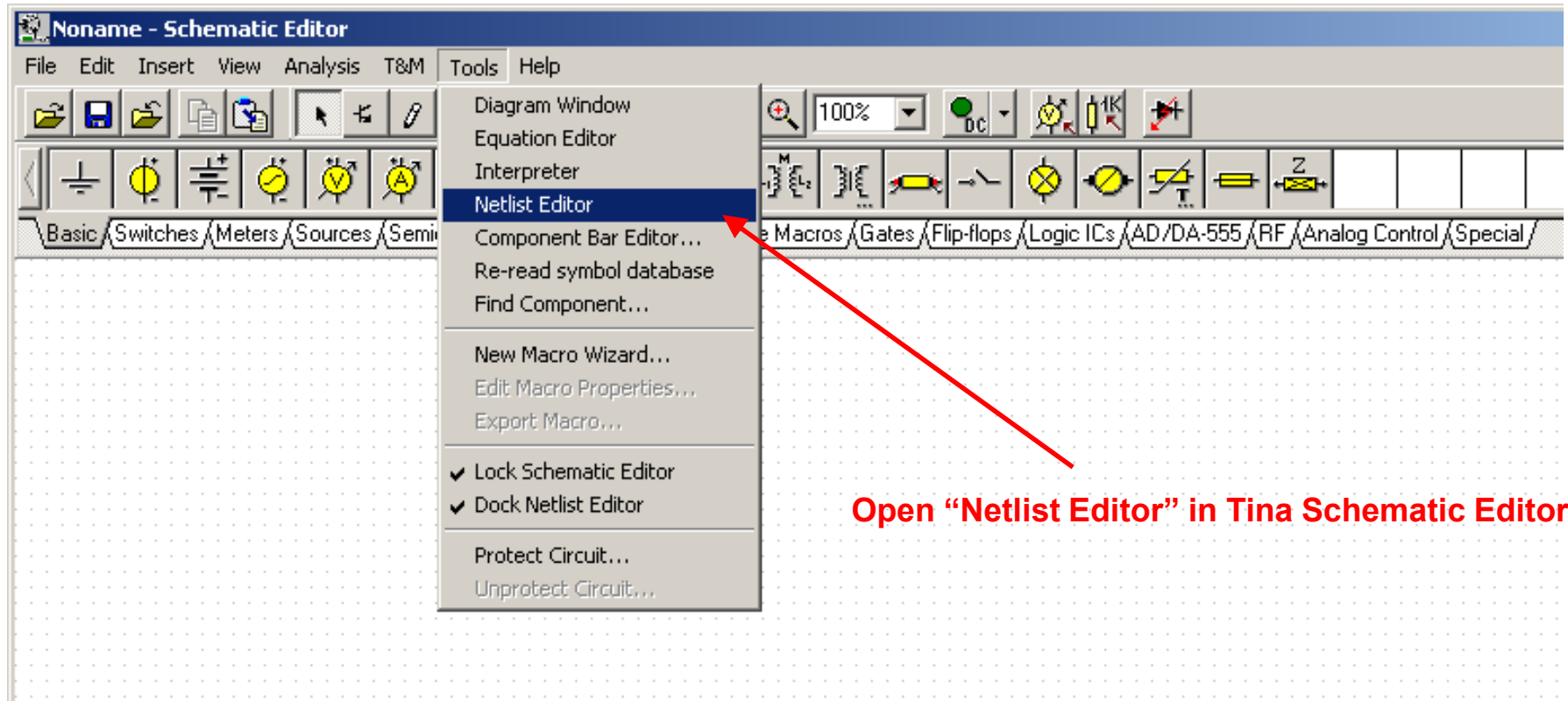
.SUBCKT statement w/subcircuit name (LPF)
All external Nodes to connect subcircuit (1,2,3)

Component types/names and Node connections

End subcircuit statement w/subcircuit name (.ENDS LPF)

Part 1: Text Macro-Model Creation

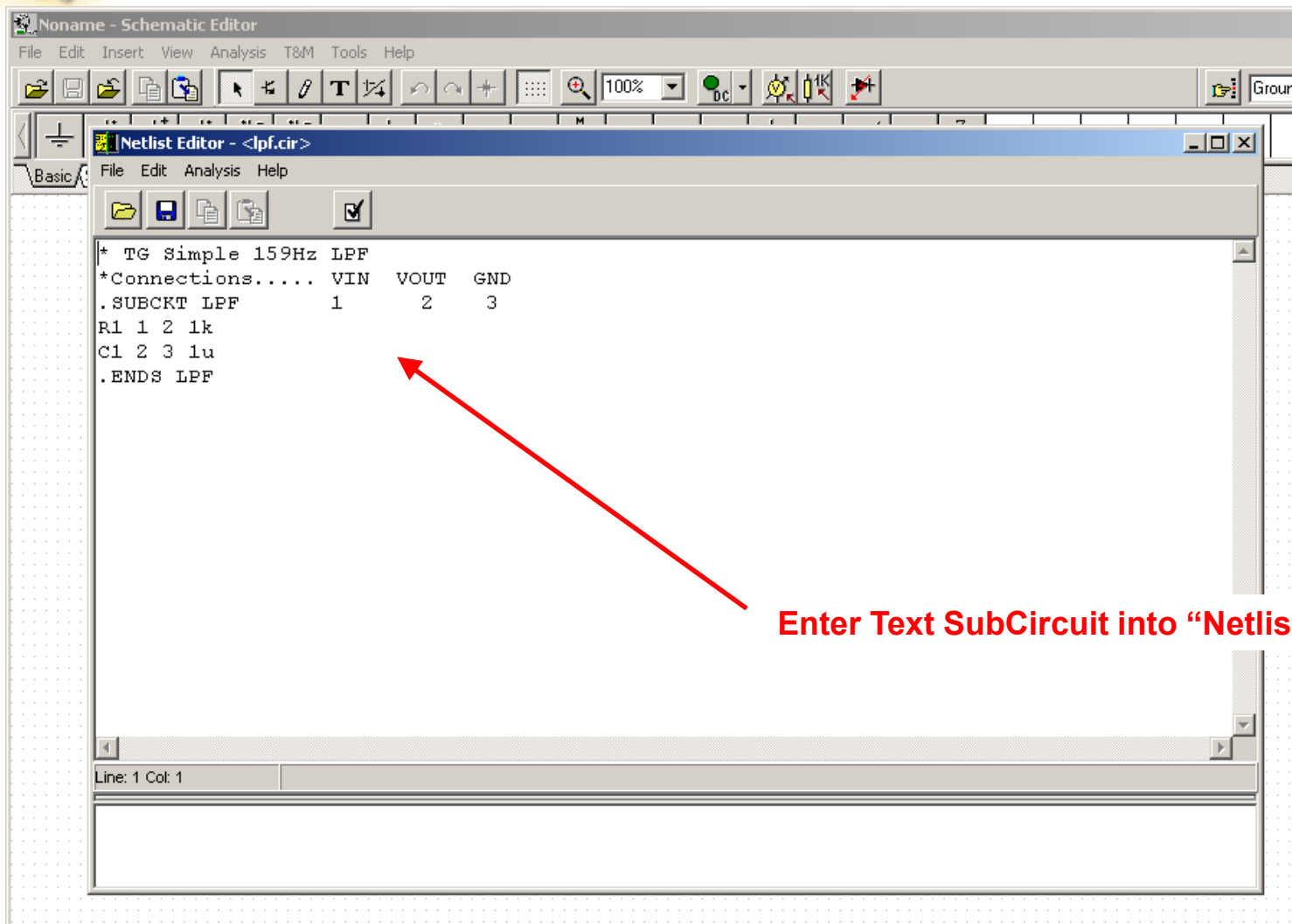
Netlist Creation



Open "Netlist Editor" in Tina Schematic Editor

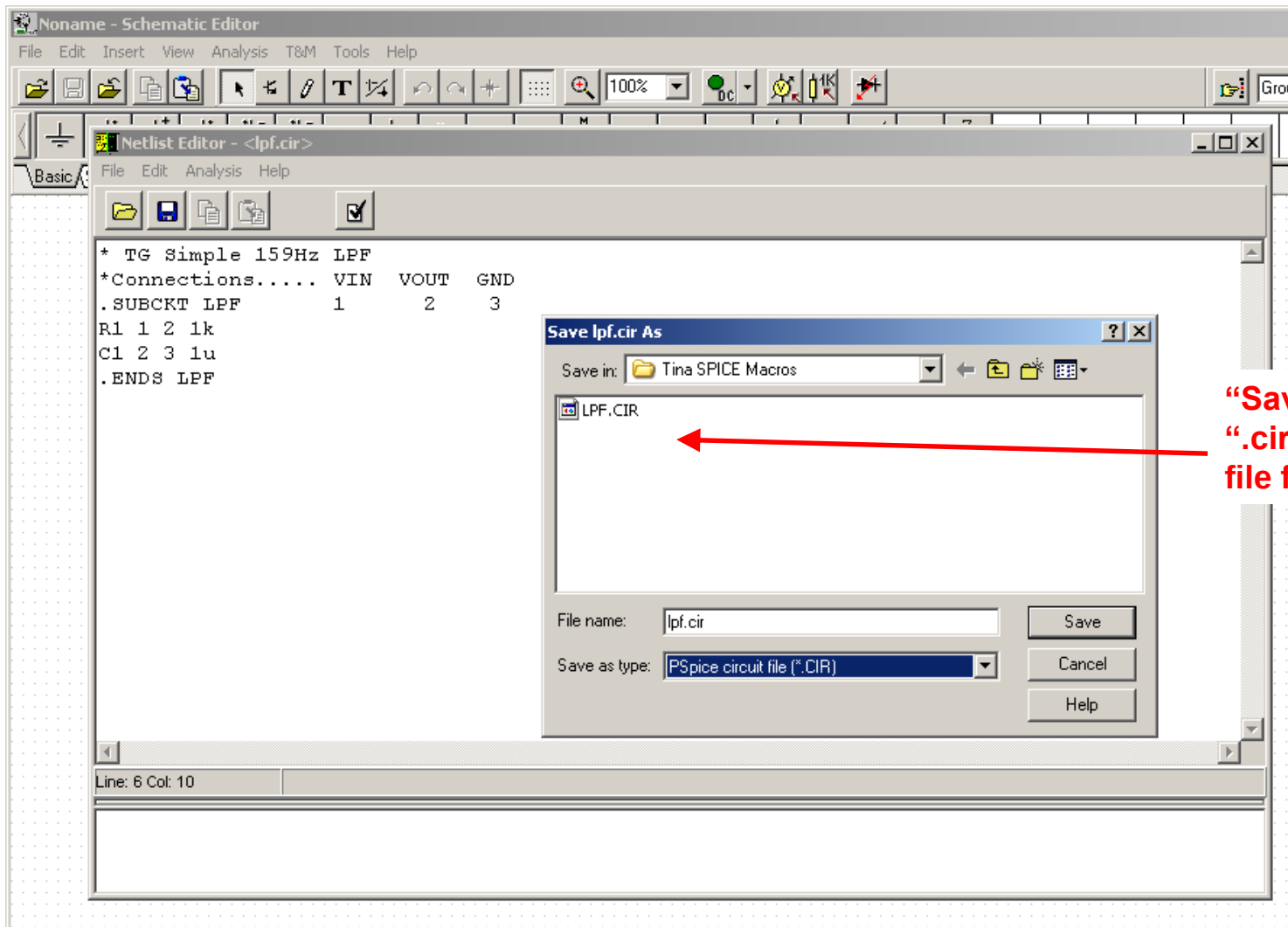
Part 1: Text Macro-Model Creation

Netlist Creation



Part 1: Text Macro-Model Creation

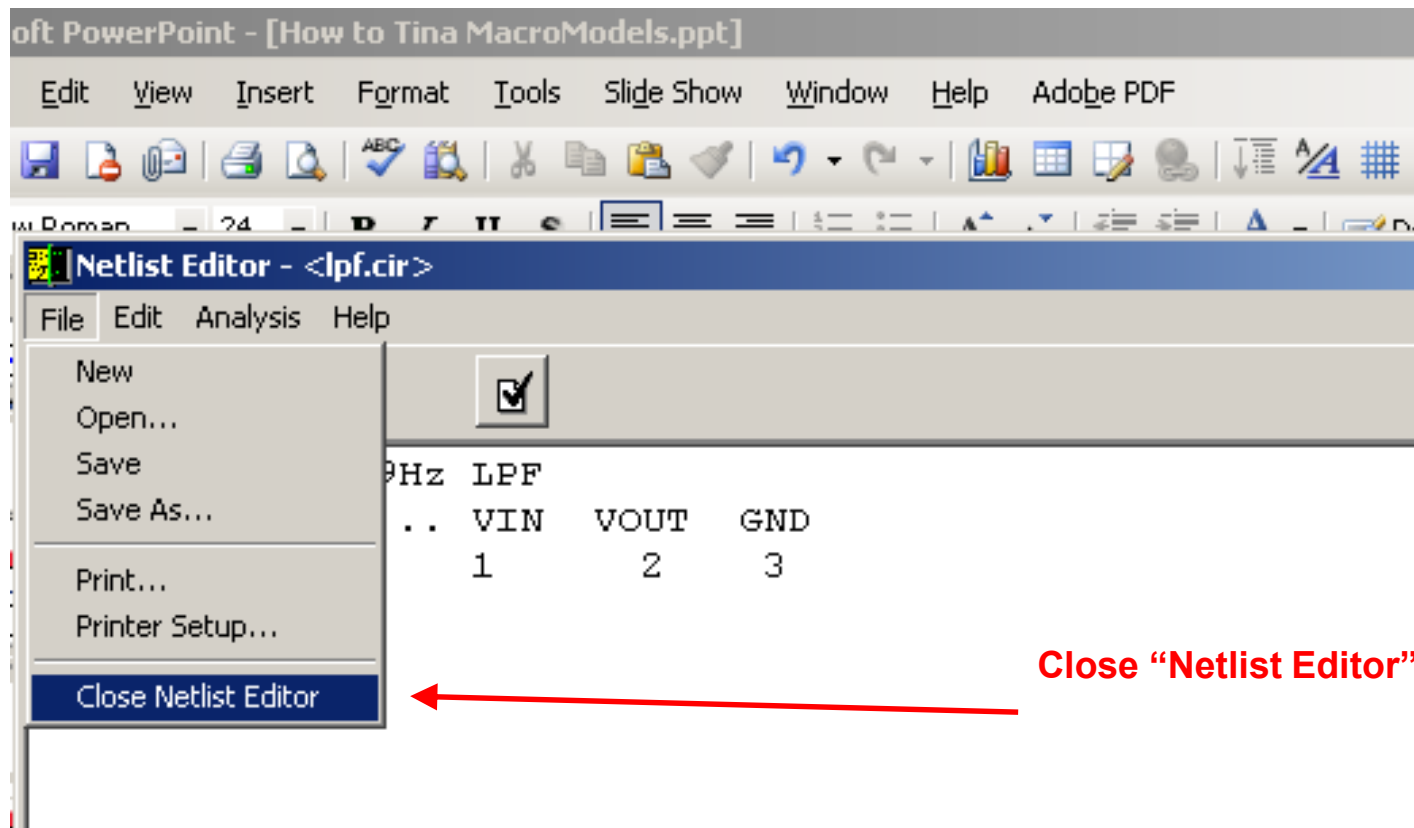
Text SubCircuit Creation



**“Save File As” either
“.cir” or “.mod” in any
file folder convenient**

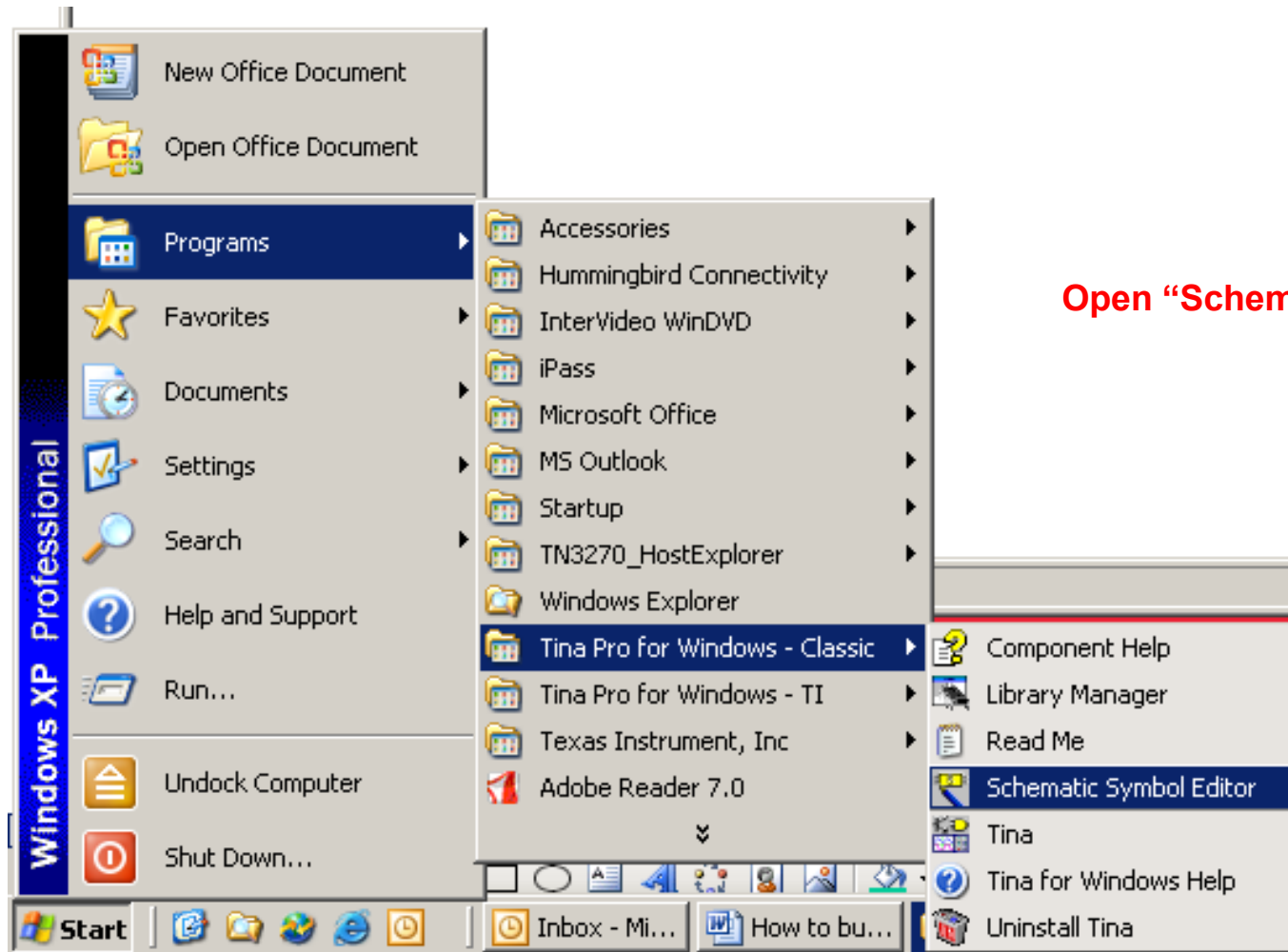
Part 1: Text Macro-Model Creation

Text SubCircuit Creation



Part 1: Text Macro-Model Creation

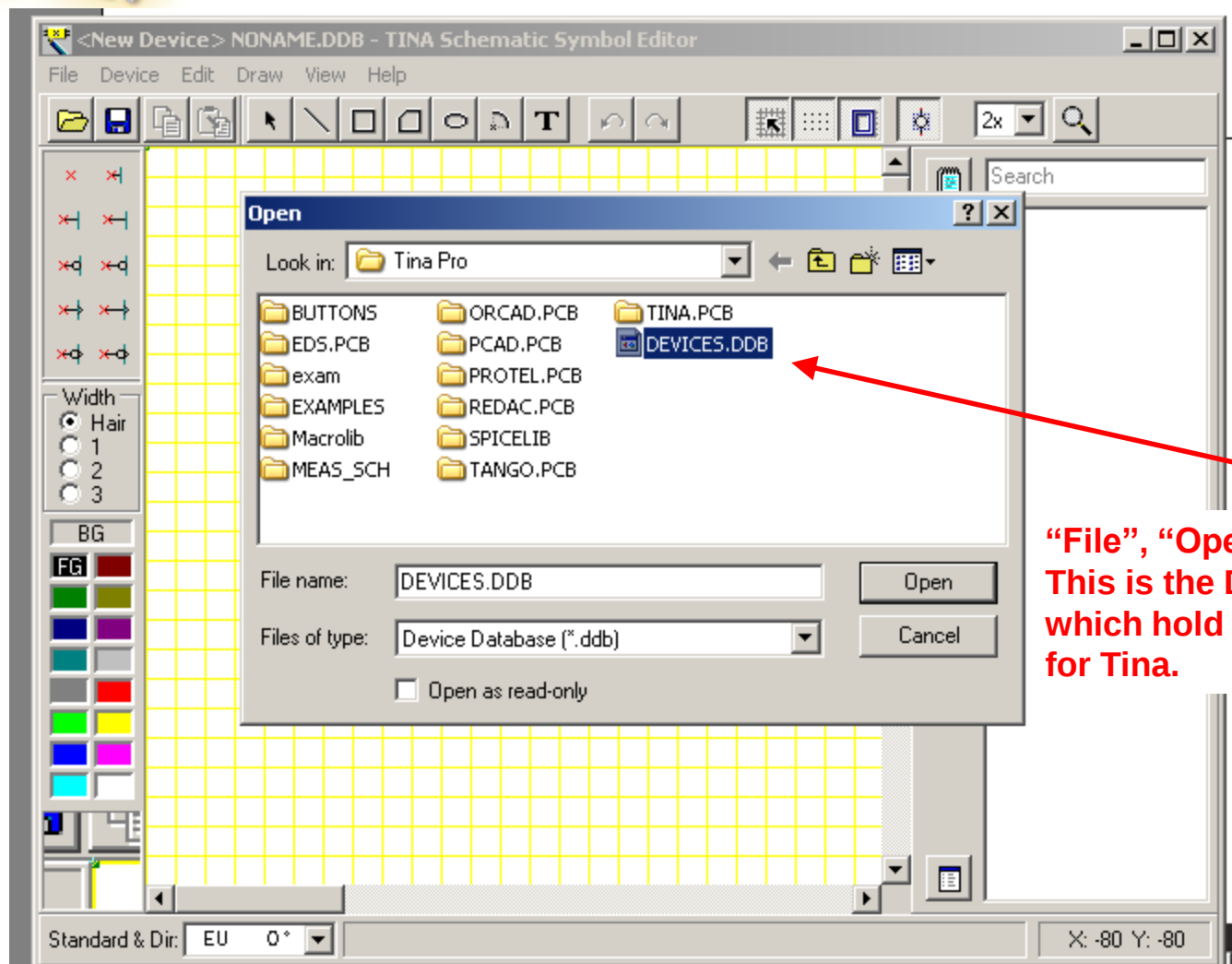
Symbol Creation



Open "Schematic Symbol Editor"

Part 1: Text Macro-Model Creation

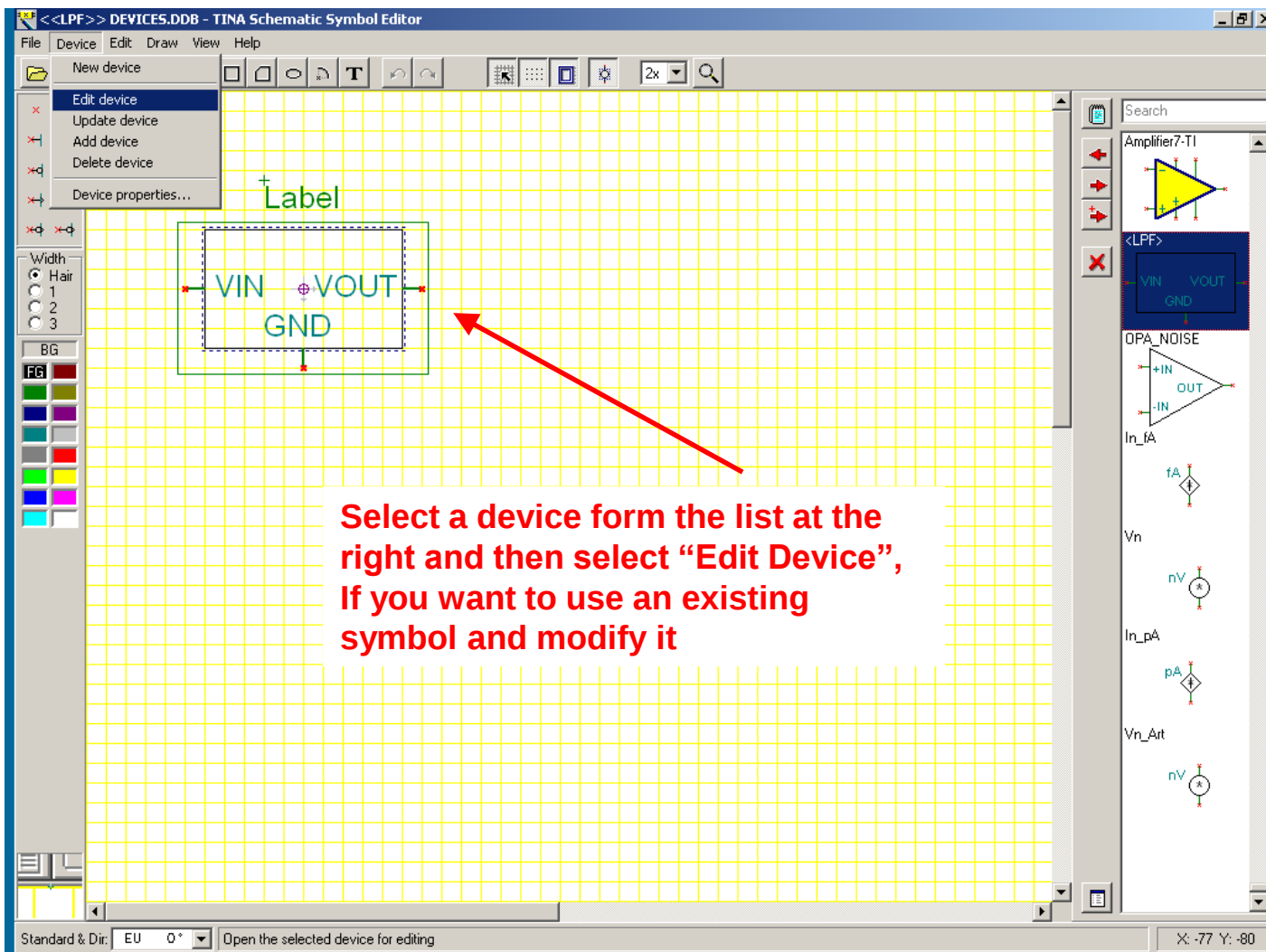
Symbol Creation



"File", "Open", "Devices.DDB"
This is the Device Database file
which hold all schematic symbols
for Tina.

Part 1: Text Macro-Model Creation

Symbol Creation - Modify



Label

VIN VOUT
GND

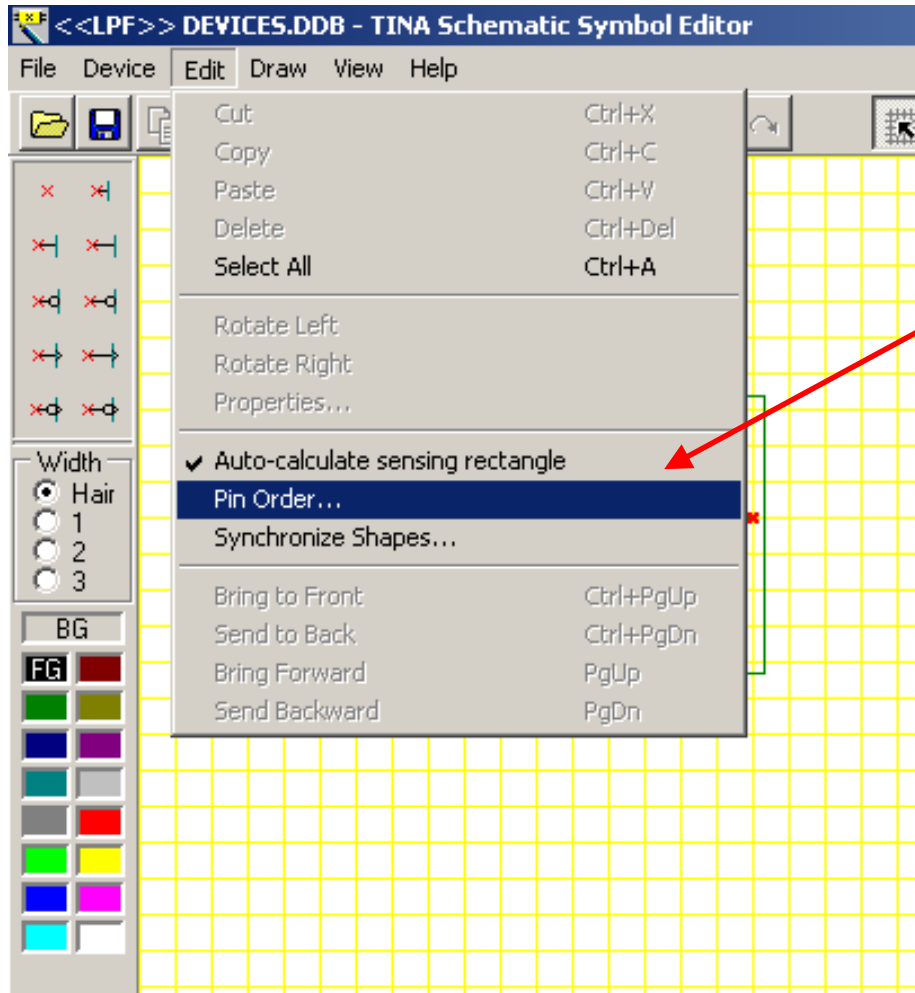
Select a device form the list at the right and then select "Edit Device",
If you want to use an existing symbol and modify it

Amplifier7-TI
<LPF>
OPA_NOISE
In_fA
Vn
In_pA
Vn_Art
nV

Standard & Dir: EU 0° Open the selected device for editing X: -77 Y: -80

Part 1: Text Macro-Model Creation

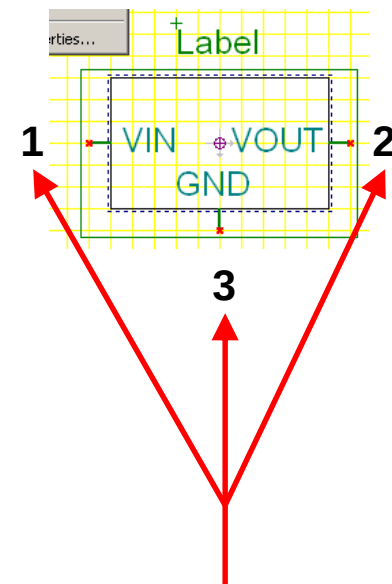
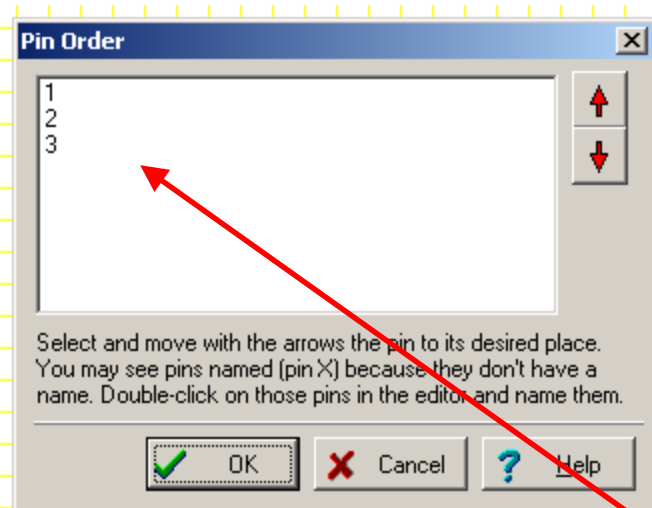
Symbol Creation - Modify



After existing symbol edits are completed, select “Edit”, “Pin Order...”

Part 1: Text Macro-Model Creation

Symbol Creation - Modify



* TG Simple 159Hz LPF

*Connections..... VIN VOUT GND

.SUBCKT LPF 1 2 3

R1 1 2 1k

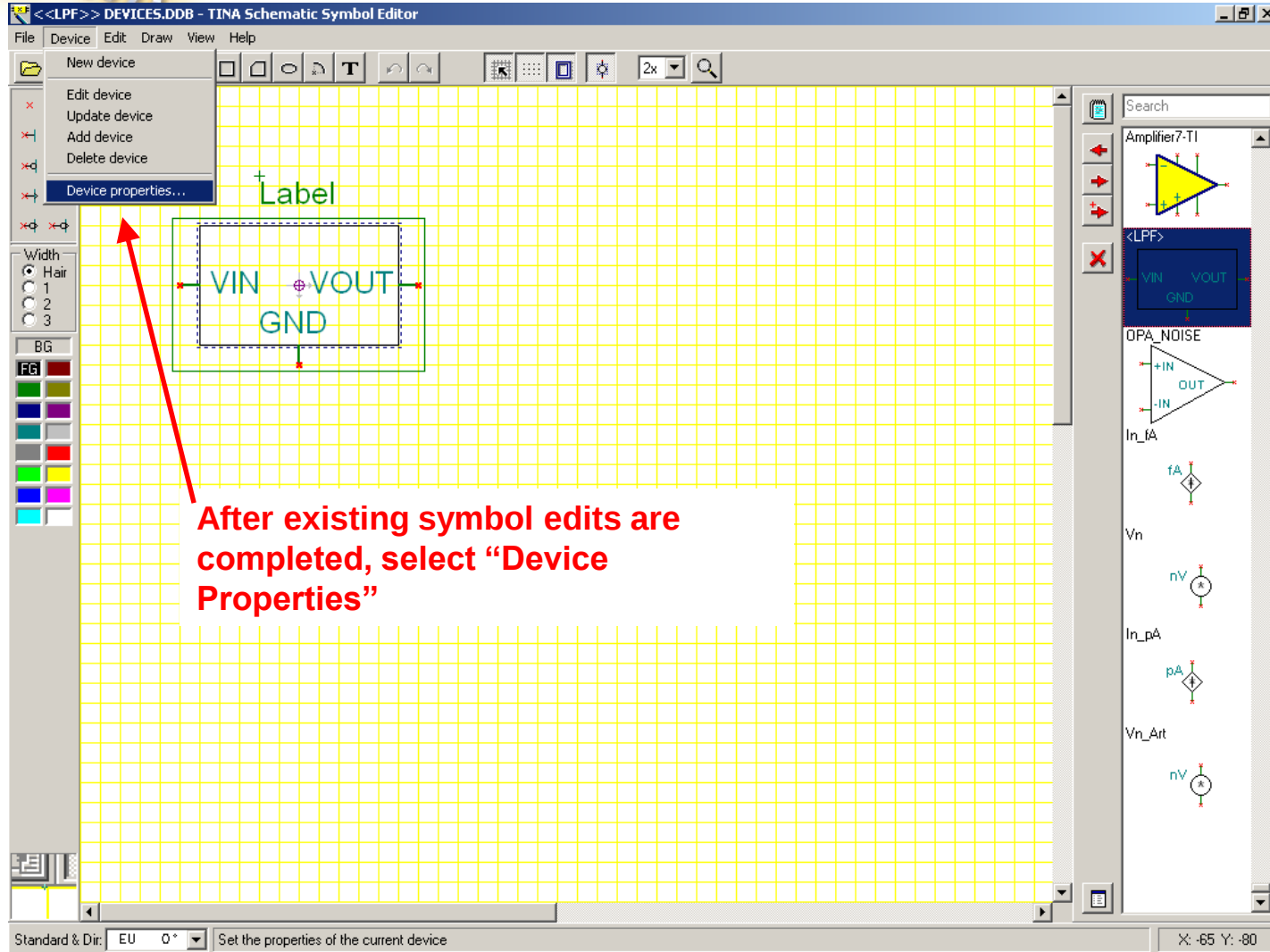
C1 2 3 1u

.ENDS LPF

Ensure "Pin Order" matches the order in the Text SubCircuit. If not re-arrange the pin order using the up and down arrows until the pin order is correct.

Part 1: Text Macro-Model Creation

Symbol Creation - Modify

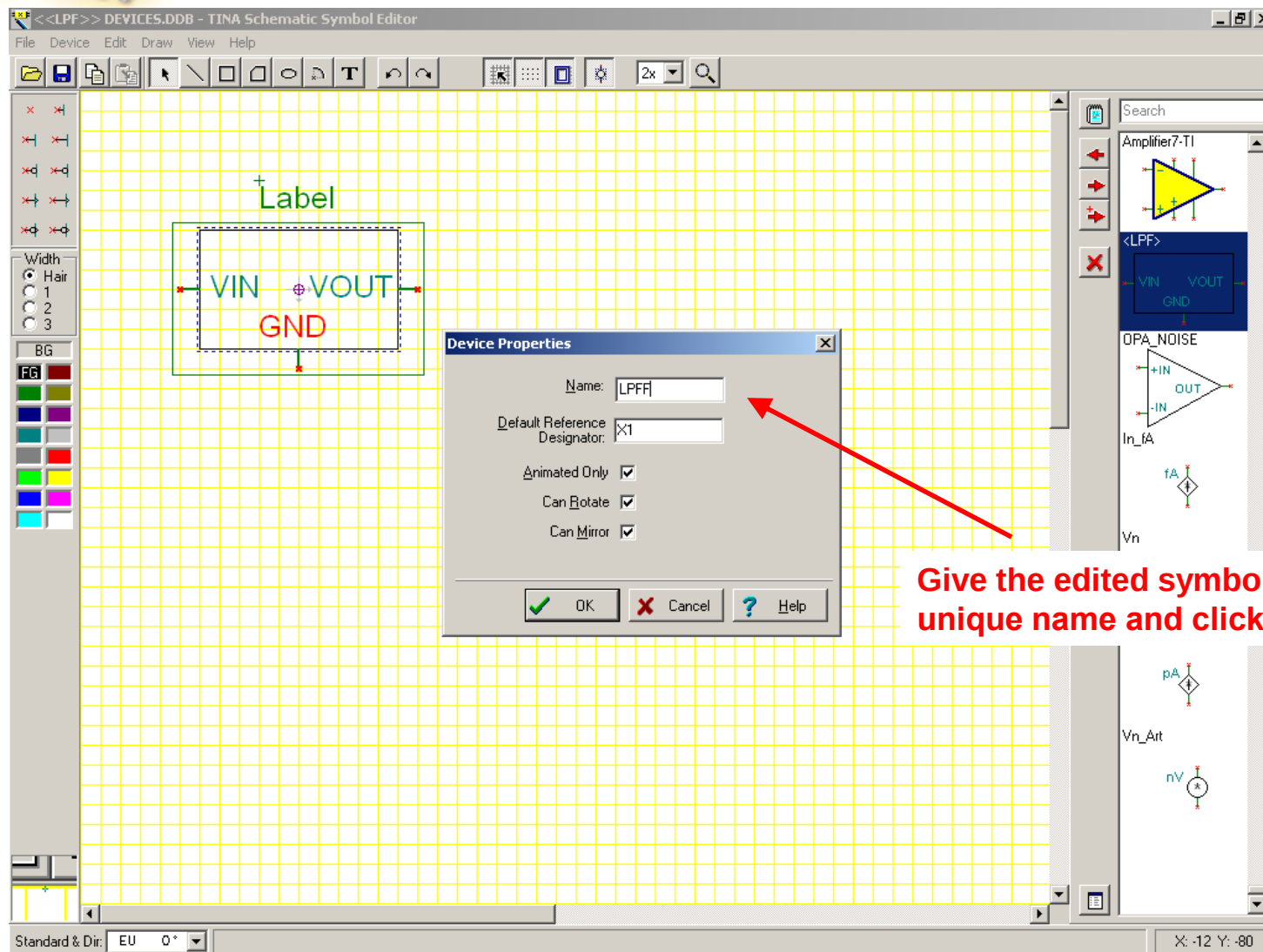


The screenshot shows the TINA Schematic Symbol Editor interface. The main workspace is a yellow grid. A device symbol is being created, with pins labeled VIN, VOUT, and GND. A green label 'Label' is positioned above the symbol. A red arrow points from a text box to the 'Device properties...' option in the 'Device' menu. The right sidebar shows a list of components, including 'Amplifier7-T1', '<LPF>', 'OPA_NOISE', 'In_fA', 'Vn', 'In_pA', 'Vn_Art', and 'nV'. The bottom status bar indicates 'Standard & Dir: EU 0°' and 'Set the properties of the current device'.

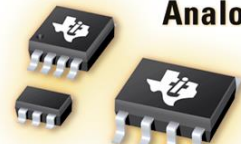
After existing symbol edits are completed, select "Device Properties"

Part 1: Text Macro-Model Creation

Symbol Creation - Modify



Give the edited symbol a new, unique name and click "OK"



Part 1: Text Macro-Model Creation

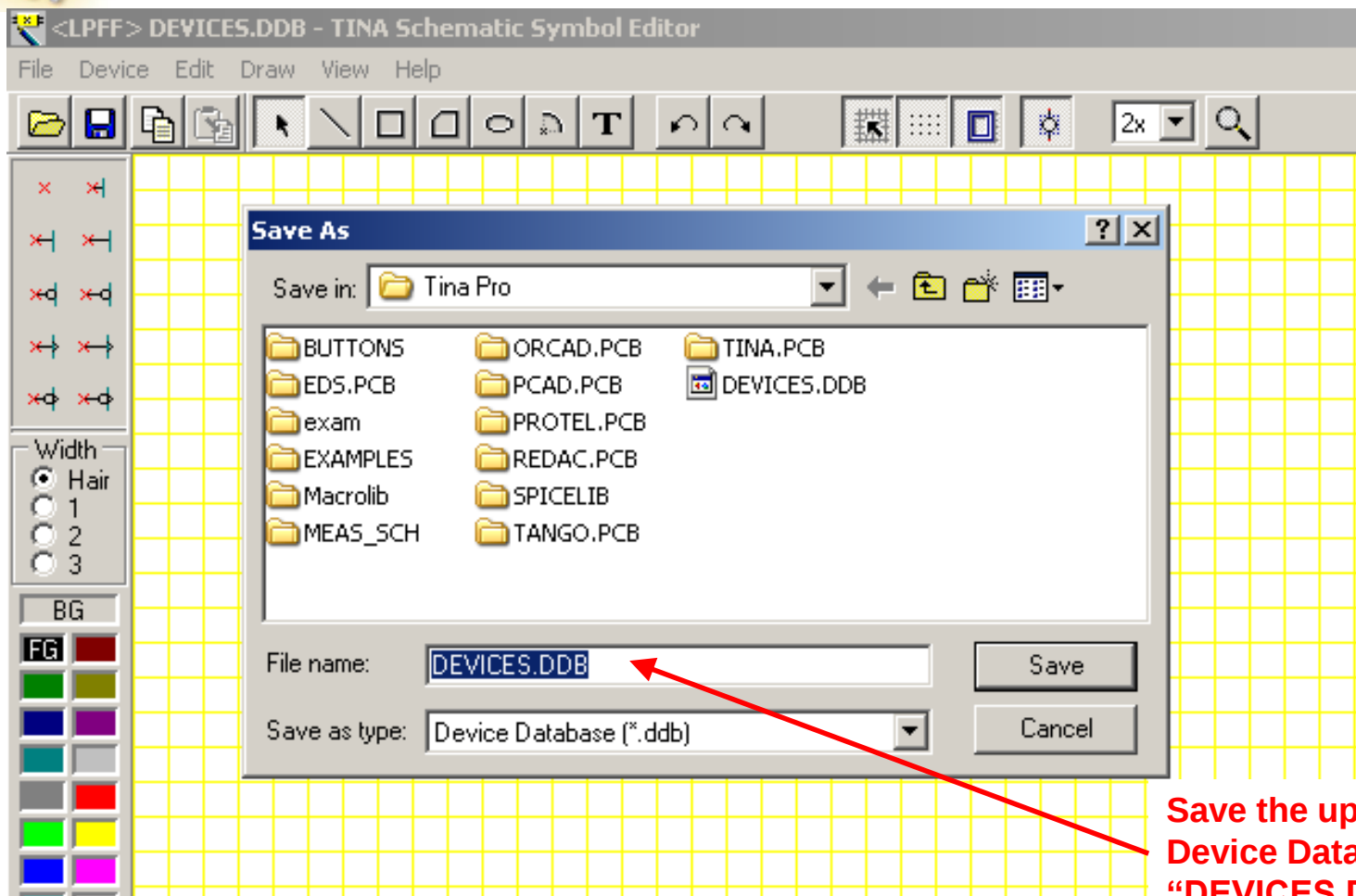
Symbol Creation - Modify

The screenshot shows the TINA Schematic Symbol Editor interface. The title bar reads "<LPFF> DEVICES.DDB - TINA Schematic Symbol Editor". The menu bar includes File, Device, Edit, Draw, View, and Help. The 'Device' menu is open, showing options: New device, Edit device, Update device, Add device (highlighted), Delete device, and Device properties... A red arrow points from the 'Add device' menu item to a new symbol being created on the grid. The symbol is a rectangle with a dashed border, containing the text 'VIN', 'VOUT', and 'GND'. A red arrow points from the symbol to the right-hand panel. The right-hand panel shows a list of devices: <LPFF>, OPA_NOISE, In_1A, fA, Vn, nV, In_pA, pA, Vn_Art, nV, and LPFF. The 'LPFF' device is selected, and its symbol is shown at the bottom of the panel. The bottom status bar shows 'Standard & Dir: EU 0°' and 'Add the current device to the device database as a new entry'.

**“Add Device”,
Add the new
symbol to the
list at the right.
Use +→ button
or drop down
menu.**

Part 1: Text Macro-Model Creation

Symbol Creation - Modify

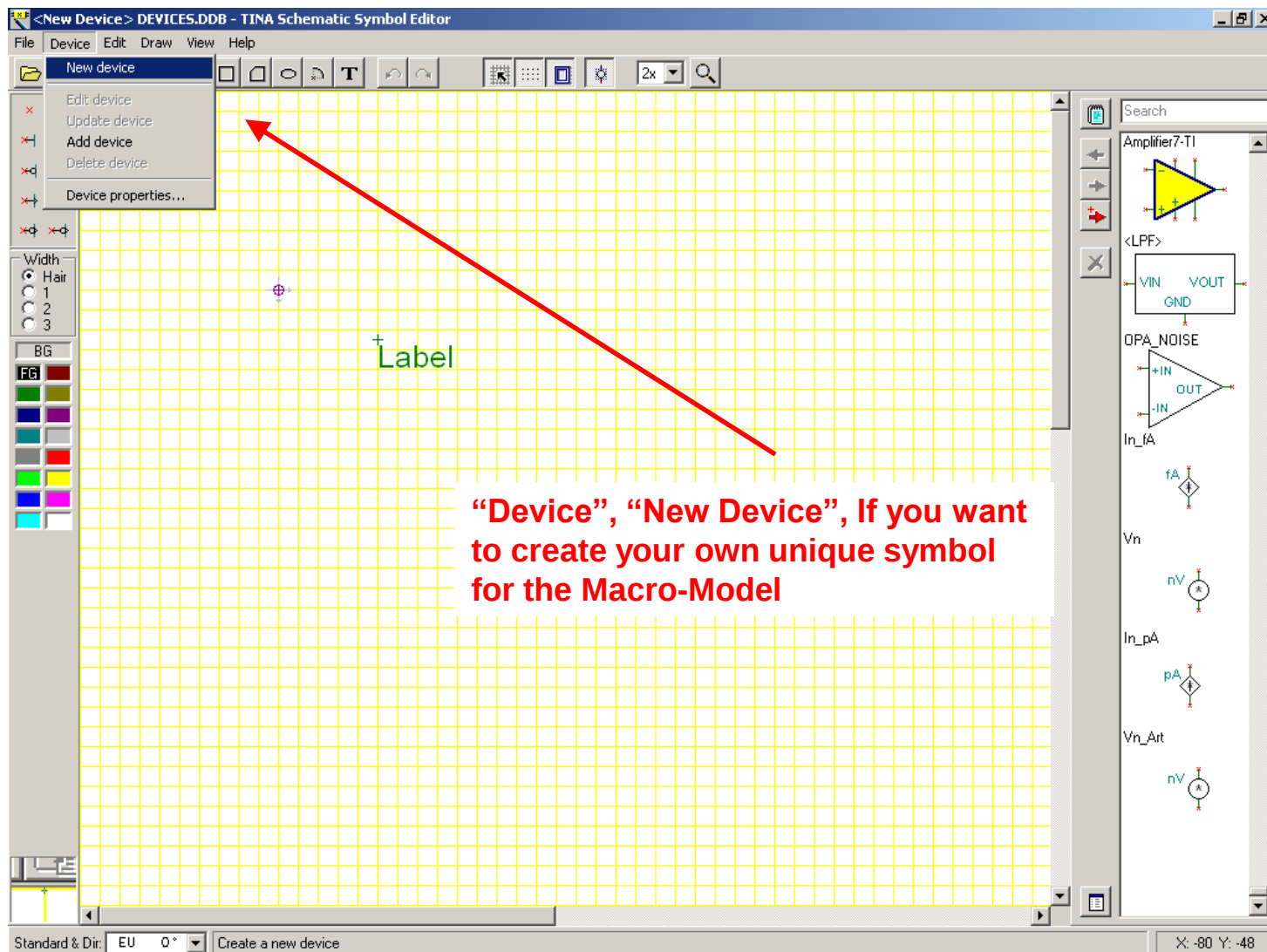


Save the updated
Device Database as
"DEVICES.DDB"

Warning: Backup the original "DEVICES.DDB" File in case there is a problem.

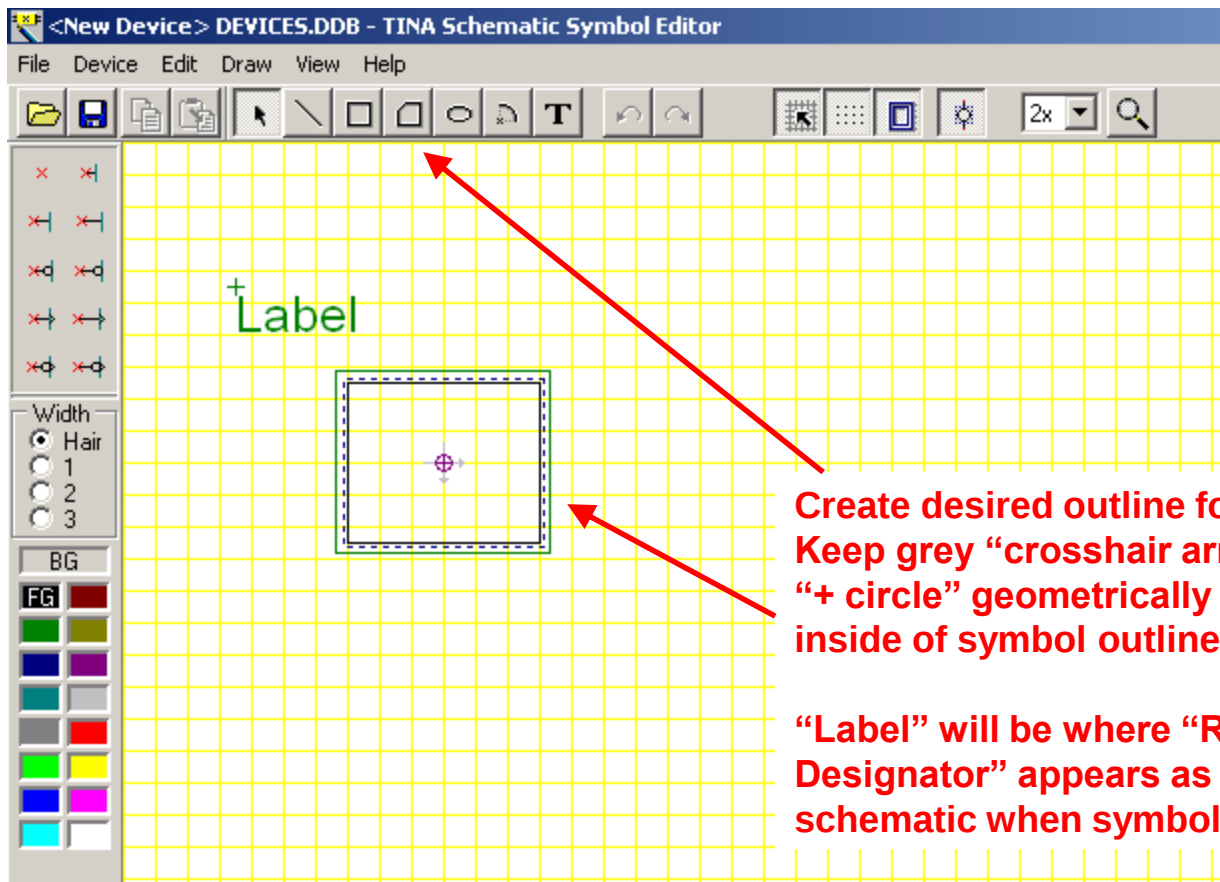
Part 1: Text Macro-Model Creation

Symbol Creation - New



Part 1: Text Macro-Model Creation

Symbol Creation - New

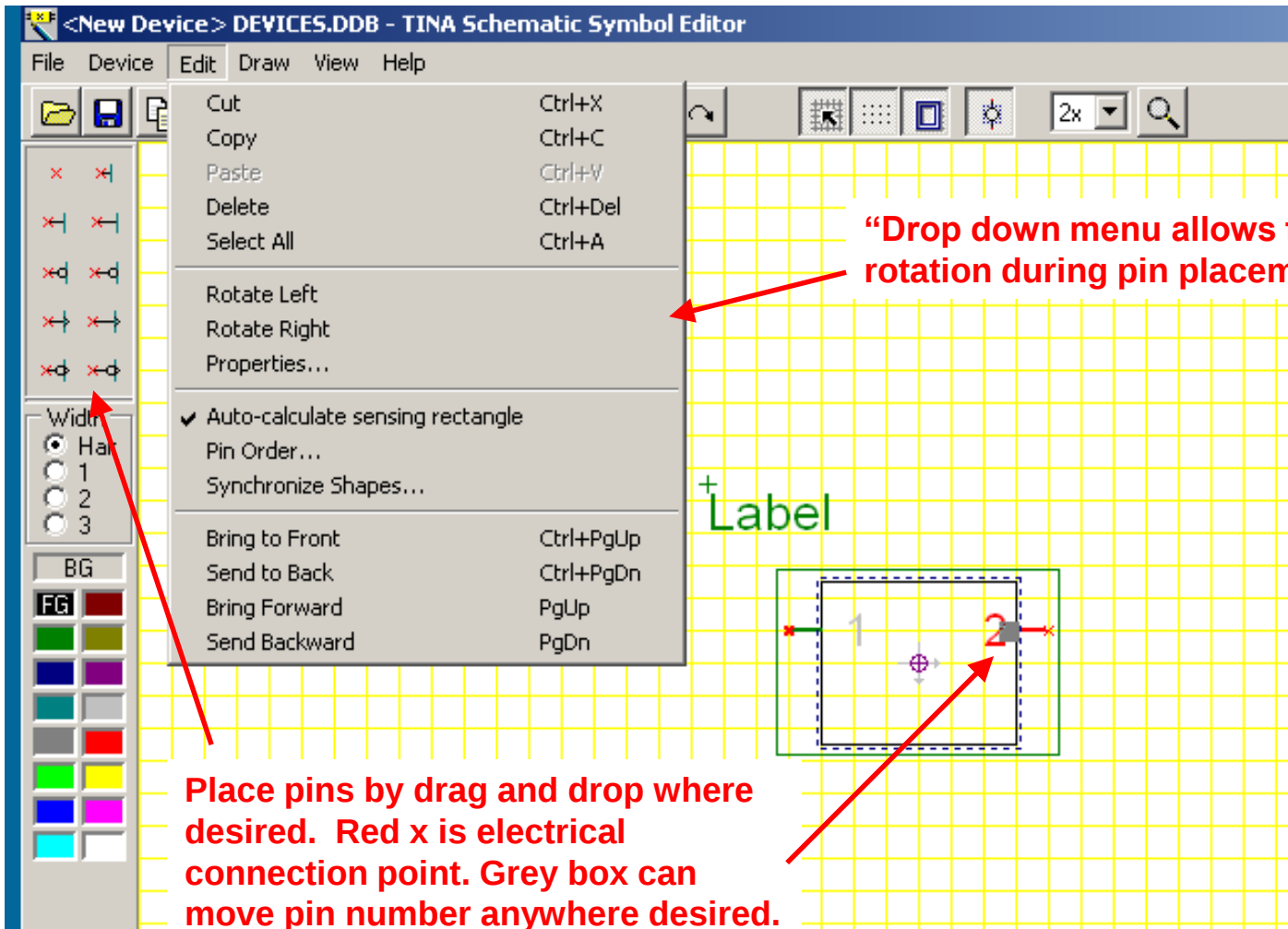


Create desired outline for symbol. Keep grey “crosshair arrows” and “+ circle” geometrically centered inside of symbol outline.

“Label” will be where “Ref Designator” appears as default on schematic when symbol is placed.

Part 1: Text Macro-Model Creation

Symbol Creation - New



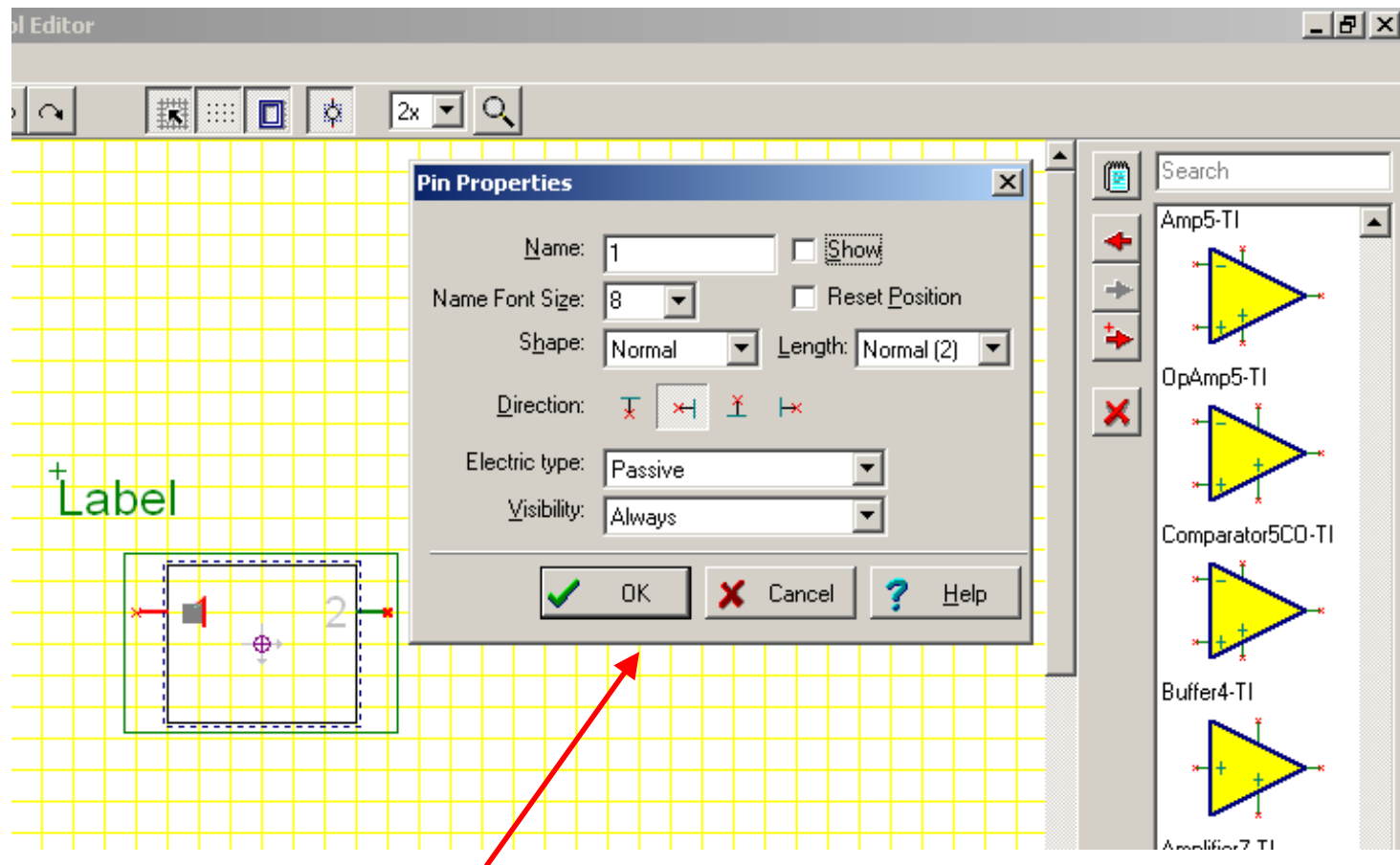
The screenshot shows the 'TINA Schematic Symbol Editor' window. The 'Edit' menu is open, displaying various editing options. A red arrow points from the 'Pin' icon in the left toolbar to the 'Edit' menu. Another red arrow points from the 'Drop down menu allows for pin rotation during pin placement' text to the 'Rotate Left' and 'Rotate Right' options in the 'Edit' menu. A third red arrow points from the 'Place pins by drag and drop where desired. Red x is electrical connection point. Grey box can move pin number anywhere desired.' text to the schematic diagram. The schematic diagram shows a green label 'Label' with a '+' sign, and a dashed box containing two pins. Pin 1 is on the left and Pin 2 is on the right. Pin 2 has a red 'x' at its electrical connection point and a grey box around the number '2'.

“Drop down menu allows for pin rotation during pin placement”

Place pins by drag and drop where desired. Red x is electrical connection point. Grey box can move pin number anywhere desired.

Part 1: Text Macro-Model Creation

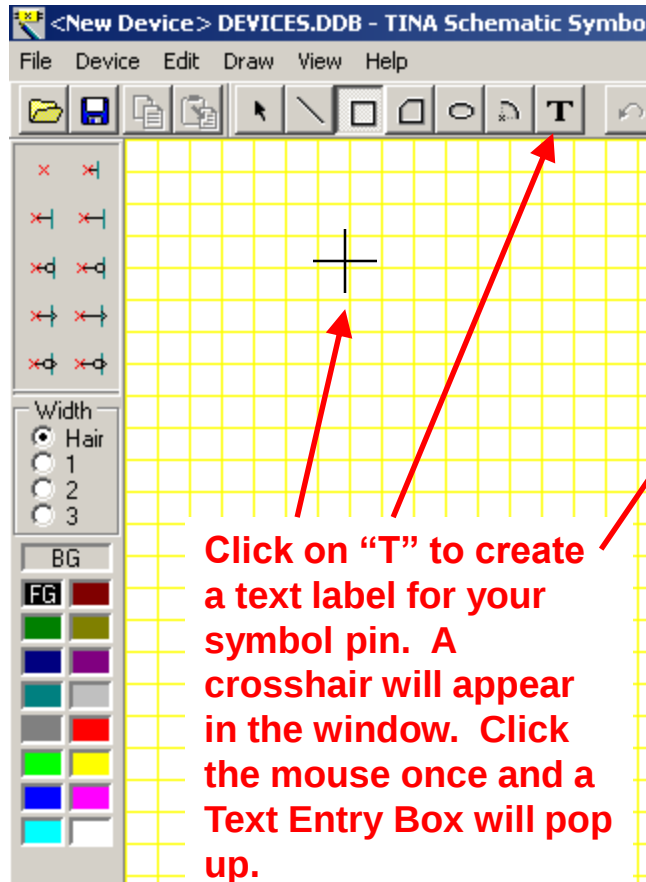
Symbol Creation - New



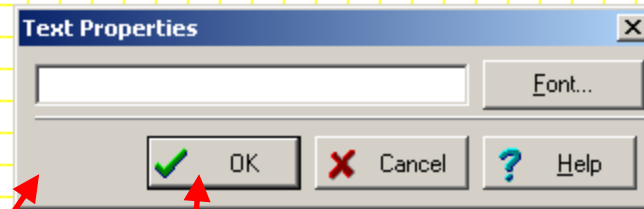
Double click on each pin to select the “Pin Properties”. If you want a name on your pin unselect “Show”. For your Symbol to be available to build your Macro-Model form the Text SubCircuit it must be built with the EXACT same number of pins and use the same pin numbers on the pins as your Text SubCircuit

Part 1: Text Macro-Model Creation

Symbol Creation - New

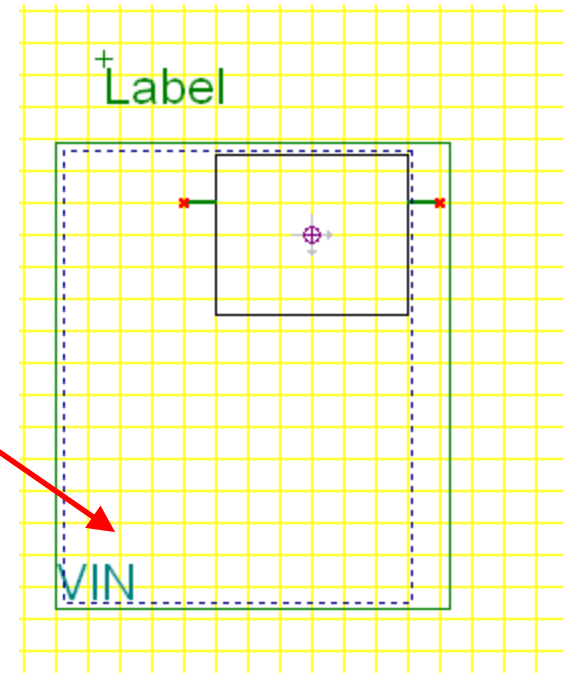


Click on "T" to create a text label for your symbol pin. A crosshair will appear in the window. Click the mouse once and a Text Entry Box will pop up.



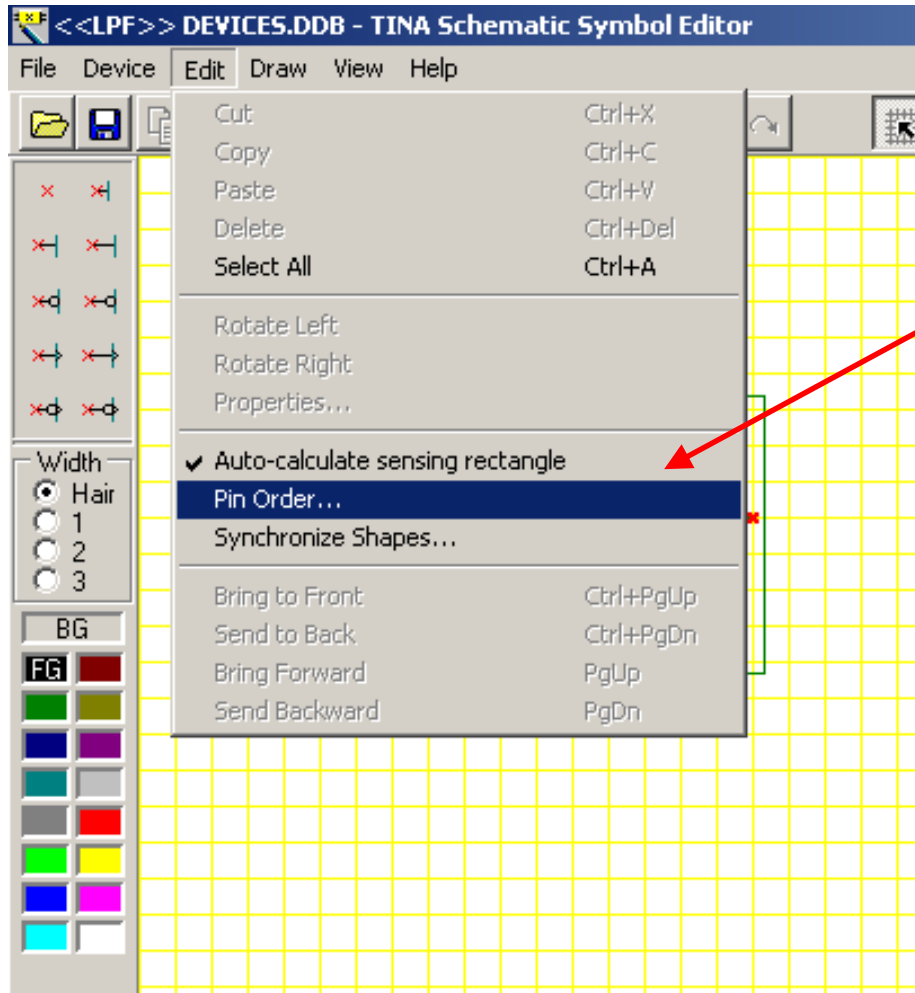
Enter desired pin name here

After text entry click "OK" and name will appear. Move pin name where desired.



Part 1: Text Macro-Model Creation

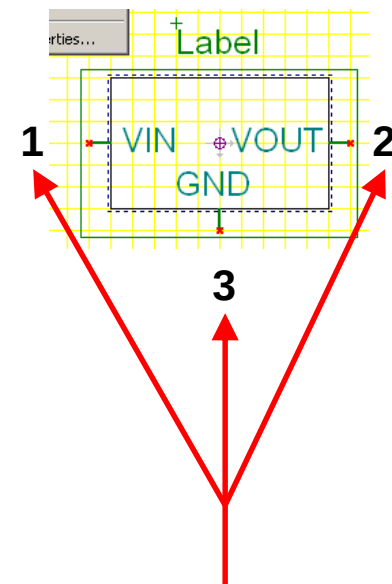
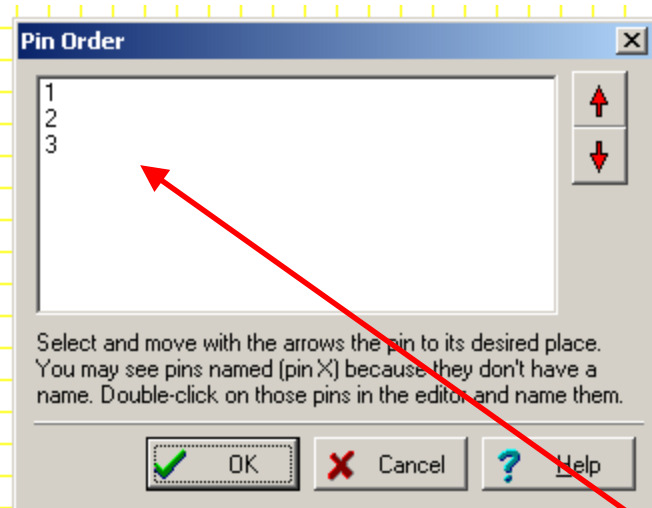
Symbol Creation - New



After existing symbol edits are completed, select “Edit”, “Pin Order...”

Part 1: Text Macro-Model Creation

Symbol Creation - New



Hidden Pin Numbers

* TG Simple 159Hz LPF

*Connections..... VIN VOUT GND

.SUBCKT LPF 1 2 3

R1 1 2 1k

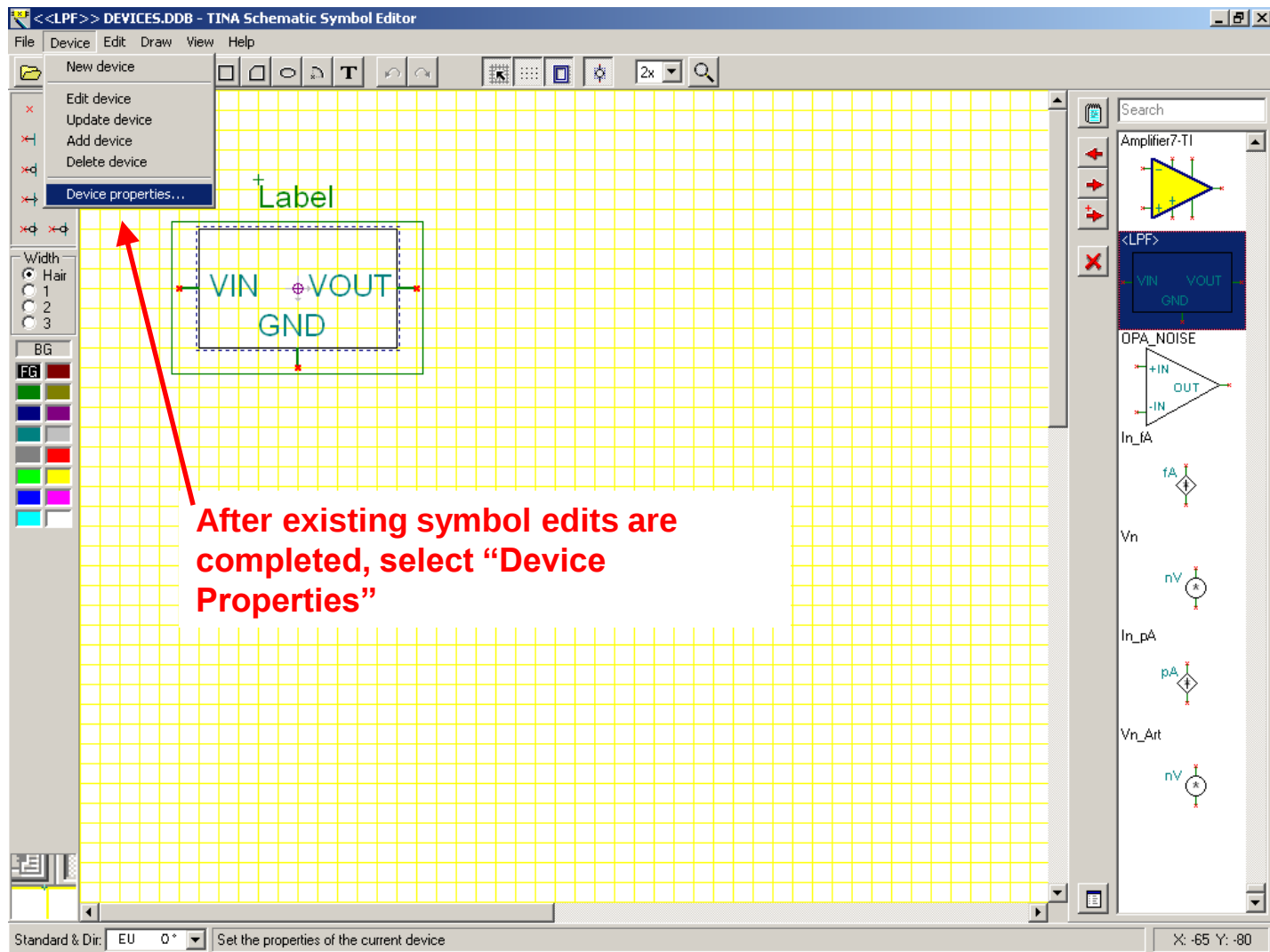
C1 2 3 1u

.ENDS LPF

Ensure "Pin Order" matches the order in the Text SubCircuit. If not re-arrange the pin order using the up and down arrows until the pin order is correct.

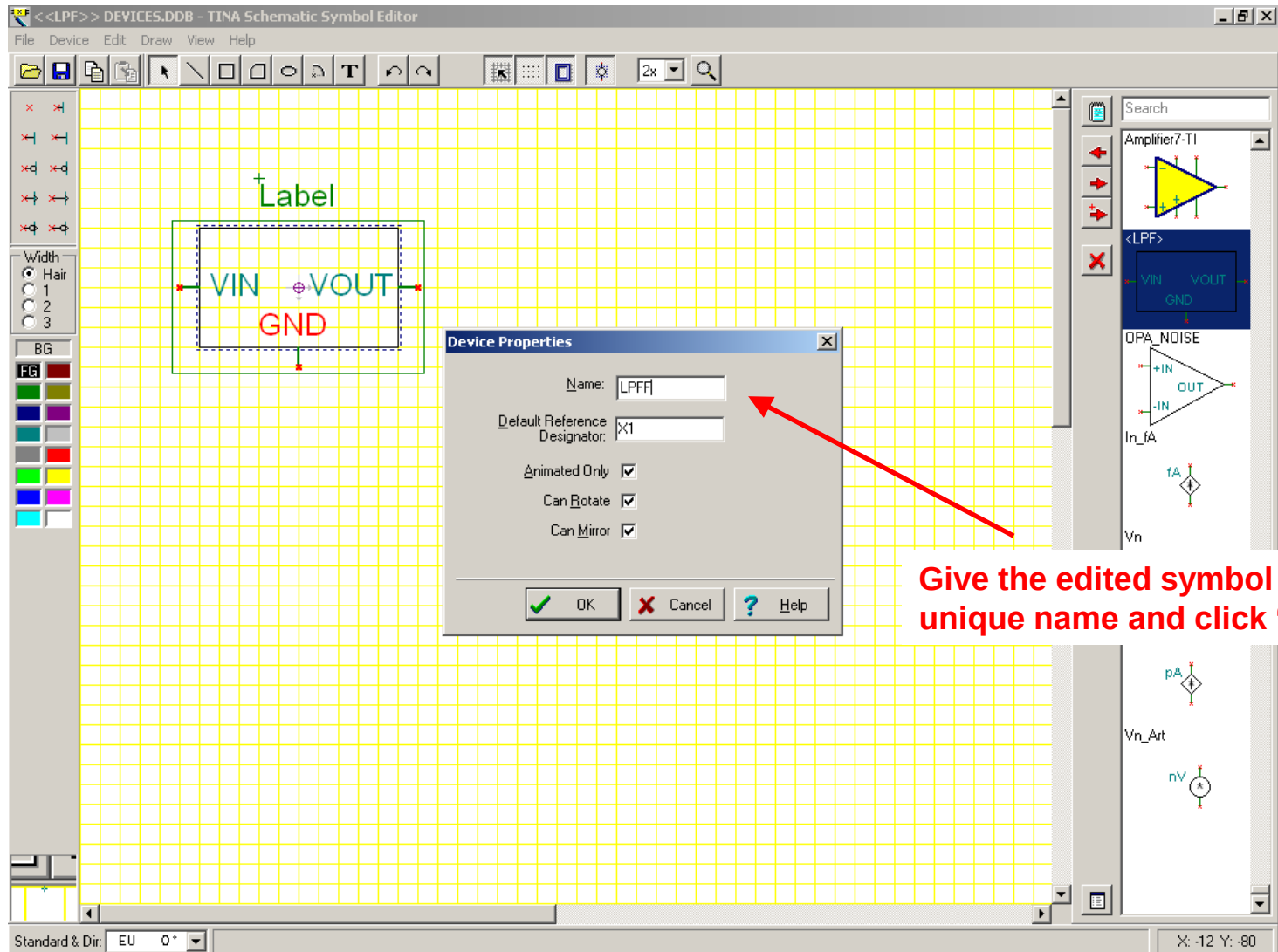
Part 1: Text Macro-Model Creation

Symbol Creation - New



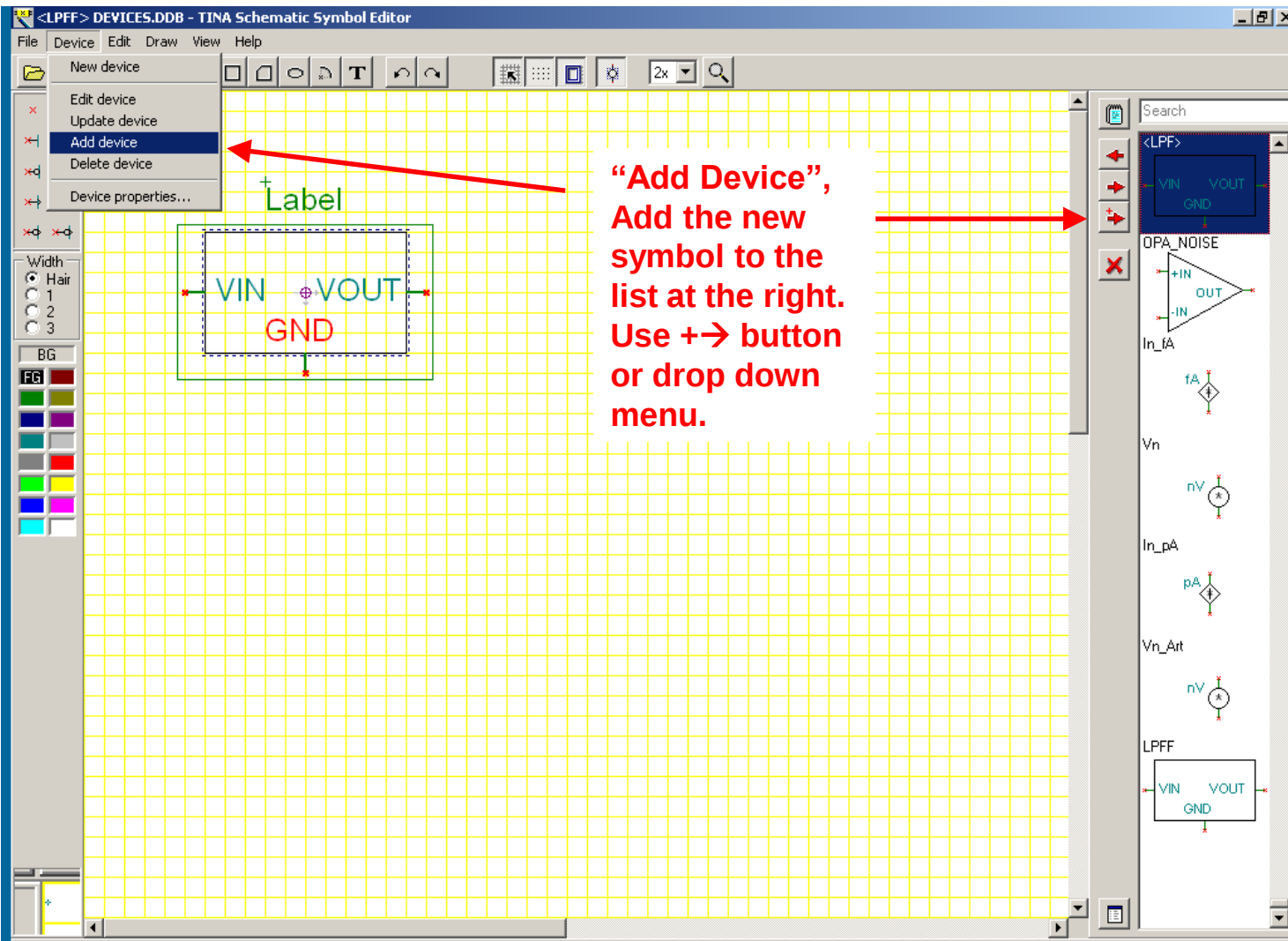
Part 1: Text Macro-Model Creation

Symbol Creation - New



Part 1: Text Macro-Model Creation

Symbol Creation - New

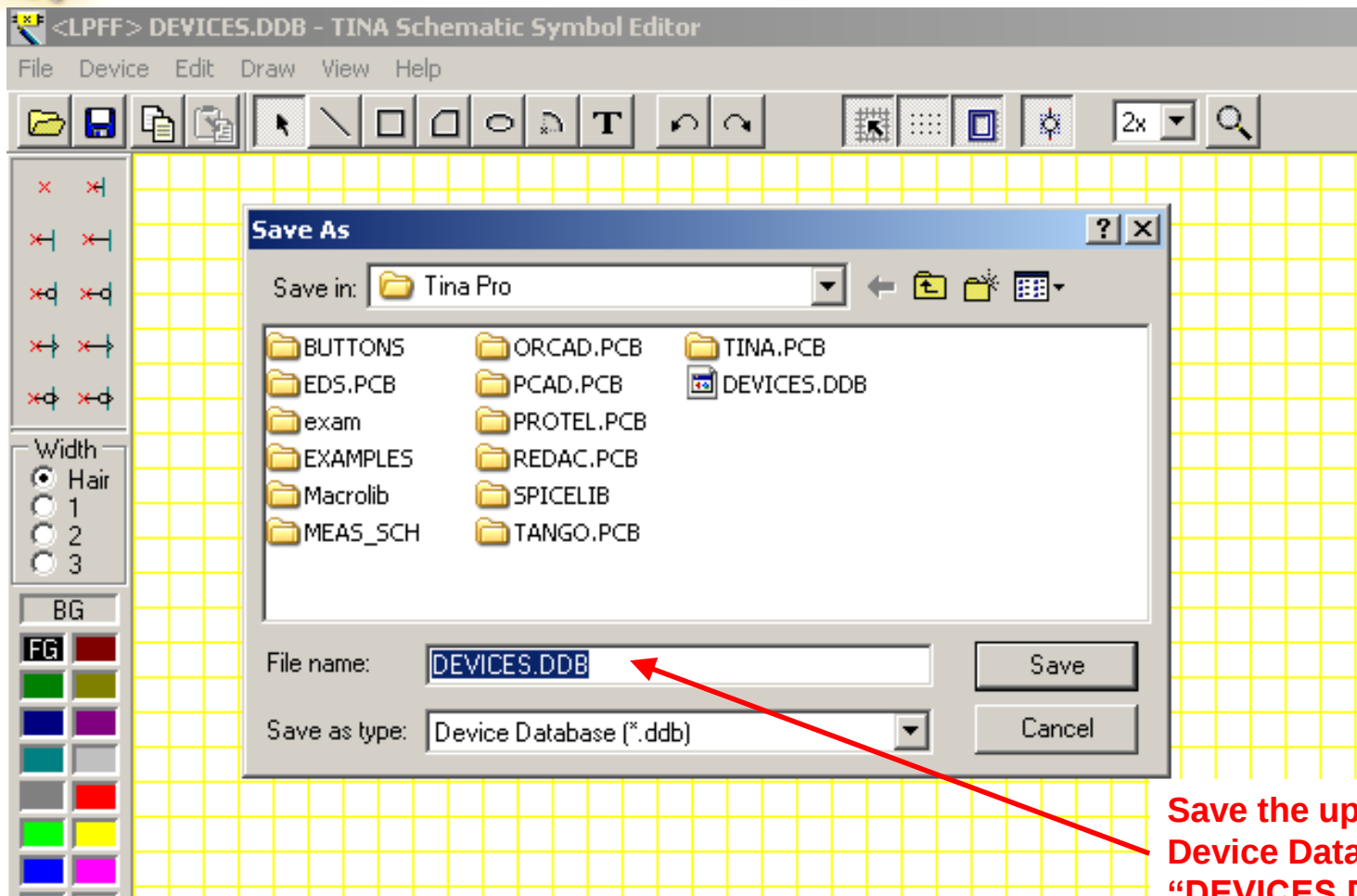


The screenshot shows the TINA Schematic Symbol Editor interface. The main workspace is a yellow grid. On the left, a menu is open with 'Add device' selected. A red arrow points from this menu item to a text box in the center. The text box contains a schematic symbol for a device with inputs 'VIN' and 'VOUT', and a ground connection 'GND'. A red arrow points from the text box to the right-hand panel, which shows a list of devices including 'OPA_NOISE', 'In_fA', 'Vn', 'In_pA', 'Vn_Art', and 'LPFF'. The 'LPFF' device is highlighted, and its symbol is shown below it. The bottom status bar indicates 'Standard & Dir: EU 0°' and 'Add the current device to the device database as a new entry'.

**“Add Device”,
Add the new
symbol to the
list at the right.
Use +→ button
or drop down
menu.**

Part 1: Text Macro-Model Creation

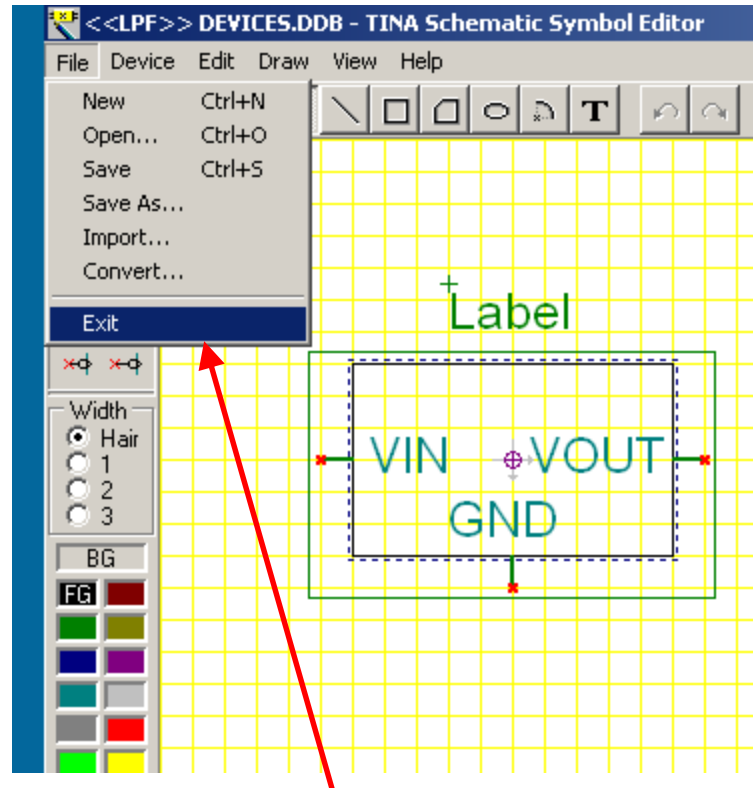
Symbol Creation - New



Warning: Backup the original "DEVICES.DDB" File in case there is a problem.

Part 1: Text Macro-Model Creation

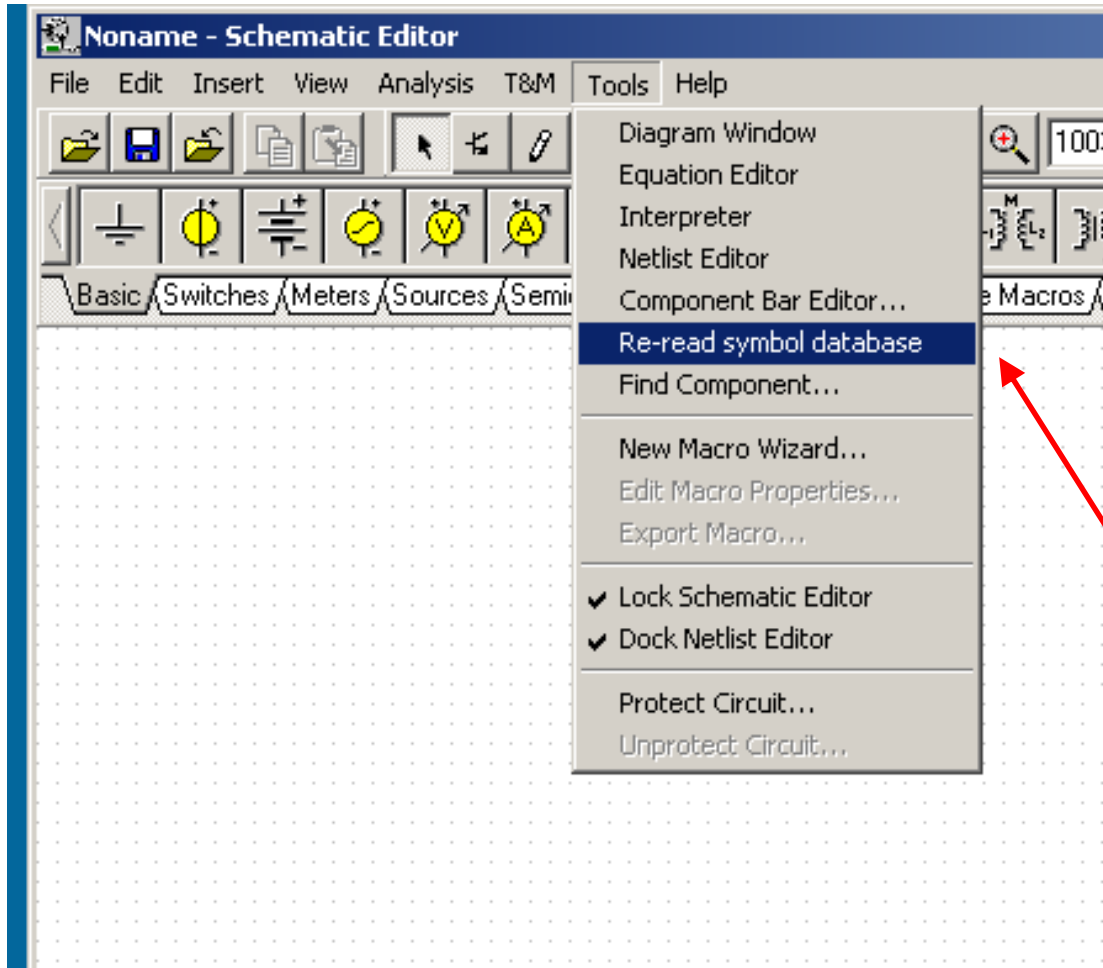
Macro-Model Creation



Either “Exit” or minimize the “Schematic Symbol Editor”

Part 1: Text Macro-Model Creation

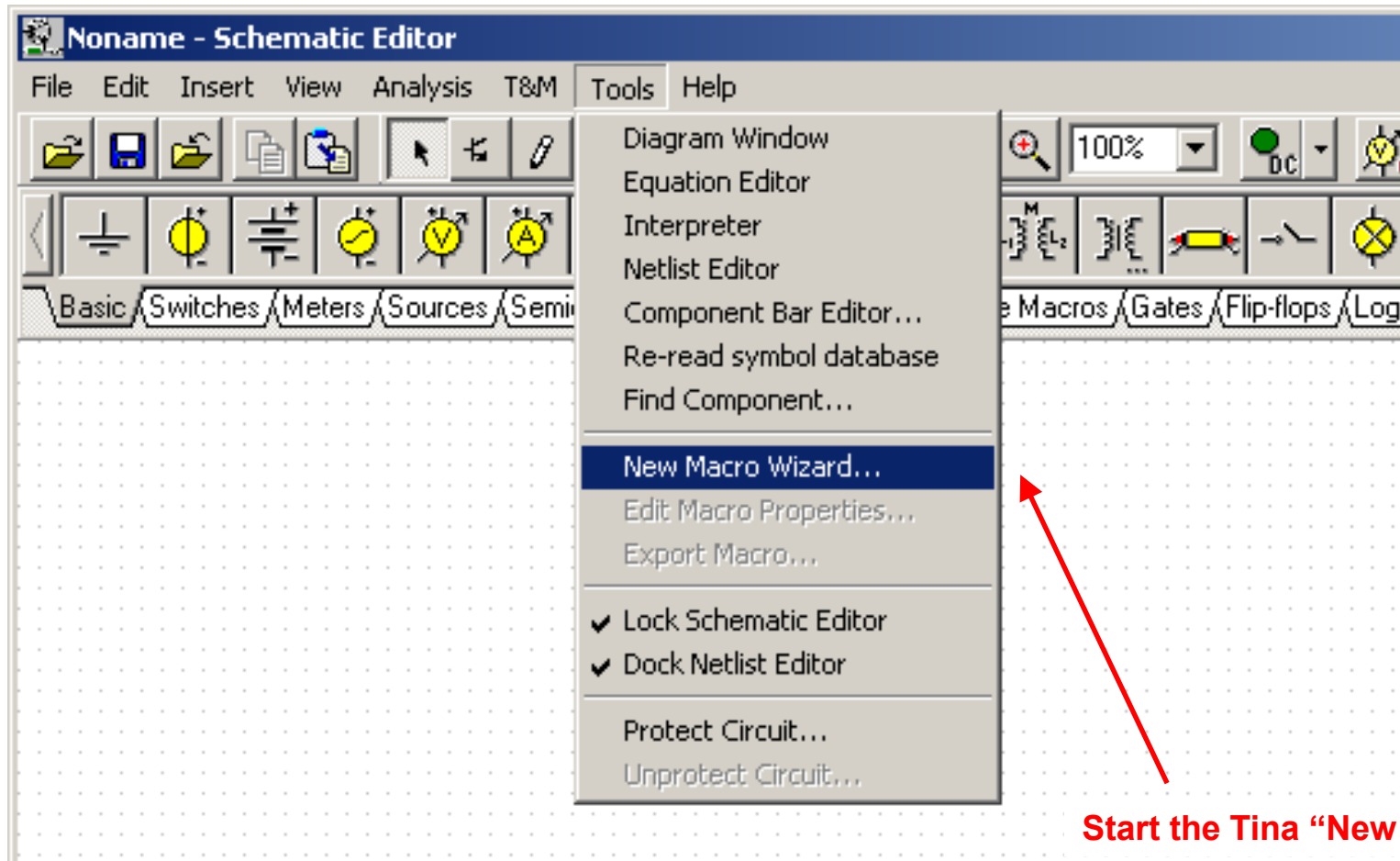
Macro-Model Creation



Select "Re-read symbol database"

Part 1: Text Macro-Model Creation

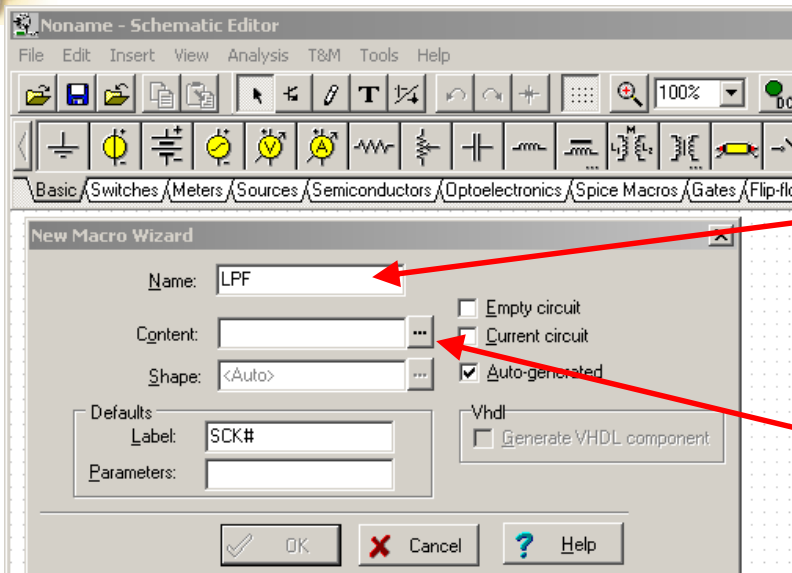
Macro-Model Creation



Start the Tina "New Macro Wizard"

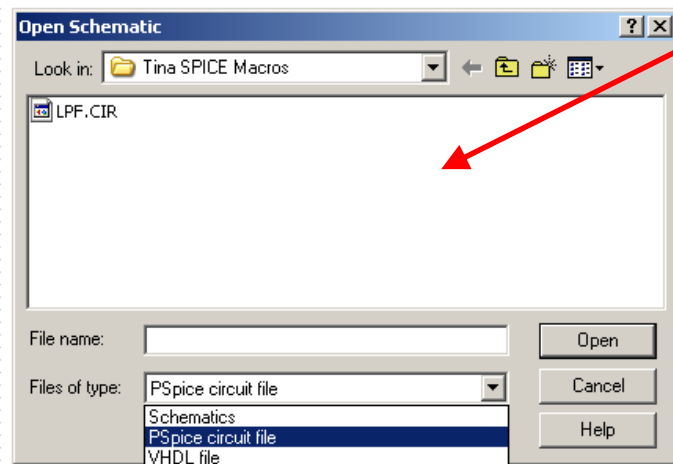
Part 1: Text Macro-Model Creation

Macro-Model Creation



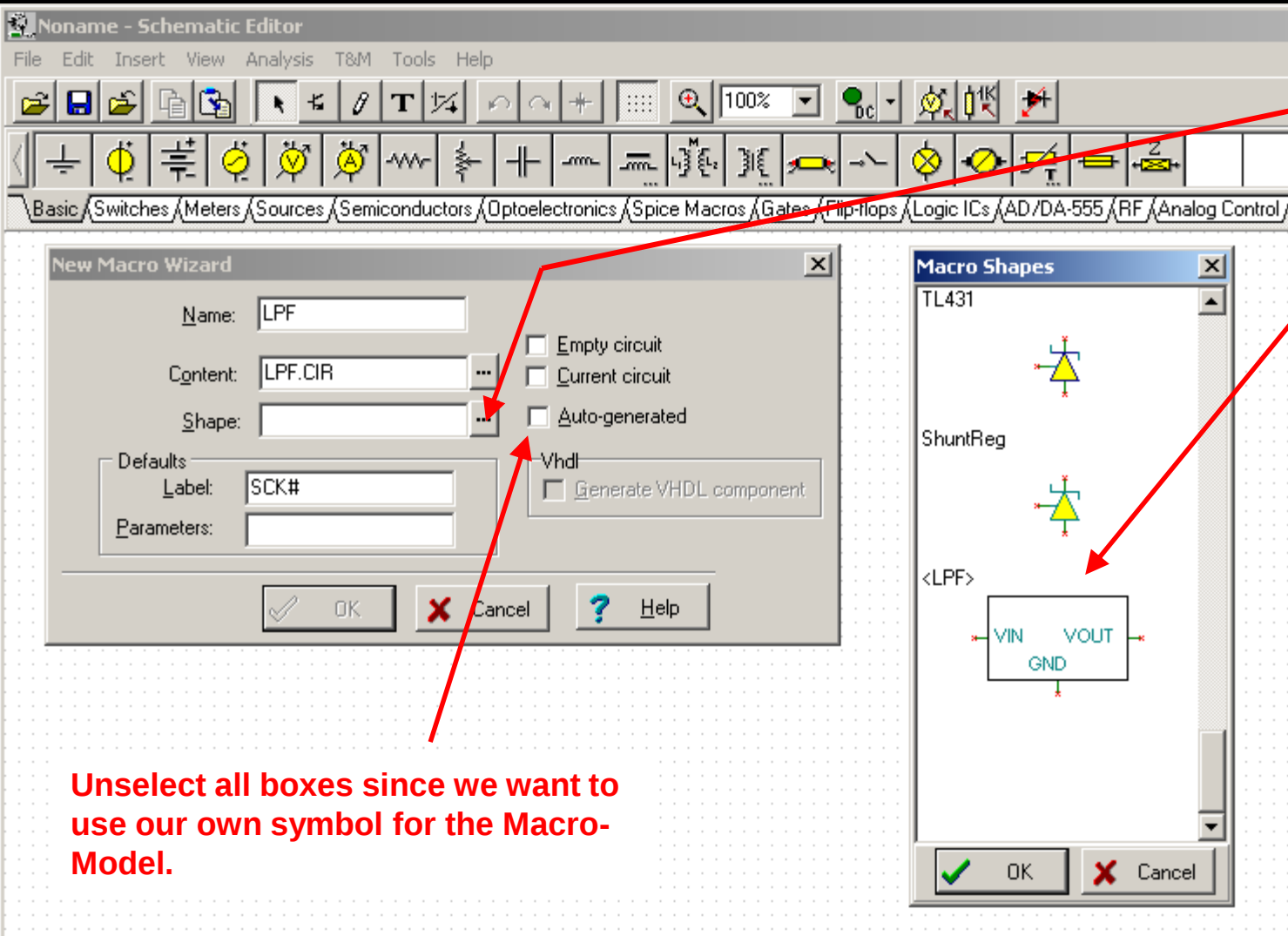
Enter "Name" for the new Macro-Model.

Click on "Content" and select the desired Text SubCircuit File. Then click "Open"



Part 1: Text Macro-Model Creation

Macro-Model Creation



The screenshot shows the 'Noname - Schematic Editor' window. The 'New Macro Wizard' dialog box is open, with the following fields and options:

- Name: LPF
- Content: LPF.CIR
- Shape: (empty)
- Defaults: Label: SCK#, Parameters: (empty)
- Options: ☐ Empty circuit, ☐ Current circuit, ☐ Auto-generated, ☐ Vhdl, ☐ Generate VHDL component

Red arrows point from the text instructions to the 'Shape' field in the 'New Macro Wizard' and the 'Macro Shapes' dialog box.

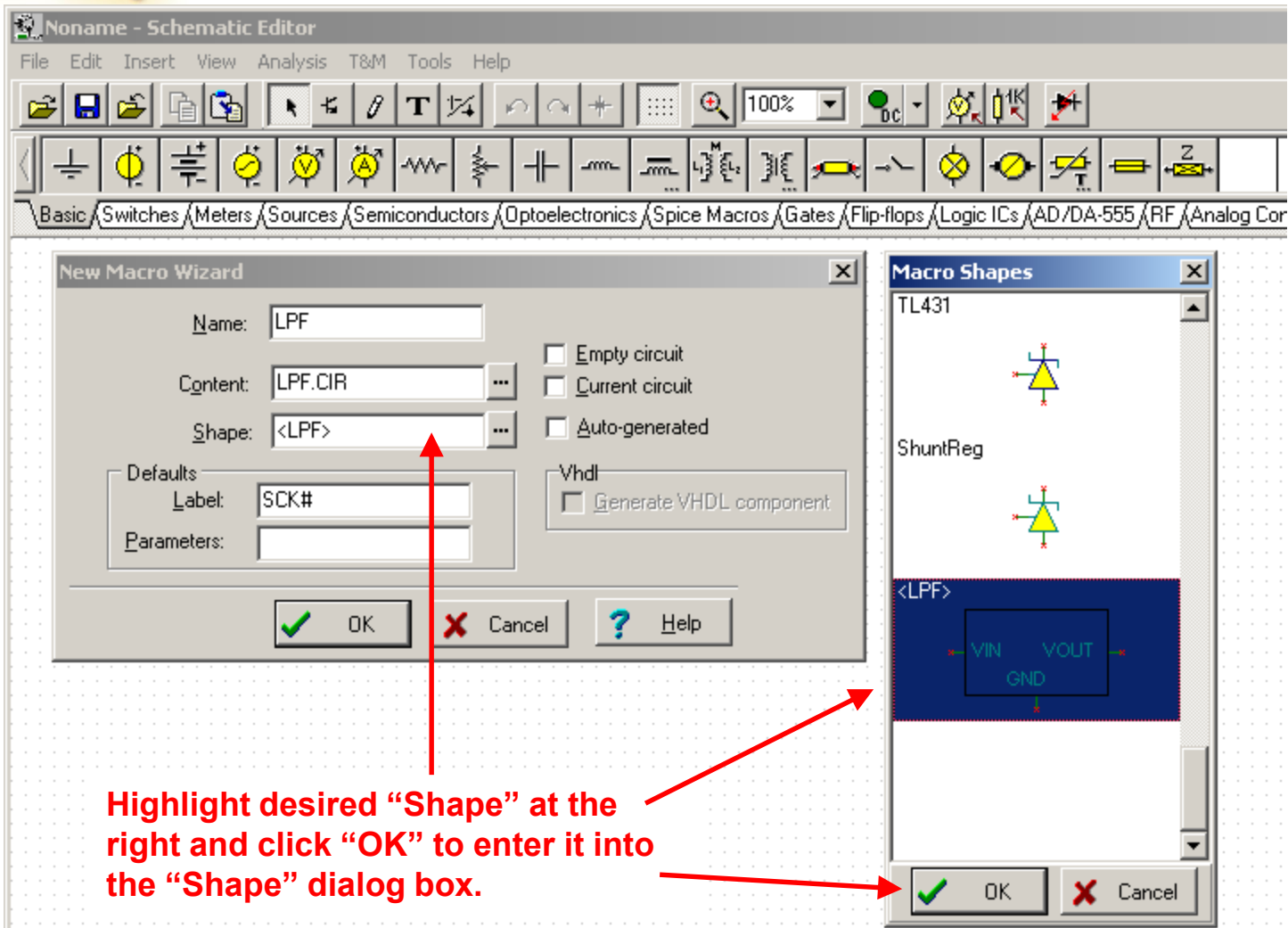
The 'Macro Shapes' dialog box shows a list of shapes, including 'TL431', 'ShuntReg', and '<LPF>'. The '<LPF>' shape is highlighted, showing a schematic symbol with three pins labeled 'VIN', 'VOUT', and 'GND'.

Click on "Shape". The **ONLY** symbol choices that will appear at the right are ones that contain pin numbers and quantities that **EXACTLY** match the Text SubCircuit. (i.e., 3 pins and pins numbered 1, 2, 3 corresponding to the Nodes in the Text Subcircuit of 1, 2, 3)

Unselect all boxes since we want to use our own symbol for the Macro-Model.

Part 1: Text Macro-Model Creation

Macro-Model Creation



The screenshot shows the Noname - Schematic Editor interface. The main window displays a toolbar with various electronic symbols and a menu bar (File, Edit, Insert, View, Analysis, T&M, Tools, Help). Below the toolbar is a category list: Basic, Switches, Meters, Sources, Semiconductors, Optoelectronics, Spice Macros, Gates, Flip-flops, Logic ICs, AD/DA-555, RF, Analog Cor.

The **New Macro Wizard** dialog box is open, showing the following fields and options:

- Name: LPF
- Content: LPF.CIR
- Shape: <LPF>
- Defaults: Label: SCK#, Parameters: (empty)
- Options: ☐ Empty circuit, ☐ Current circuit, ☐ Auto-generated
- Vhdl: ☐ Generate VHDL component

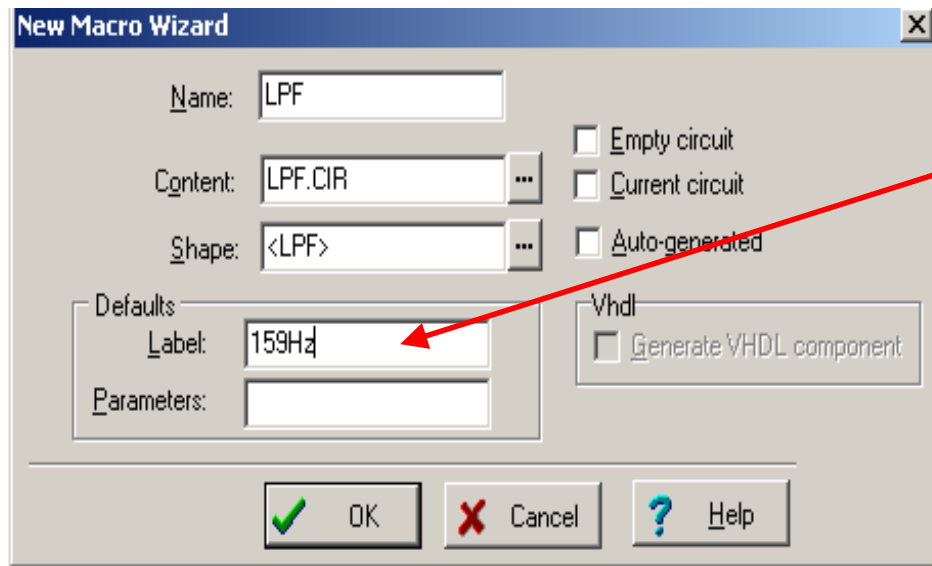
The **Macro Shapes** dialog box is also open, showing a list of shapes: TL431, ShuntReg, and <LPF>. The <LPF> shape is highlighted, and its internal circuit diagram is visible, showing a box with inputs VIN and GND, and output VOUT.

Red arrows point from the text below to the <LPF> shape in the Macro Shapes dialog and the OK button in the New Macro Wizard dialog.

Highlight desired "Shape" at the right and click "OK" to enter it into the "Shape" dialog box.

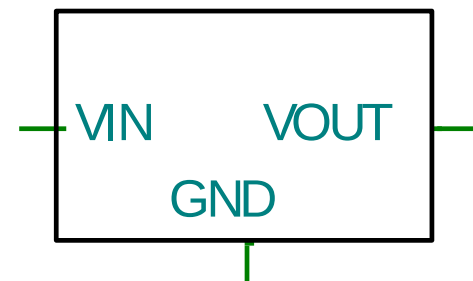
Part 1: Text Macro-Model Creation

Macro-Model Creation



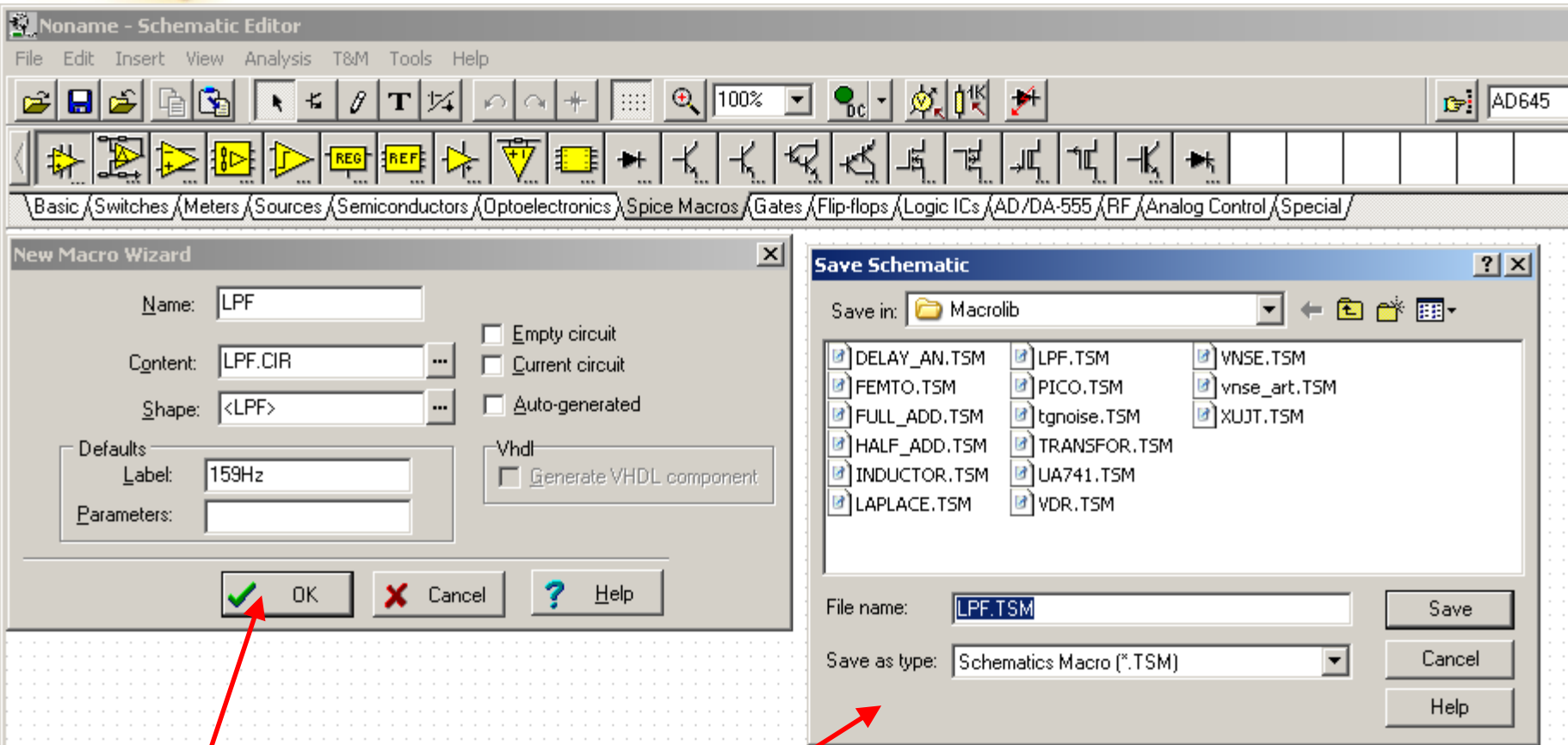
If you want a label to appear next to the Ref Designator when you place the Macro-Model on a schematic enter it here under “Label”

U2 159Hz



Part 1: Text Macro-Model Creation

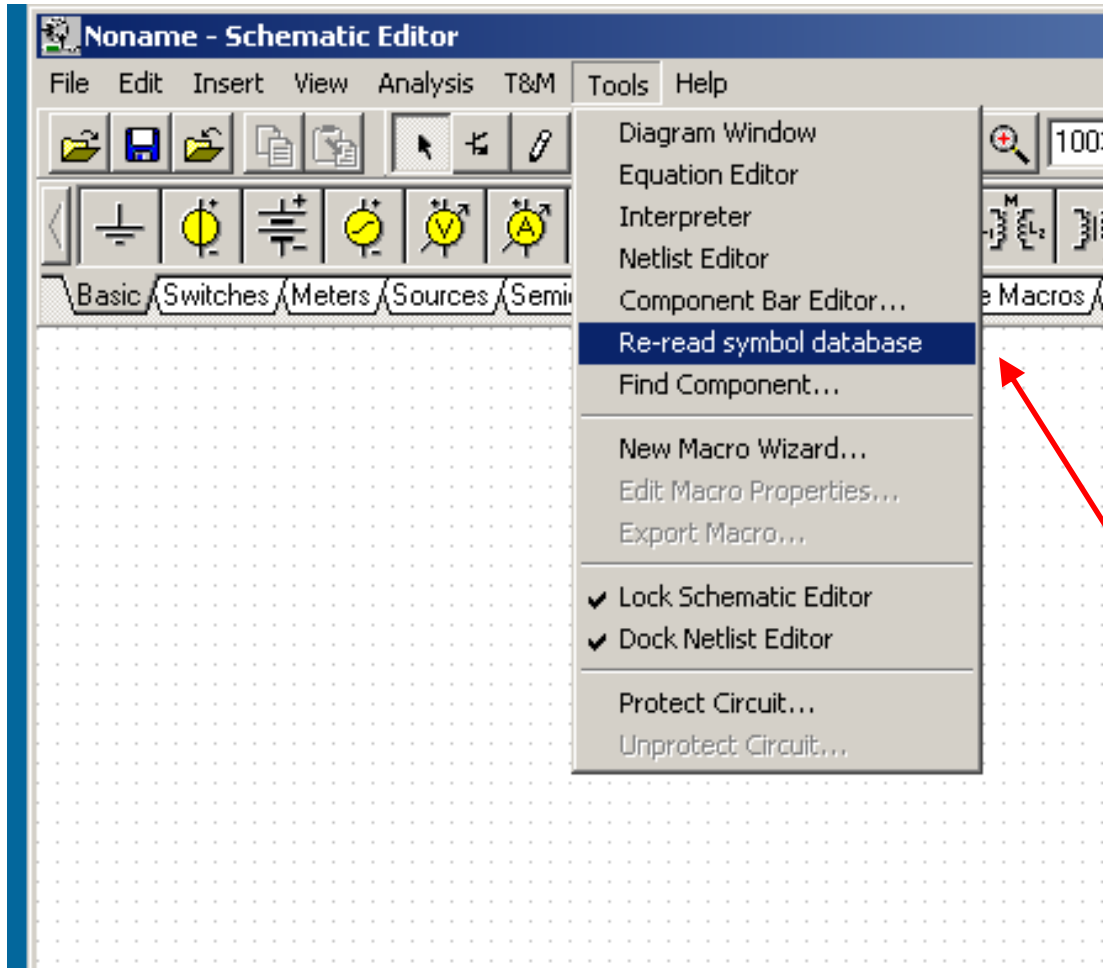
Macro-Model Creation



Finish Macro-Model creation and Save the results. Click “OK” and popup window on right will appear. Tina uses “.TSM” as Macro Schematics file extension. After creating or using default File name and choosing the folder to store the Macro-Model in click “Save”

Part 1: Text Macro-Model Creation

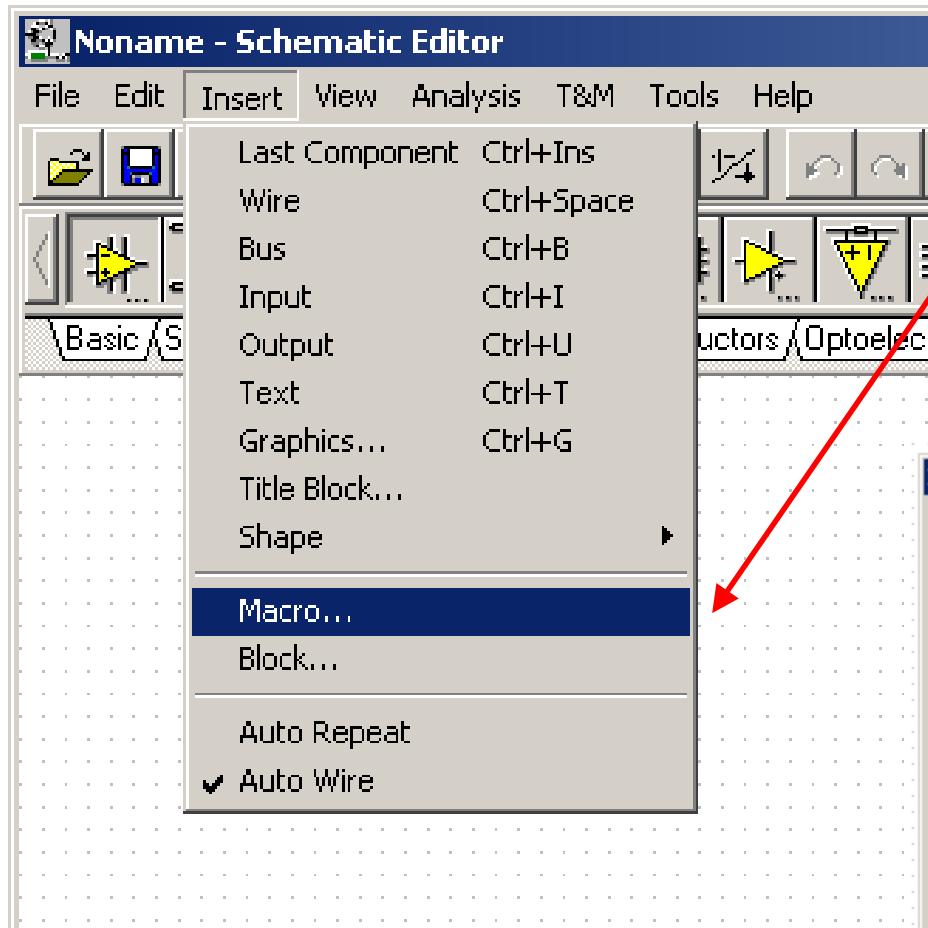
Macro-Model Use



Select "Re-read symbol database"

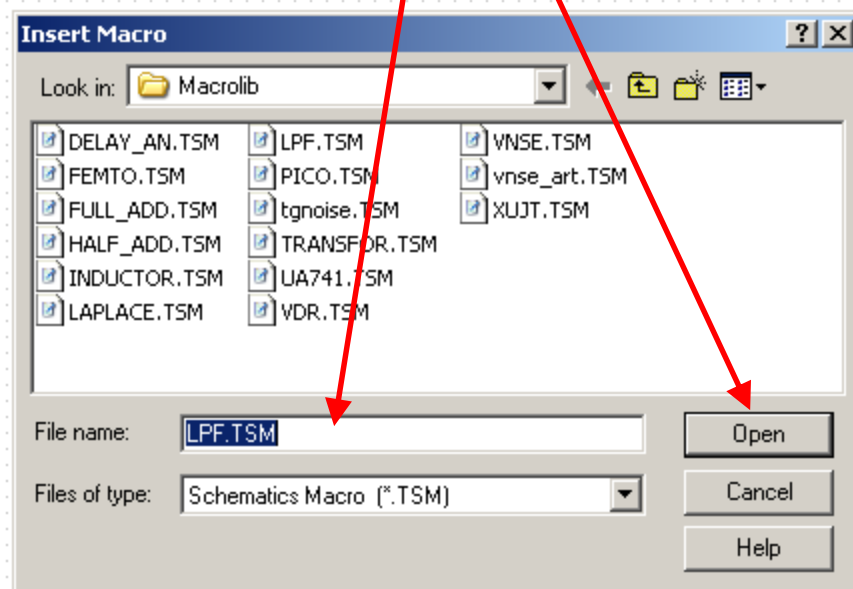
Part 1: Text Macro-Model Creation

Macro-Model Use



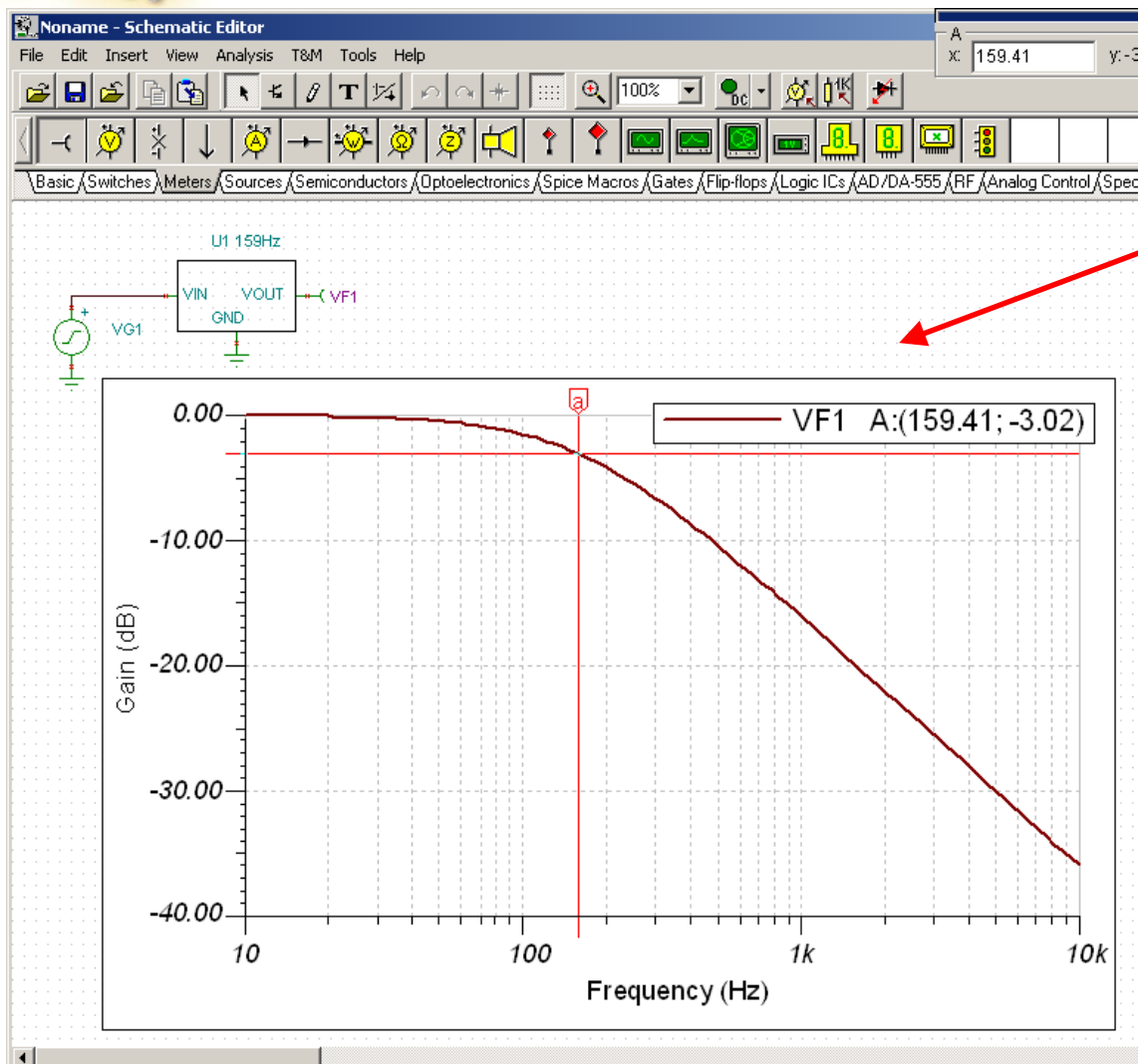
In the Tina "Schematic Editor" choose "Insert", "Macro".

From the desired folder select the Macro for insertion into the Schematic Editor and click "Open"



Part 1: Text Macro-Model Creation

Macro-Model Use



Now the Macro-Model may be connected and used in the Schematic Editor as a normal Macro-Model.

Part 2: Schematic Macro-Model Creation

Schematic SubCircuit Creation

Symbol Creation

Macro-Model Creation

Part 2: Schematic Macro-Model Creation

Schematic SubCircuit Creation

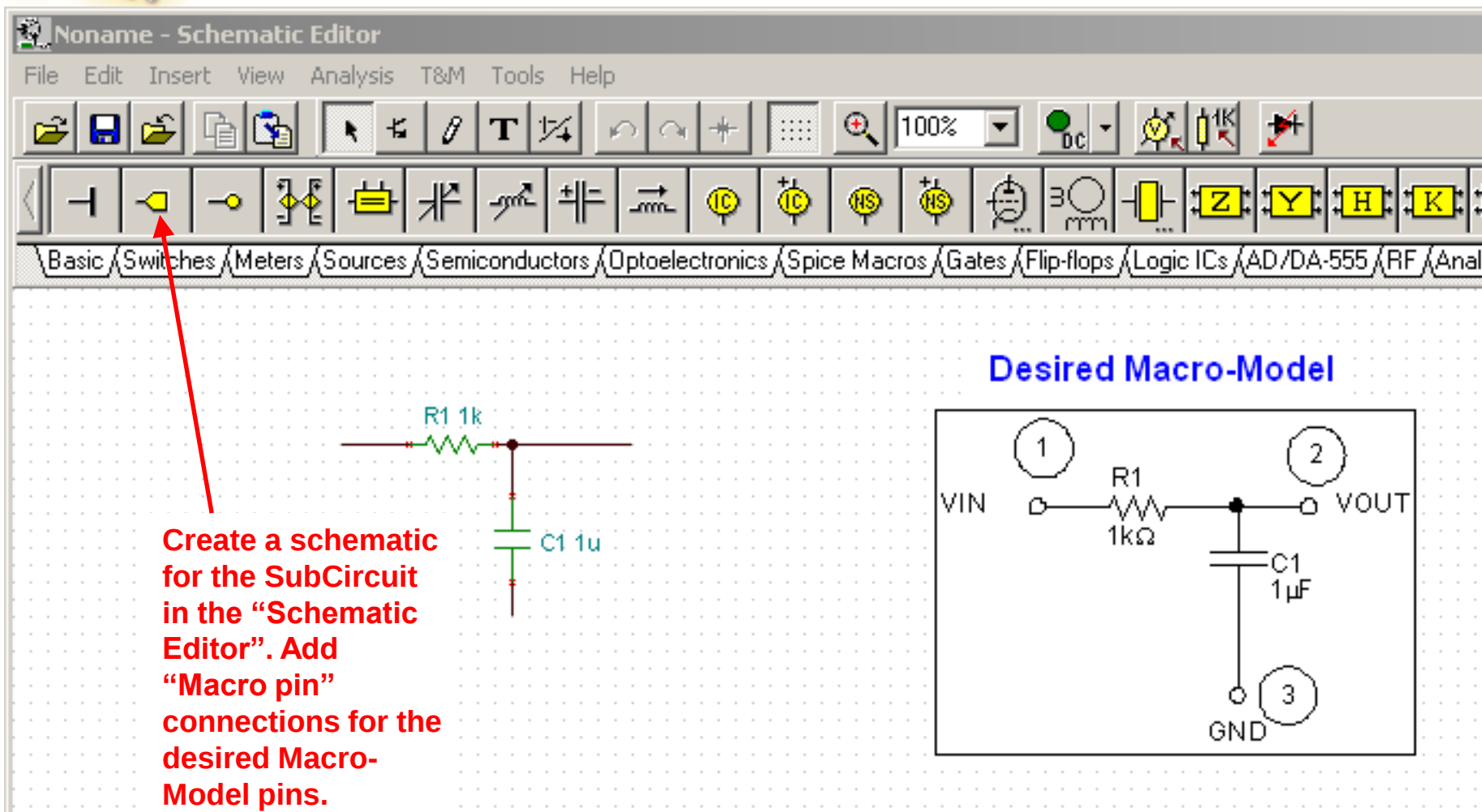
Noname - Schematic Editor

File Edit Insert View Analysis T&M Tools Help

100%

Basic Switches Meters Sources Semiconductors Optoelectronics Spice Macros Gates Flip-flops Logic ICs AD/DA-555 RF Anal

Create a schematic for the SubCircuit in the "Schematic Editor". Add "Macro pin" connections for the desired Macro-Model pins.



Desired Macro-Model

VIN

R1 1kΩ

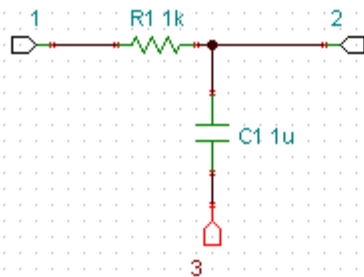
VOUT

C1 1μF

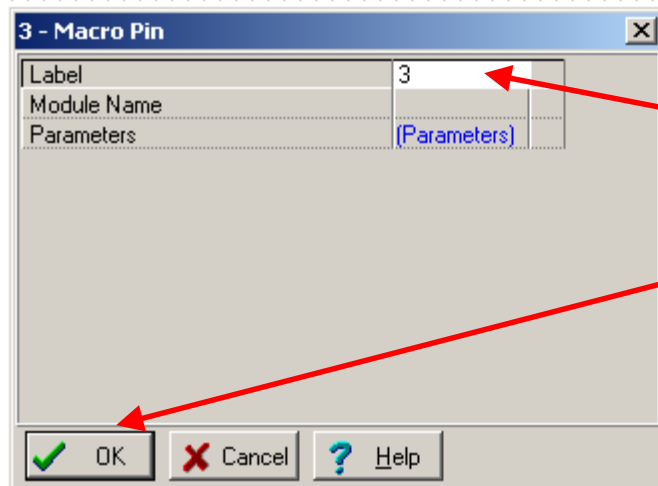
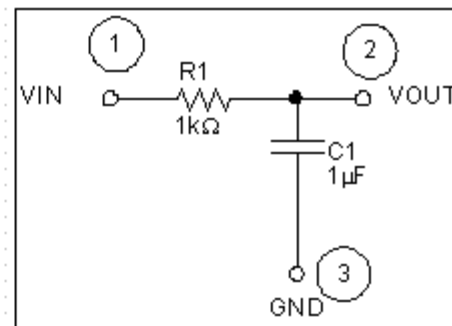
GND

Part 2: Schematic Macro-Model Creation

Schematic SubCircuit Creation

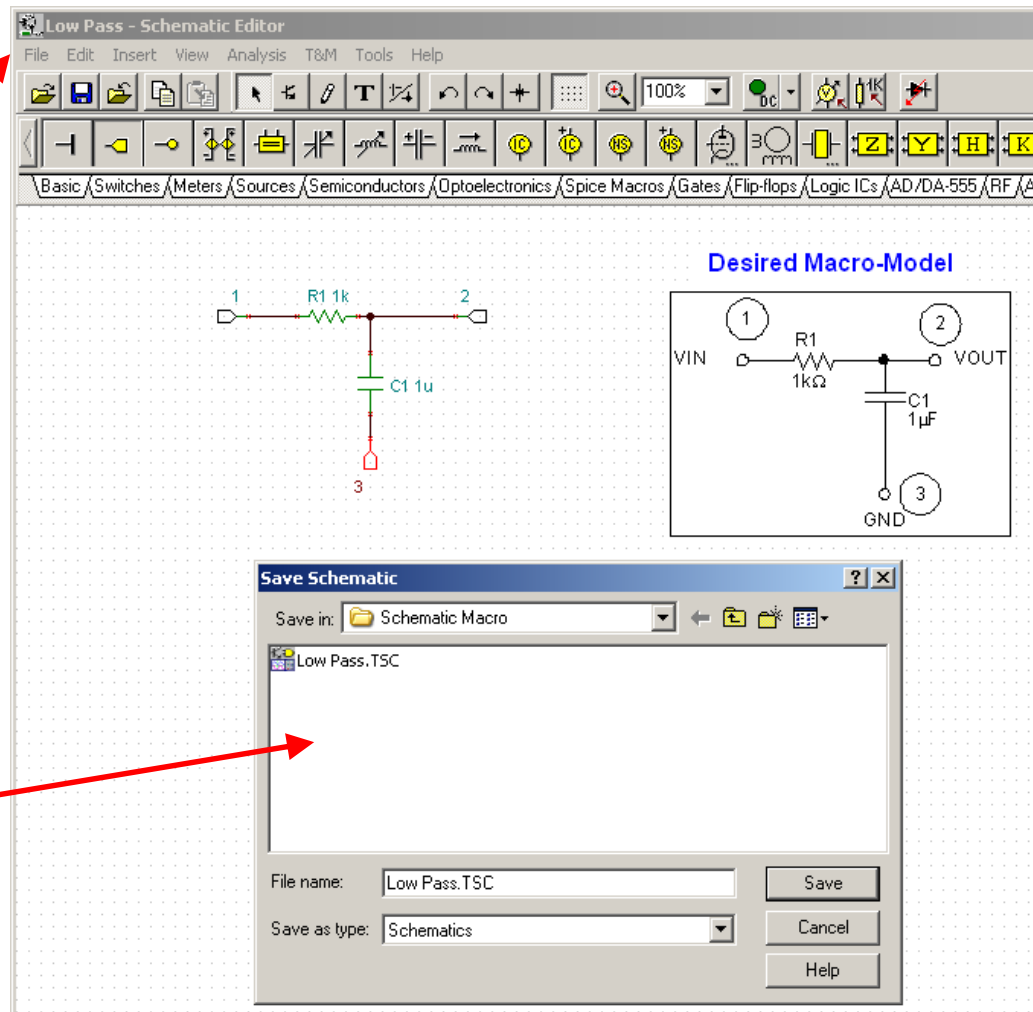


Desired Macro-Model



Double click on each “Macro pin” and type a desired pin number in the “Label” box. When completed click on “OK”.

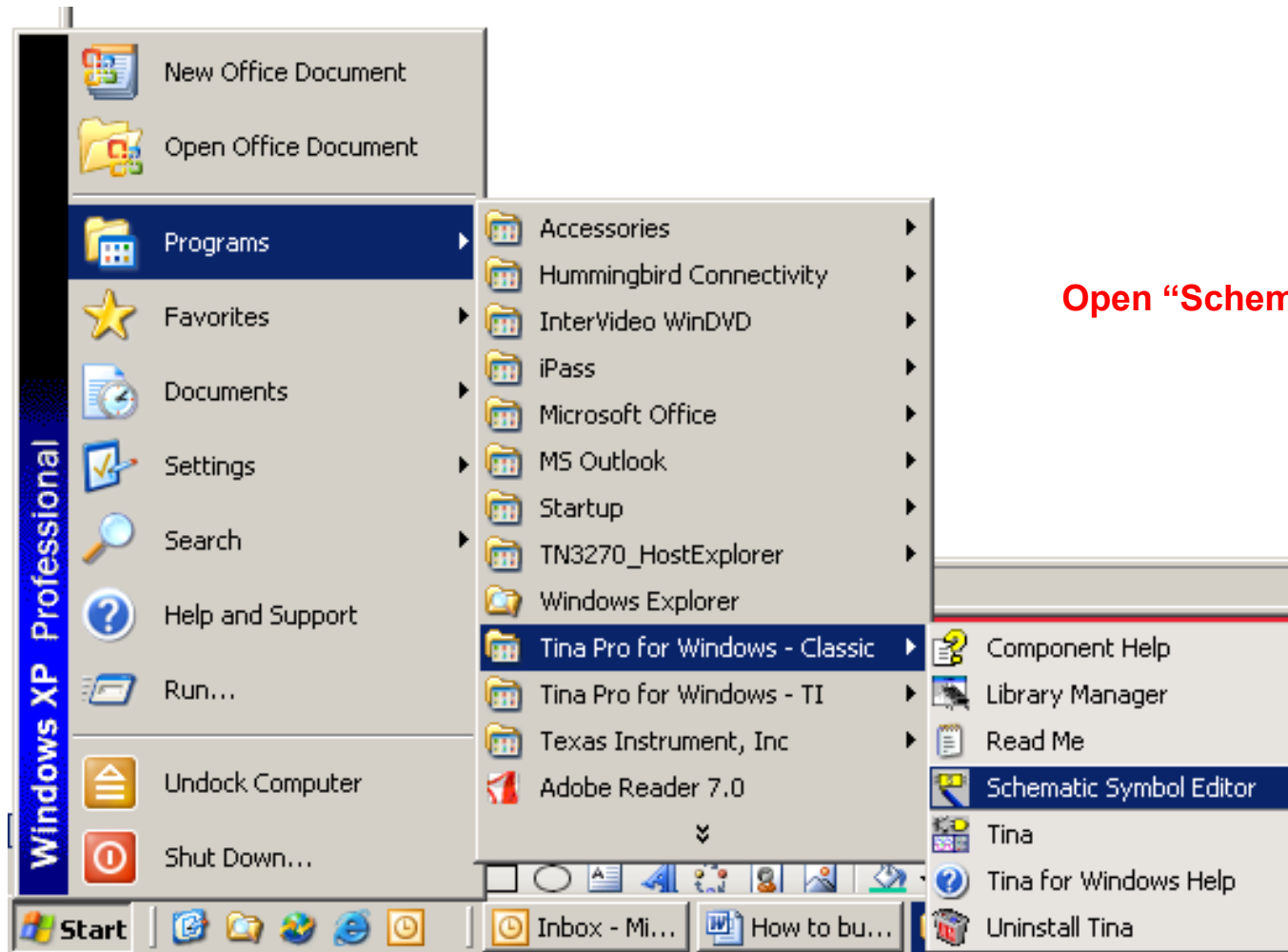
Part 2: Schematic Macro-Model Creation Schematic SubCircuit Creation



When Schematic SubCircuit is complete use "File", "Save As" in desired folder as a Tina schematic ".TSC"

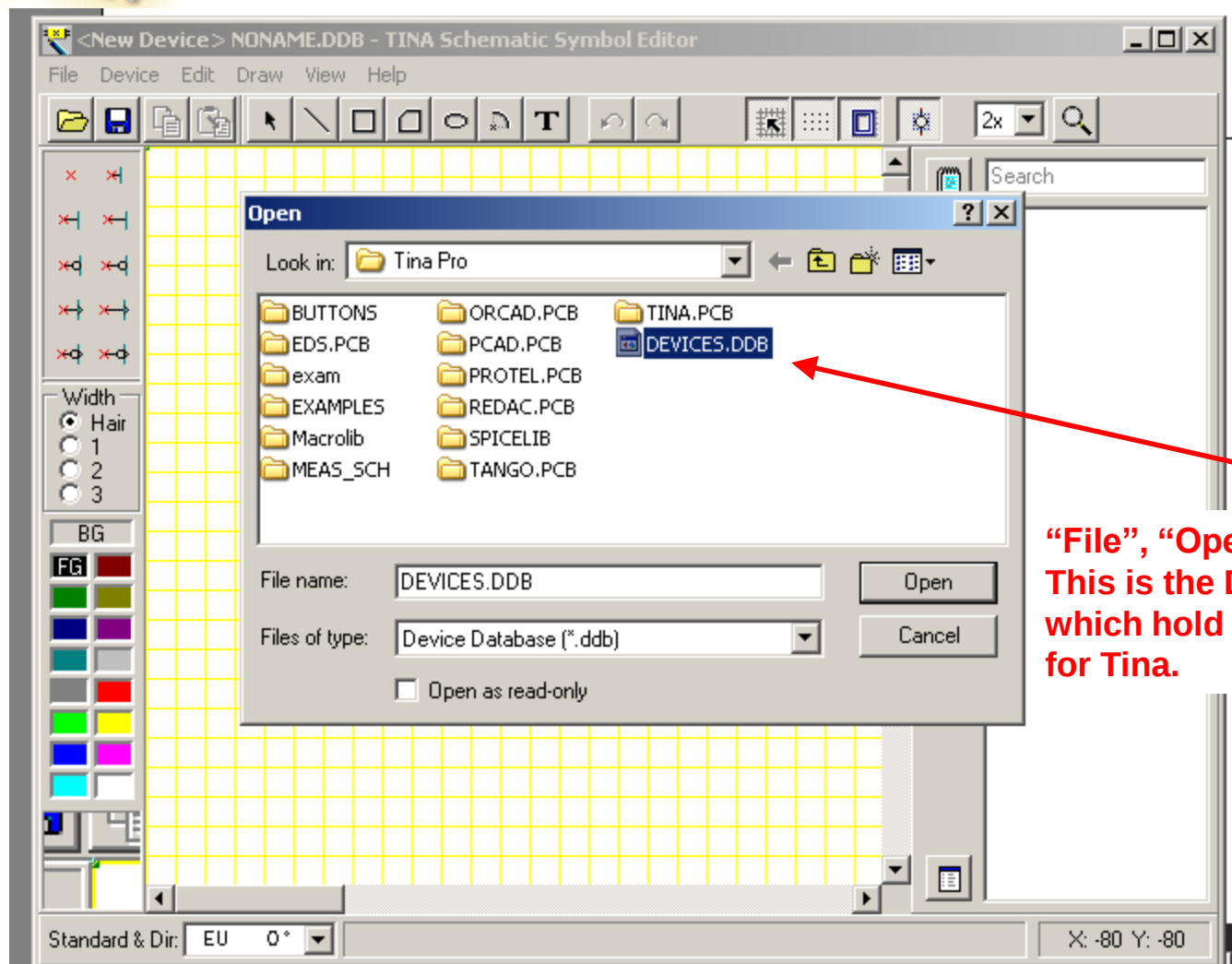
Part 2: Schematic Macro-Model Creation

Symbol Creation



Open "Schematic Symbol Editor"

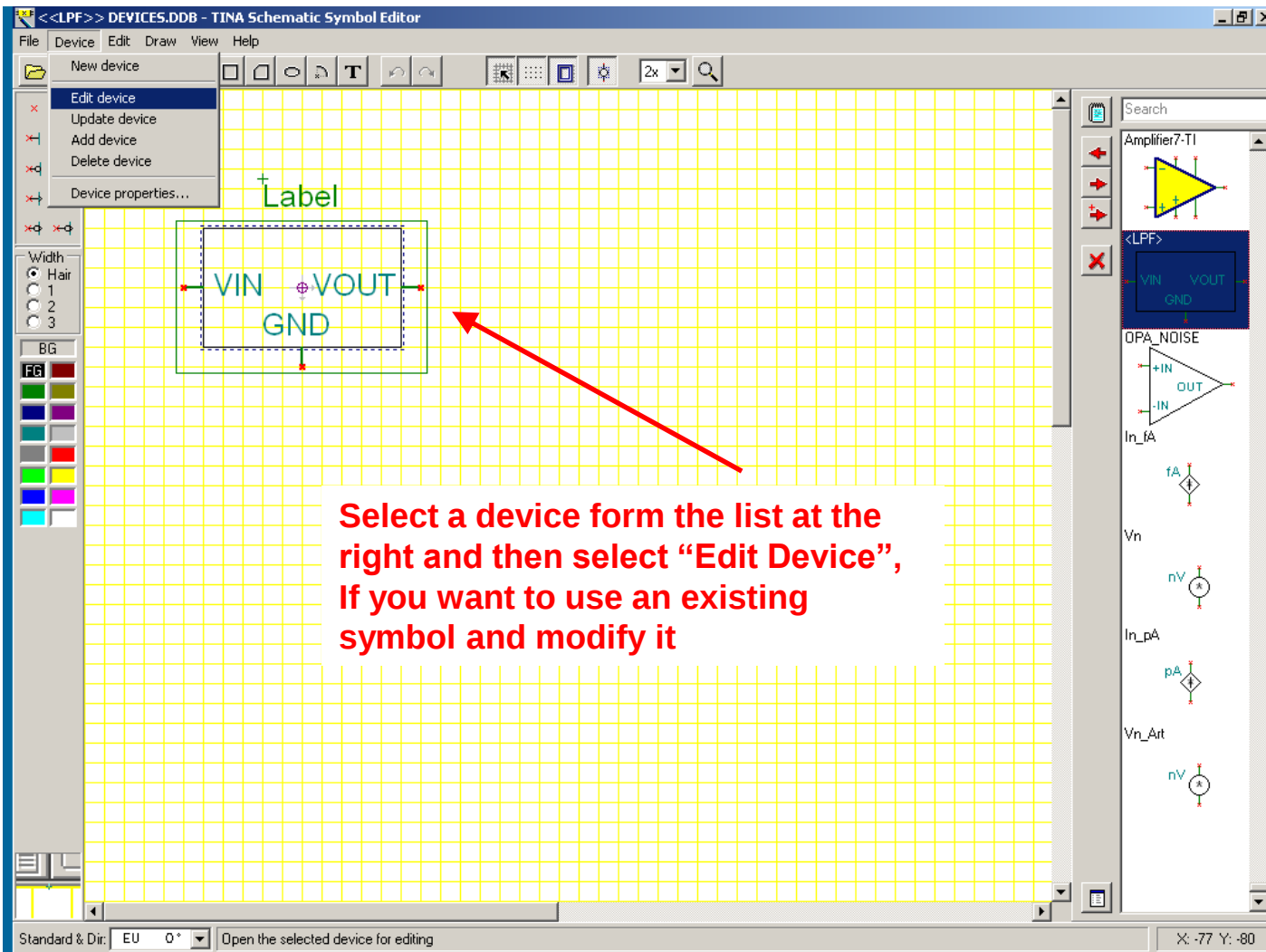
Part 2: Schematic Macro-Model Creation Symbol Creation



"File", "Open", "Devices.DDB"
This is the Device Database file
which hold all schematic symbols
for Tina.

Part 2: Schematic Macro-Model Creation

Symbol Creation - Modify



Label

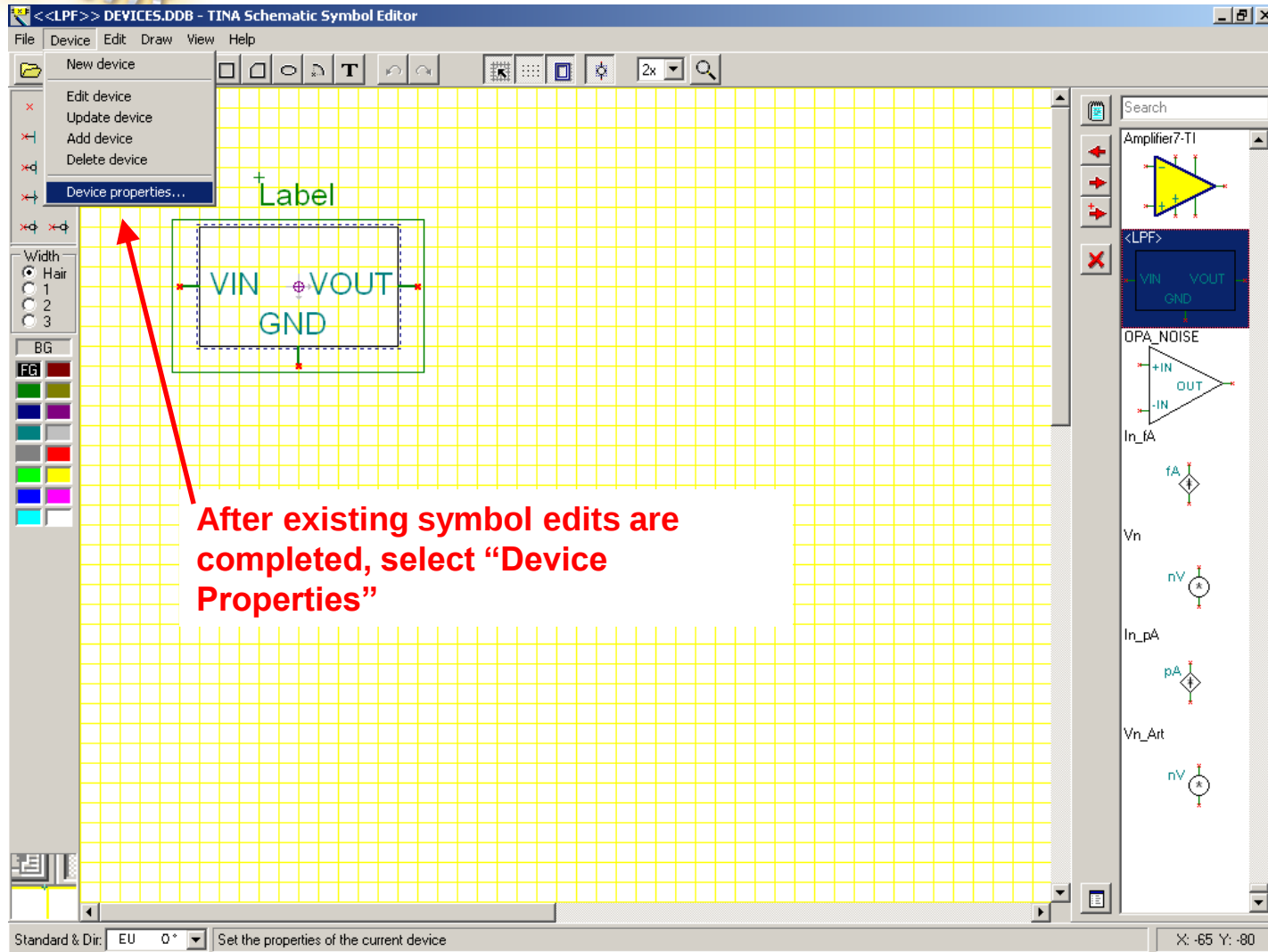
VIN VOUT
GND

Select a device form the list at the right and then select "Edit Device",
If you want to use an existing symbol and modify it

Standard & Dir: EU 0° Open the selected device for editing X: -77 Y: -80

Part 2: Schematic Macro-Model Creation

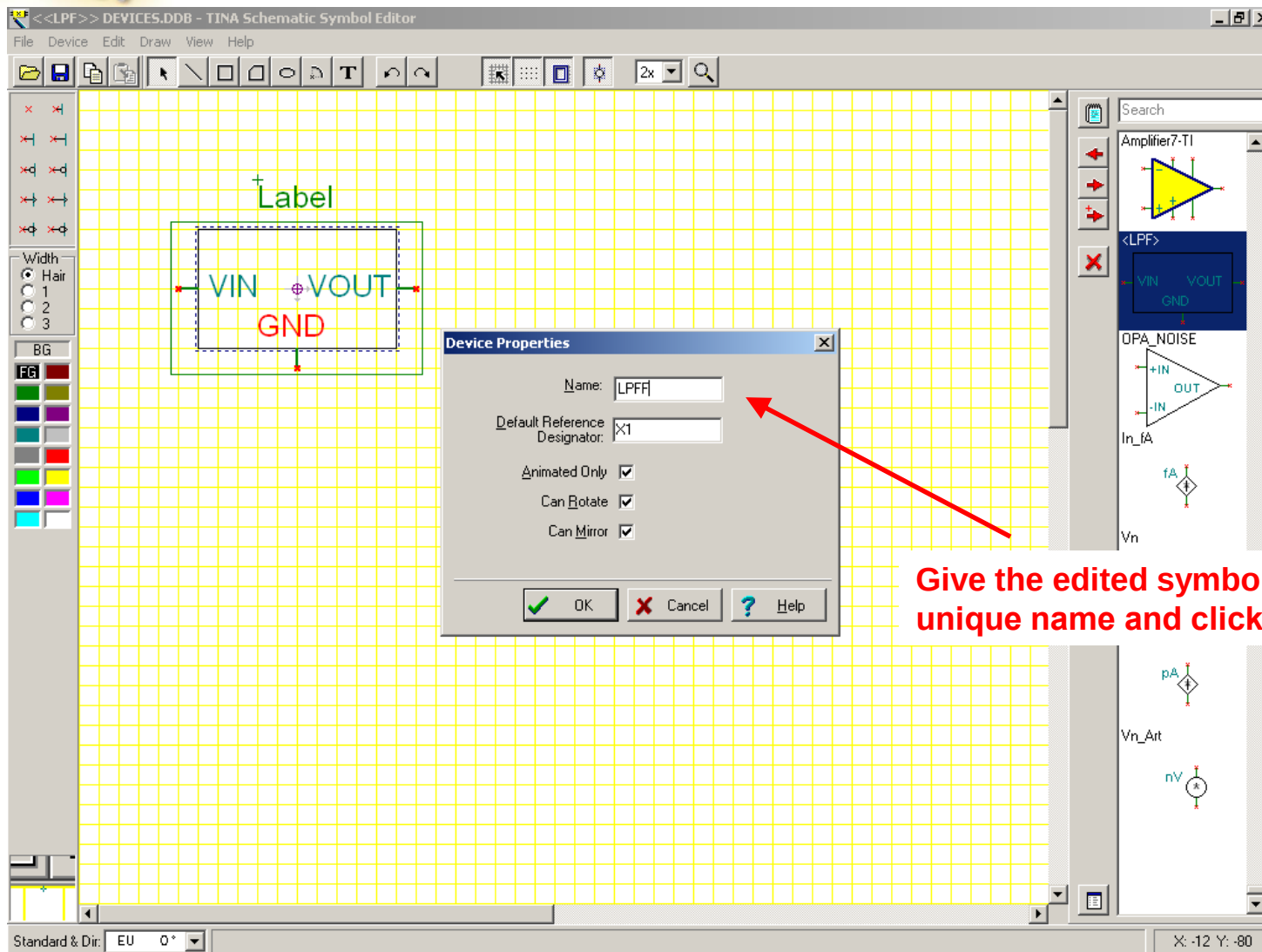
Symbol Creation - Modify



Standard & Dir: EU 0° Set the properties of the current device

After existing symbol edits are completed, select "Device Properties"

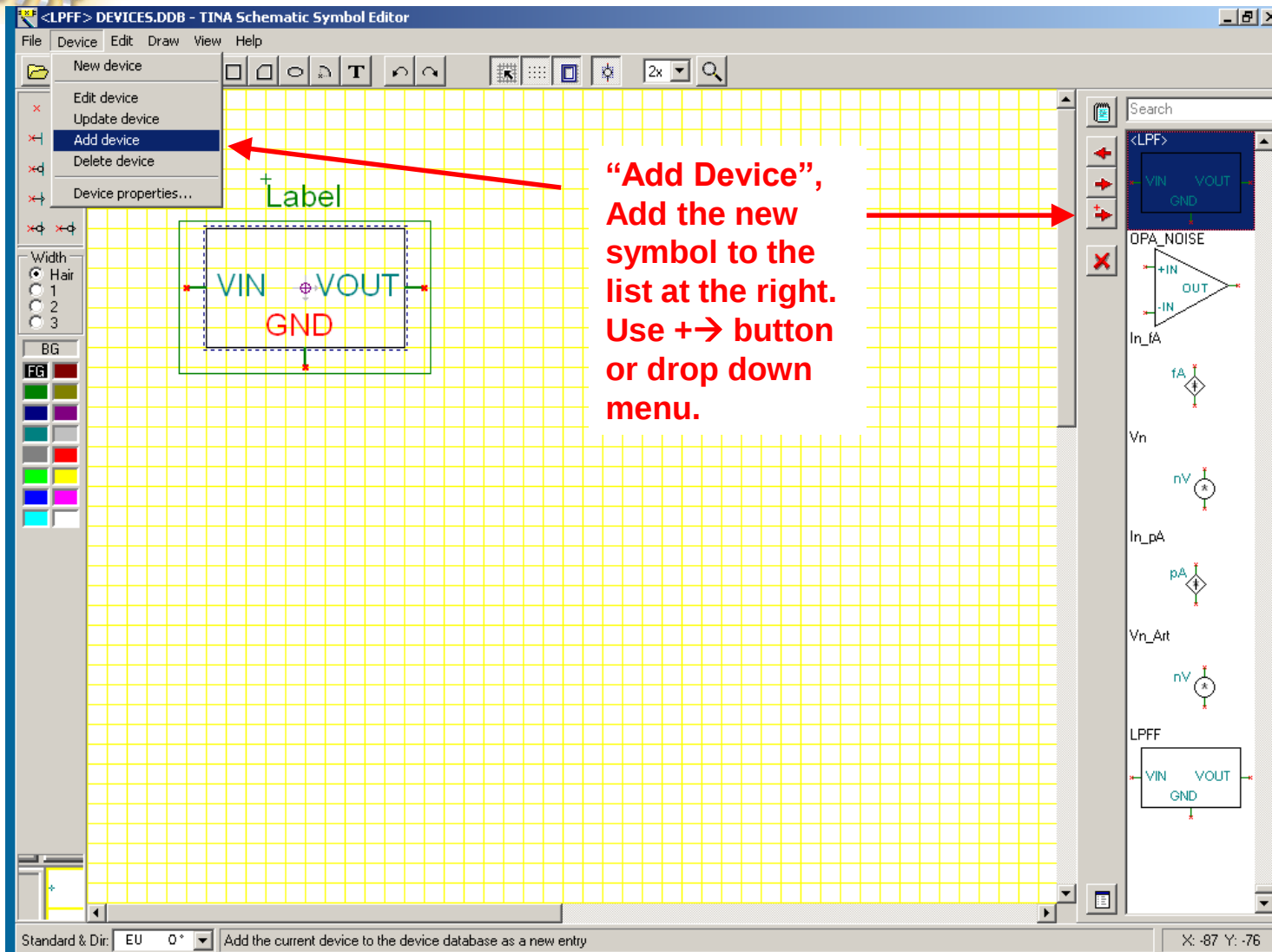
Part 2: Schematic Macro-Model Creation Symbol Creation - Modify



Give the edited symbol a new, unique name and click "OK"

Part 2: Schematic Macro-Model Creation

Symbol Creation - Modify



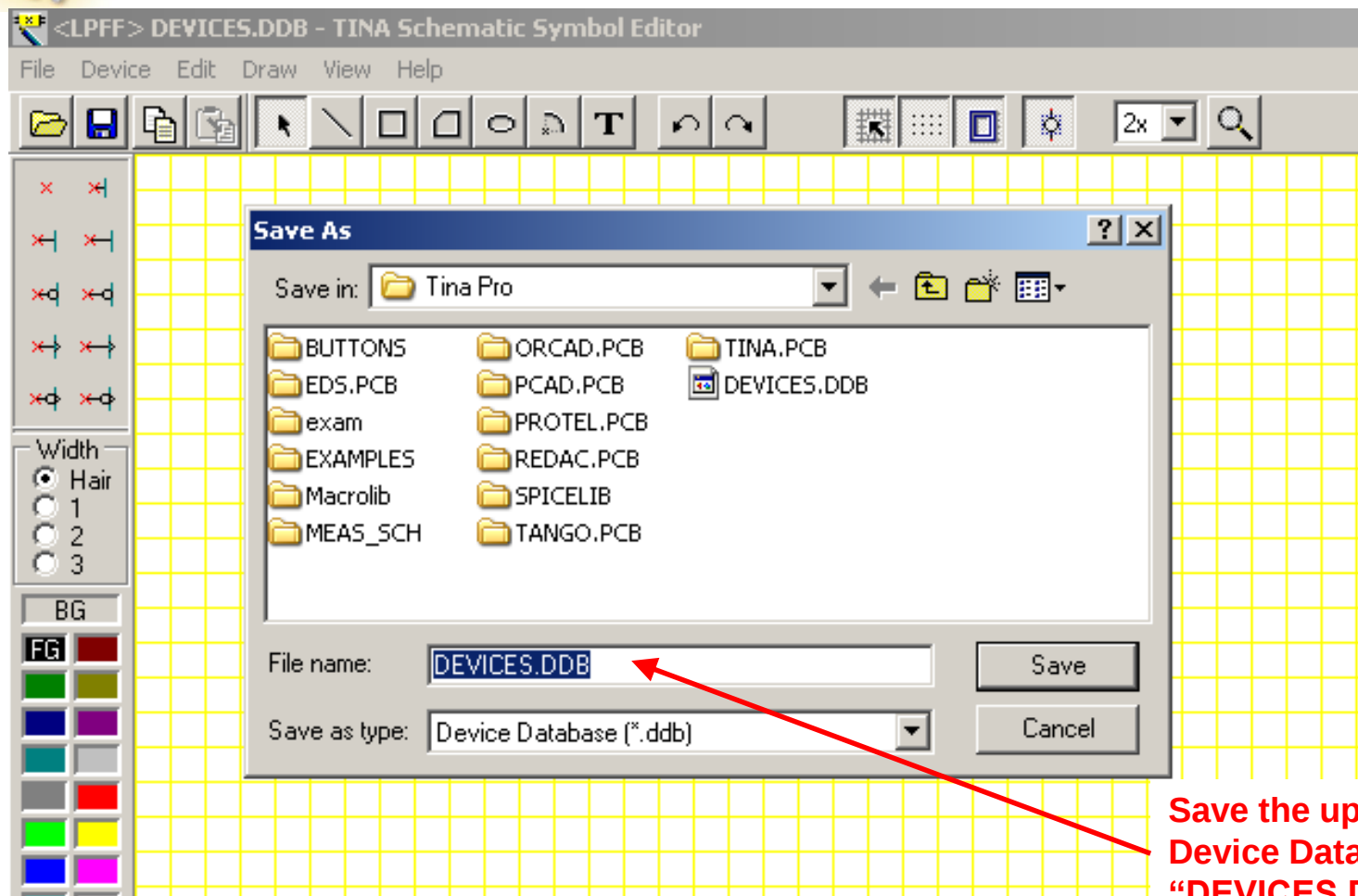
The screenshot shows the TINA Schematic Symbol Editor interface. The main workspace is a yellow grid. On the left, a menu is open with the following options: New device, Edit device, Update device, Add device (highlighted), Delete device, and Device properties... A red arrow points from the 'Add device' menu item to the central workspace. In the workspace, a schematic symbol for a device named 'LPFF' is shown, enclosed in a green dashed box. The symbol has three pins: 'VIN' (input), 'VOUT' (output), and 'GND' (ground). A green label 'Label' is placed above the symbol. On the right, a list of devices is shown, including 'LPFF', 'OPA_NOISE', 'In_fA', 'Vn', 'nV', 'In_pA', 'pA', 'Vn_Art', 'nV', and 'LPFF'. A red arrow points from the 'Add device' menu item to the 'LPFF' device in the list. A red text box in the center of the workspace contains the following text: "Add Device", Add the new symbol to the list at the right. Use +→ button or drop down menu.

Standard & Dir: EU 0° Add the current device to the device database as a new entry

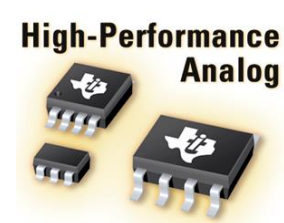
X: -87 Y: -76

Part 2: Schematic Macro-Model Creation

Symbol Creation - Modify

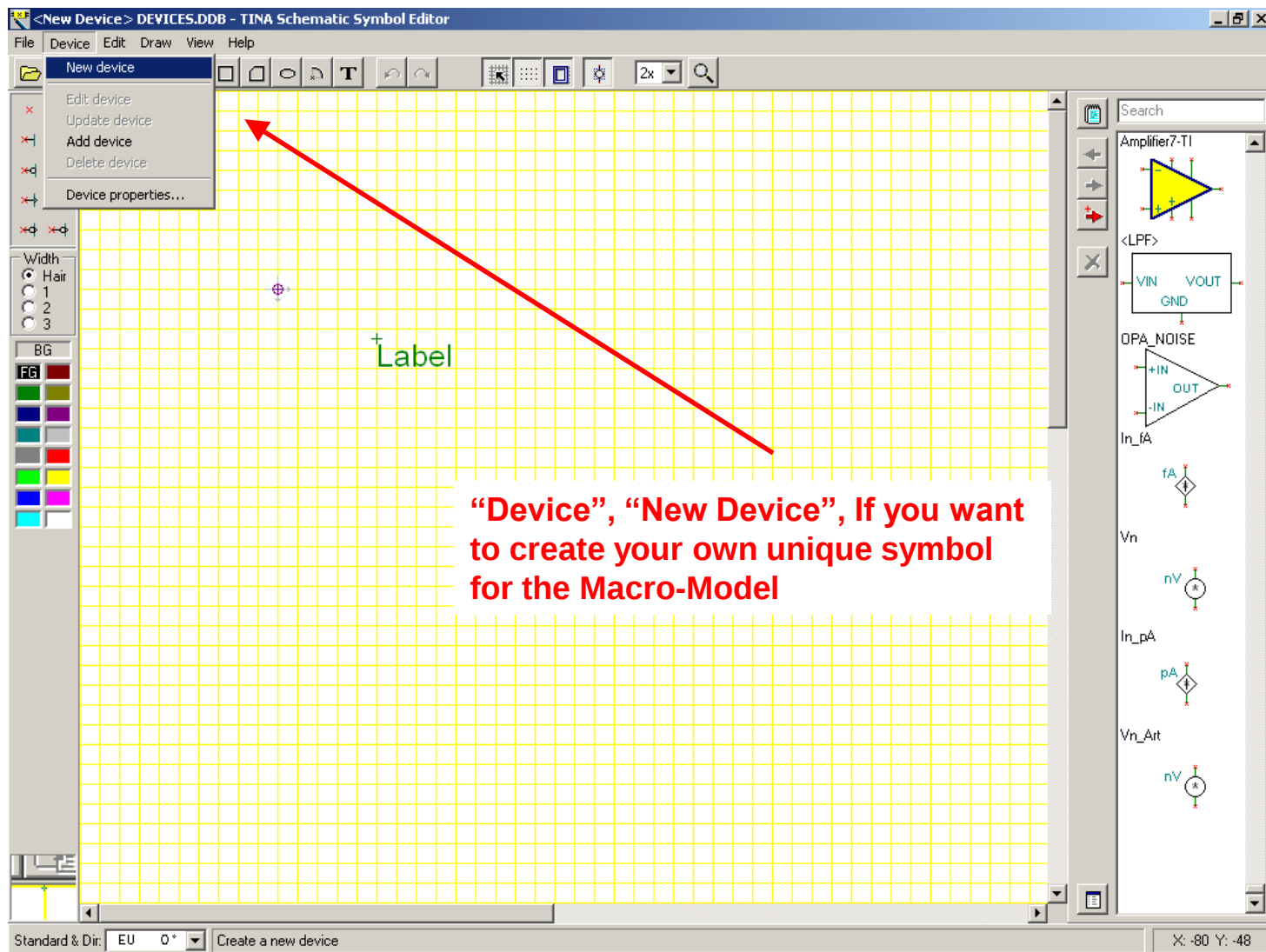


Warning: Backup the original "DEVICES.DDB" File in case there is a problem.



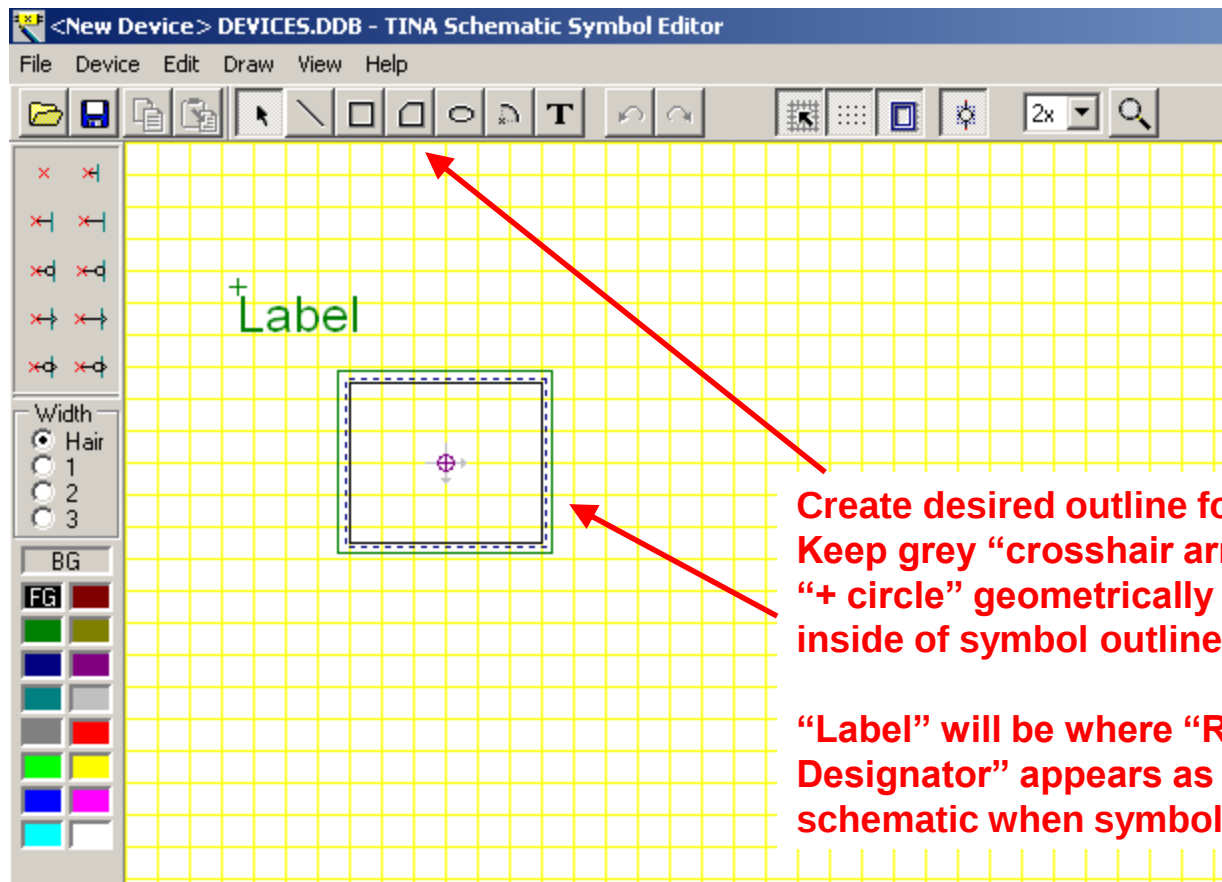
Part 2: Schematic Macro-Model Creation

Symbol Creation - New



Part 2: Schematic Macro-Model Creation

Symbol Creation - New

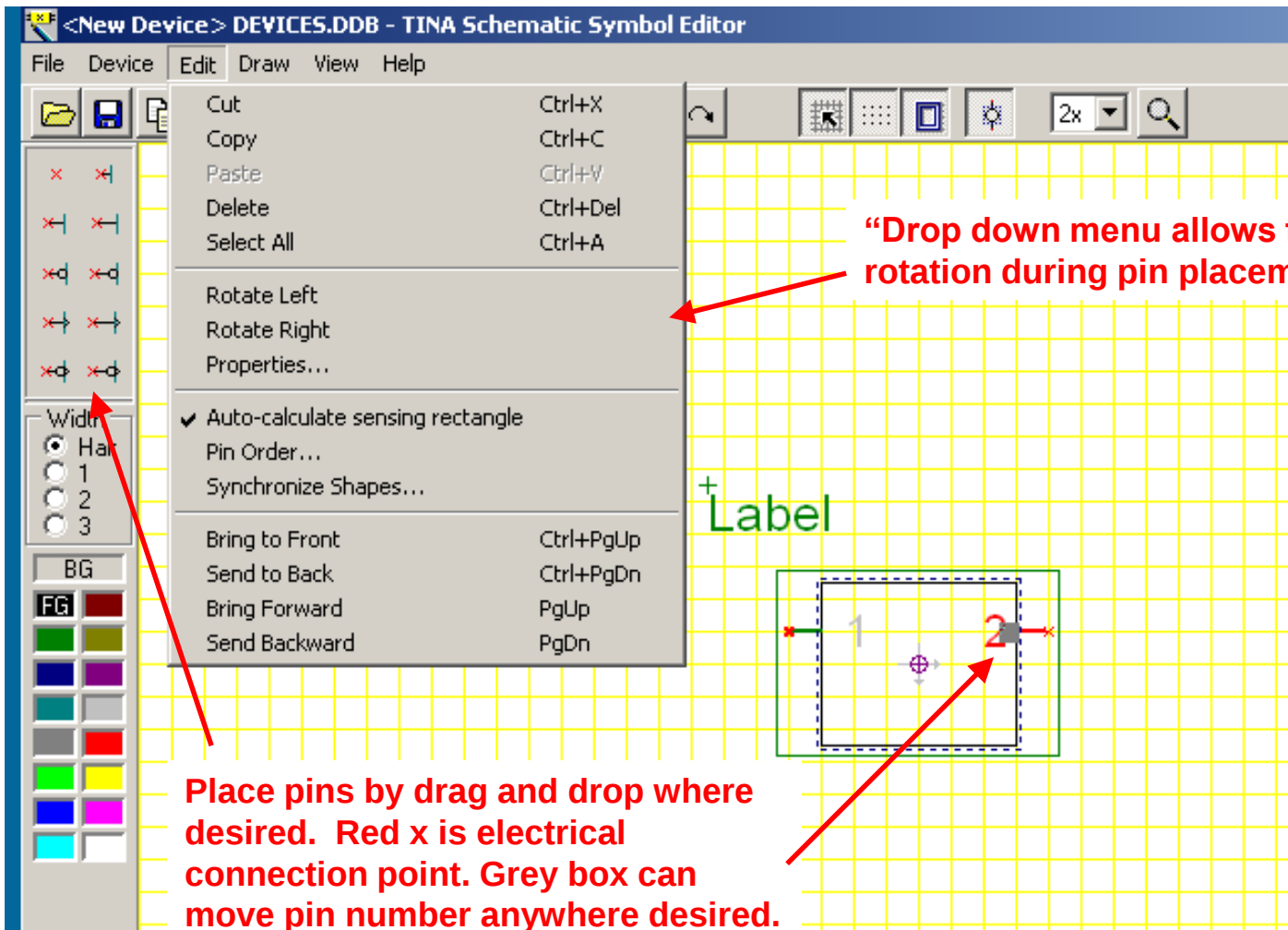


Create desired outline for symbol. Keep grey "crosshair arrows" and "+ circle" geometrically centered inside of symbol outline.

"Label" will be where "Ref Designator" appears as default on schematic when symbol is placed.

Part 2: Schematic Macro-Model Creation

Symbol Creation - New



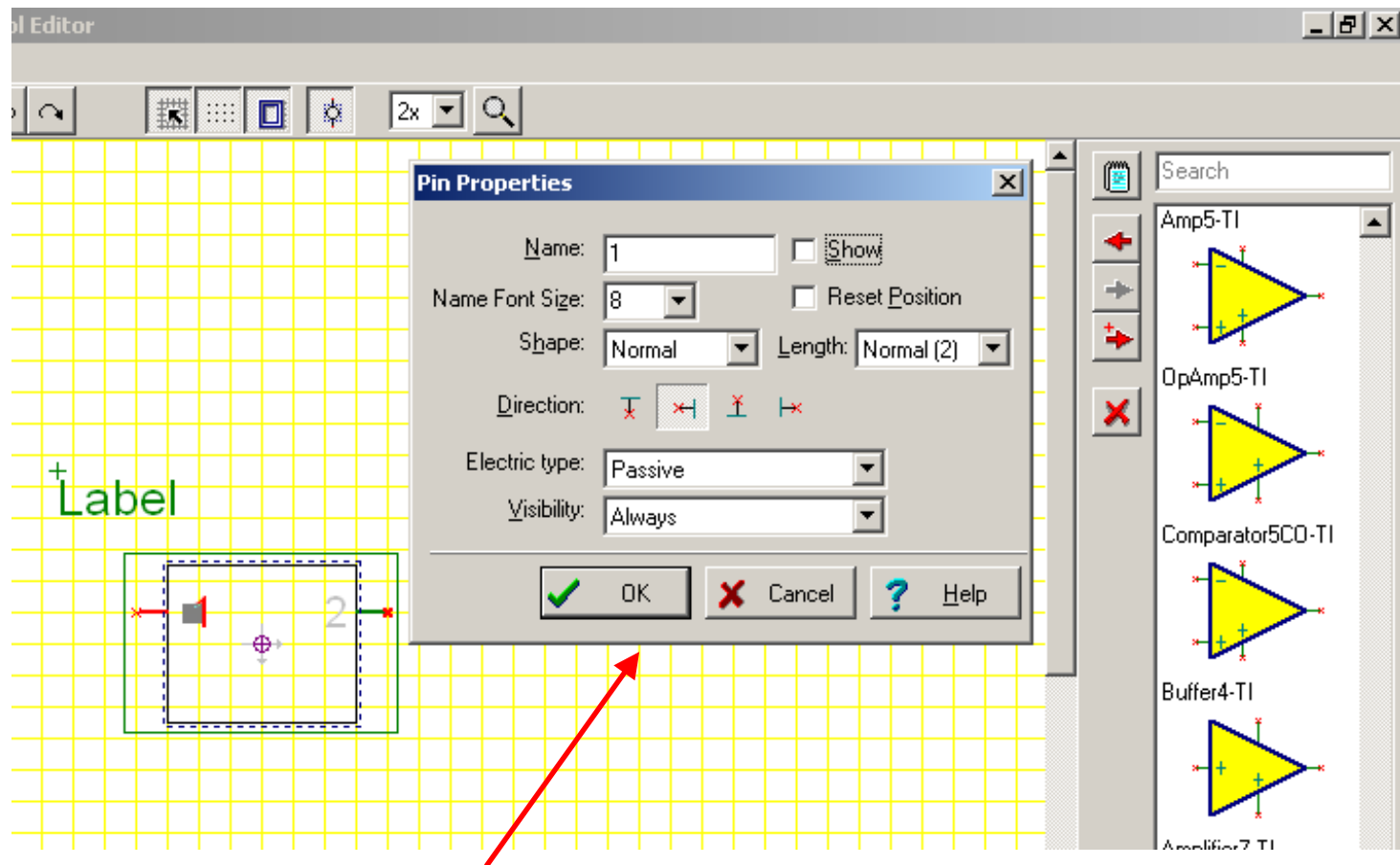
The screenshot shows the 'TINA Schematic Symbol Editor' window. The 'Edit' menu is open, displaying various editing options. A red arrow points from the 'Pin' icon in the left toolbar to the 'Edit' menu. Another red arrow points from the 'Drop down menu allows for pin rotation during pin placement' text to the 'Rotate Left' and 'Rotate Right' options in the 'Edit' menu. A third red arrow points from the 'Place pins by drag and drop where desired. Red x is electrical connection point. Grey box can move pin number anywhere desired.' text to the schematic diagram. The schematic diagram shows a green label 'Label' with a '+' sign, and a dashed box containing two pins. Pin 1 is on the left and Pin 2 is on the right. Pin 2 has a red 'x' at its electrical connection point and a grey box around its number '2'. A red arrow points from the text to the grey box around pin 2.

“Drop down menu allows for pin rotation during pin placement”

Place pins by drag and drop where desired. Red x is electrical connection point. Grey box can move pin number anywhere desired.

Part 2: Schematic Macro-Model Creation

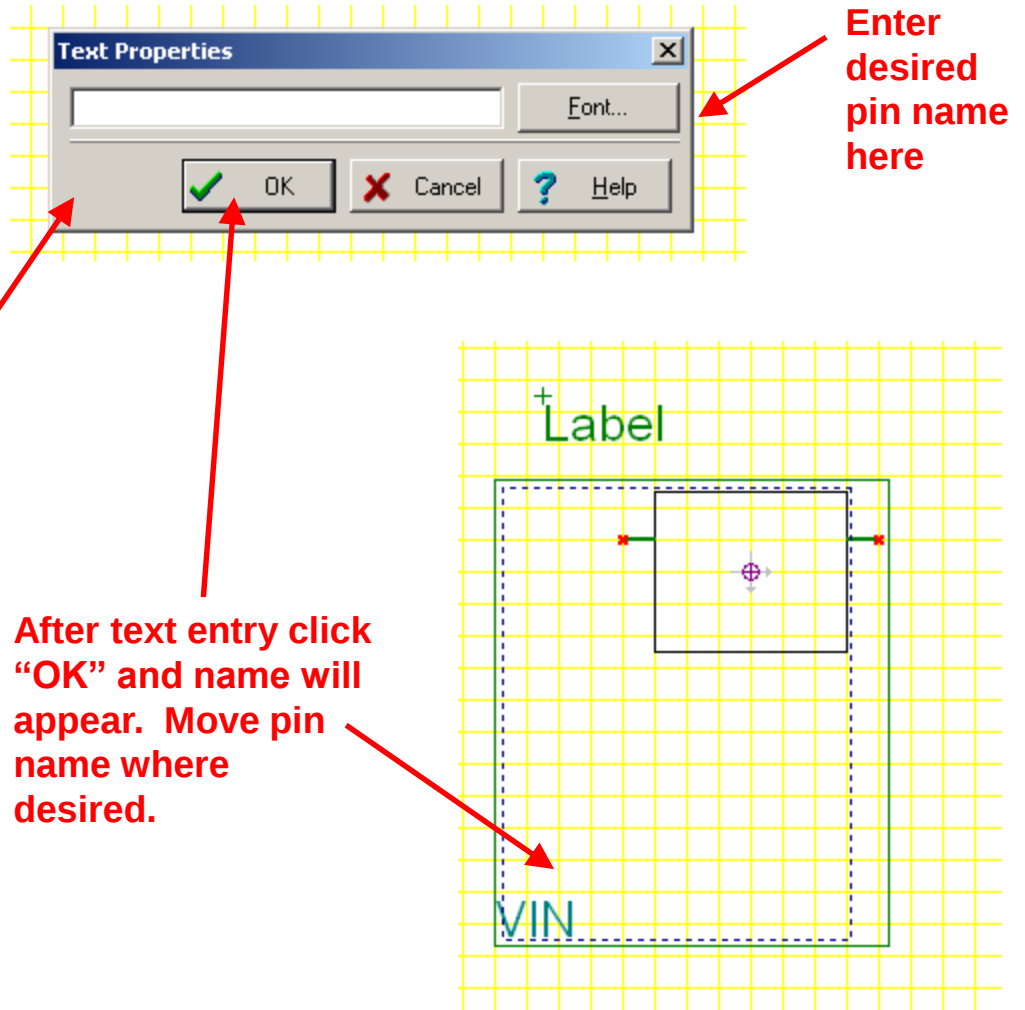
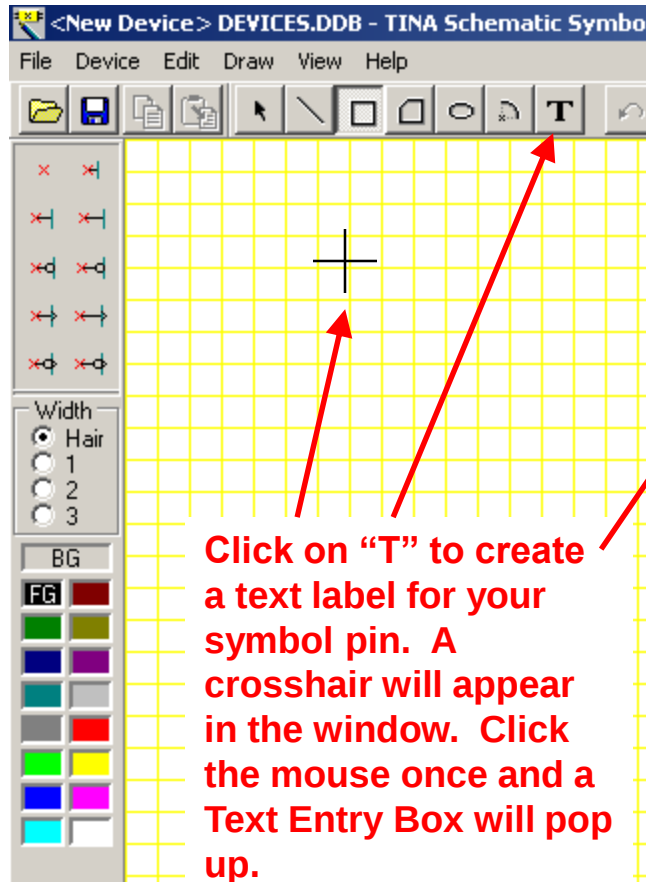
Symbol Creation - New



Double click on each pin to select the “Pin Properties”. If you want a name on your pin unselect “Show”. For your Symbol to be available to build your Macro-Model form the Text SubCircuit it must be built with the EXACT same number of pins and use the same pin numbers on the pins as your Text SubCircuit

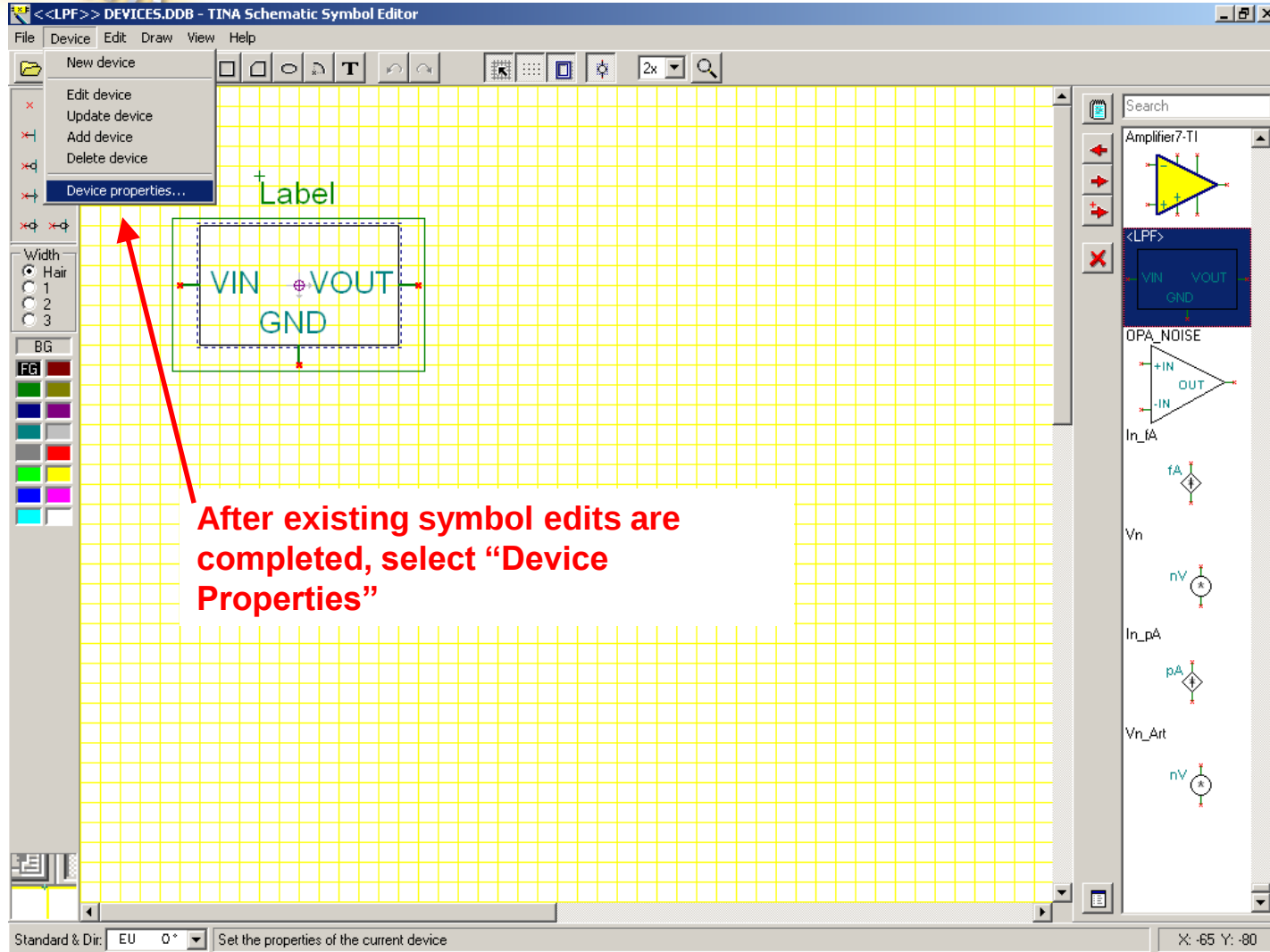
Part 2: Schematic Macro-Model Creation

Symbol Creation - New



Part 2: Schematic Macro-Model Creation

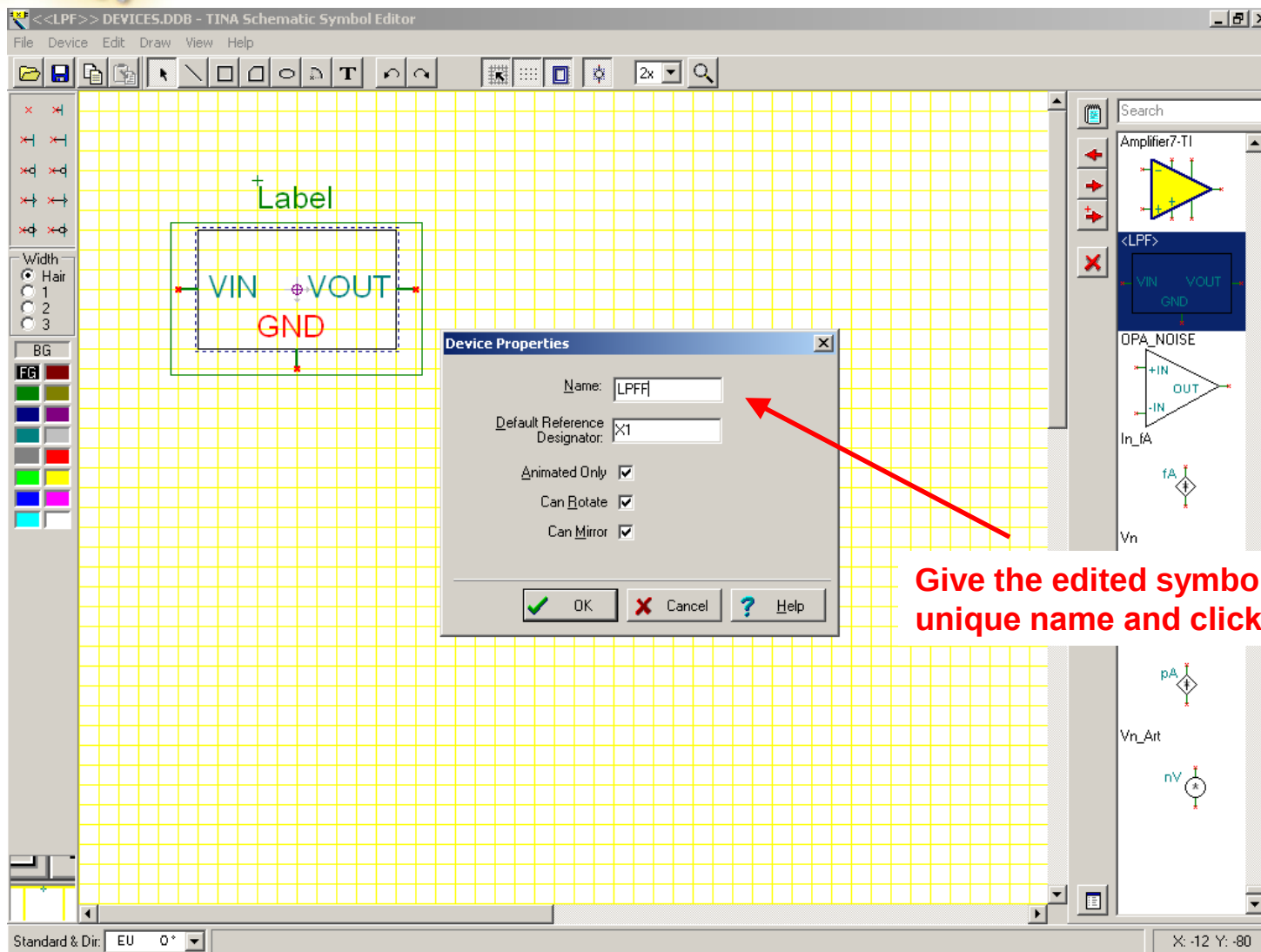
Symbol Creation - New



After existing symbol edits are completed, select "Device Properties"

Part 2: Schematic Macro-Model Creation

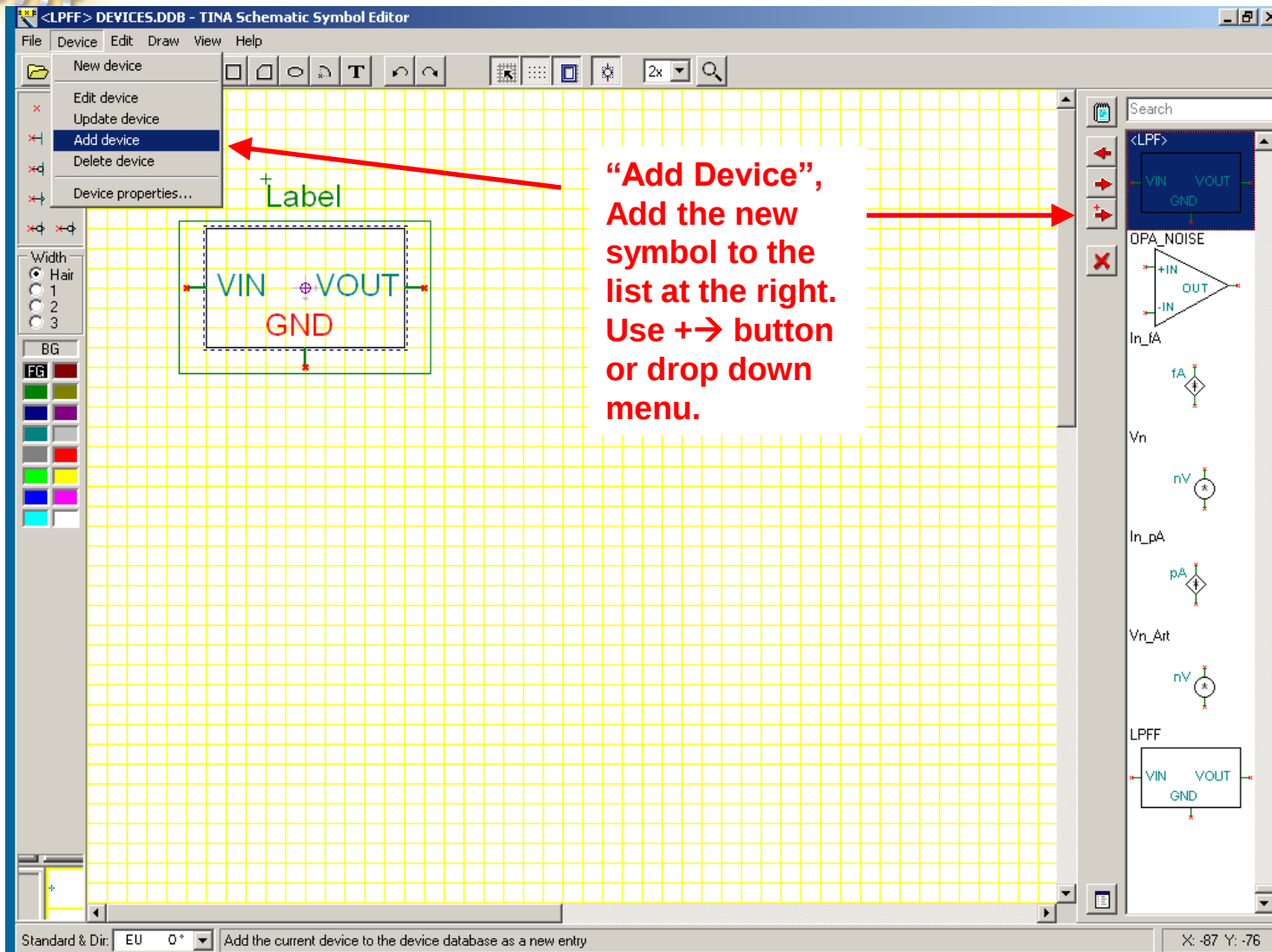
Symbol Creation - New



Give the edited symbol a new, unique name and click "OK"

Part 2: Schematic Macro-Model Creation

Symbol Creation - New



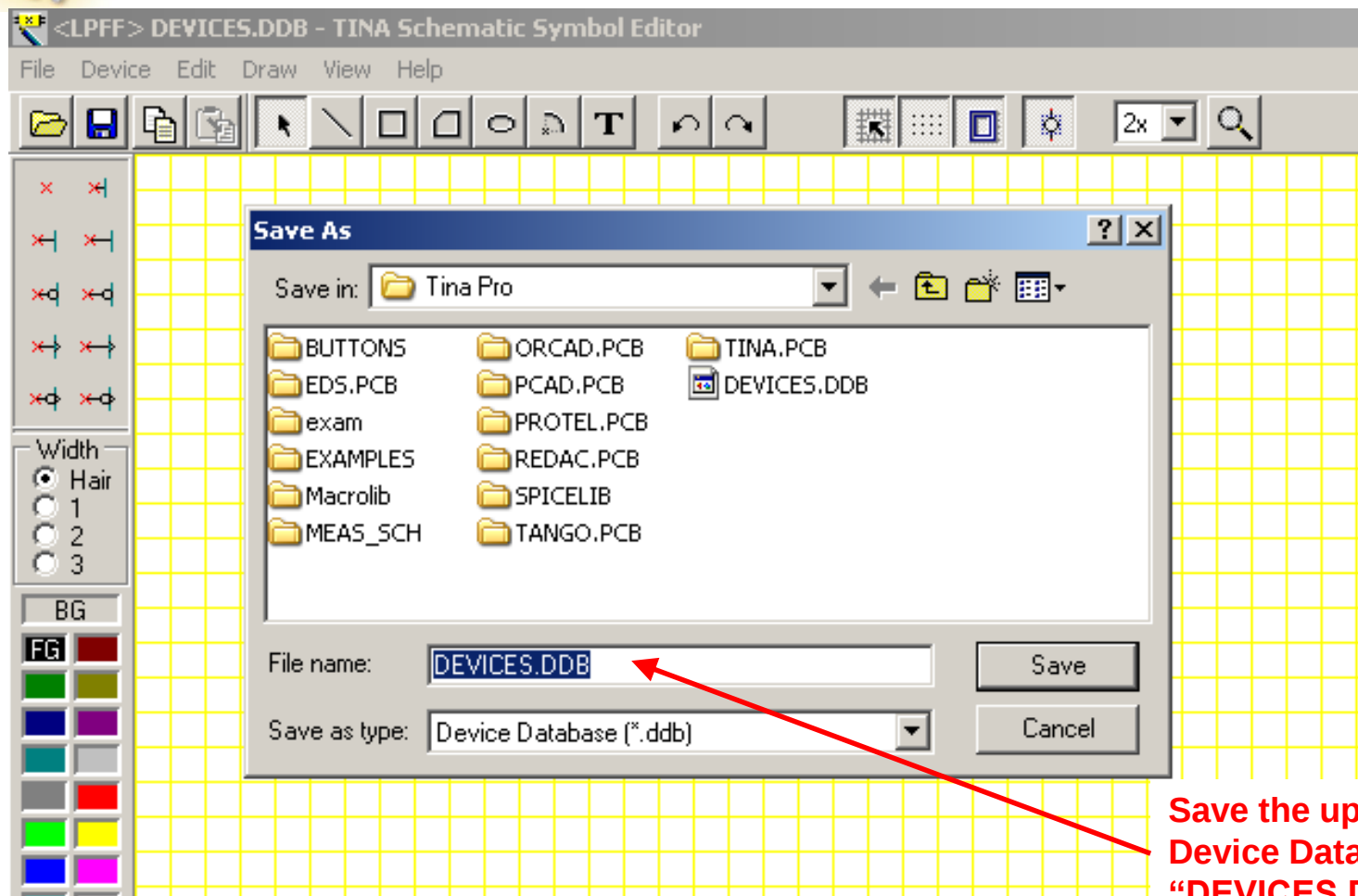
The screenshot shows the TINA Schematic Symbol Editor interface. The main workspace is a yellow grid. On the left, a menu is open with the following options: New device, Edit device, Update device, Add device (highlighted), Delete device, and Device properties... A red arrow points from the 'Add device' menu item to the central workspace. In the workspace, a schematic symbol for a device is being created, enclosed in a dashed green box. The symbol has three pins: 'VIN' on the left, 'VOUT' on the right, and 'GND' at the bottom. A green label 'Label' is positioned above the symbol. On the right side of the workspace, a vertical list of existing symbols is shown, including <LPFF>, OPA_NOISE, In_fA, Vn, nV, In_pA, pA, Vn_Art, nV, and LPFF. A red arrow points from the central workspace to the '<LPFF>' symbol in this list. A text box in the center of the workspace contains the following text: "Add Device", Add the new symbol to the list at the right. Use +→ button or drop down menu.

Standard & Dir: EU 0° Add the current device to the device database as a new entry

X: -87 Y: -76

Part 2: Schematic Macro-Model Creation

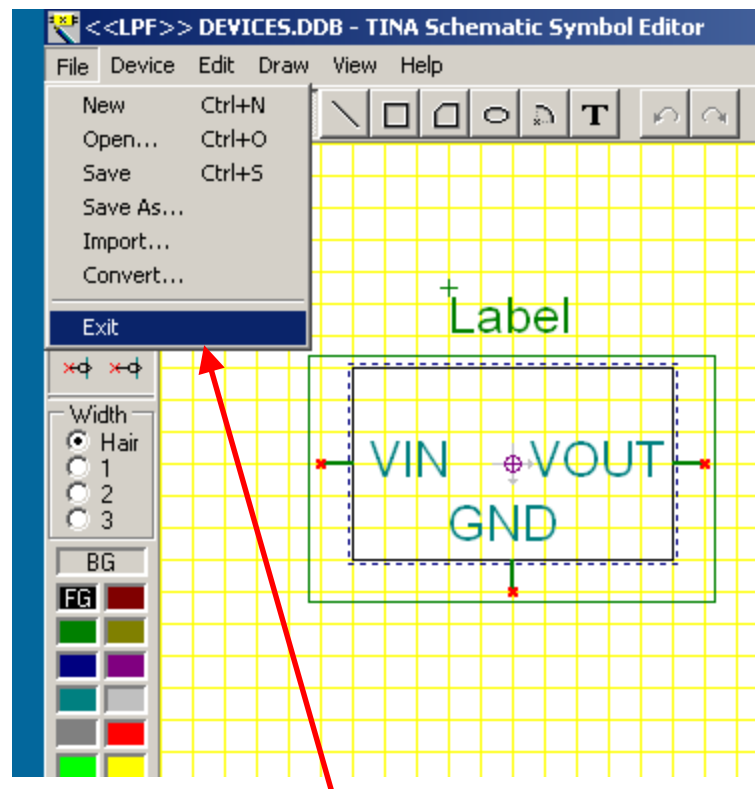
Symbol Creation - New



Warning: Backup the original "DEVICES.DDB" File in case there is a problem.

Part 2: Schematic Macro-Model Creation

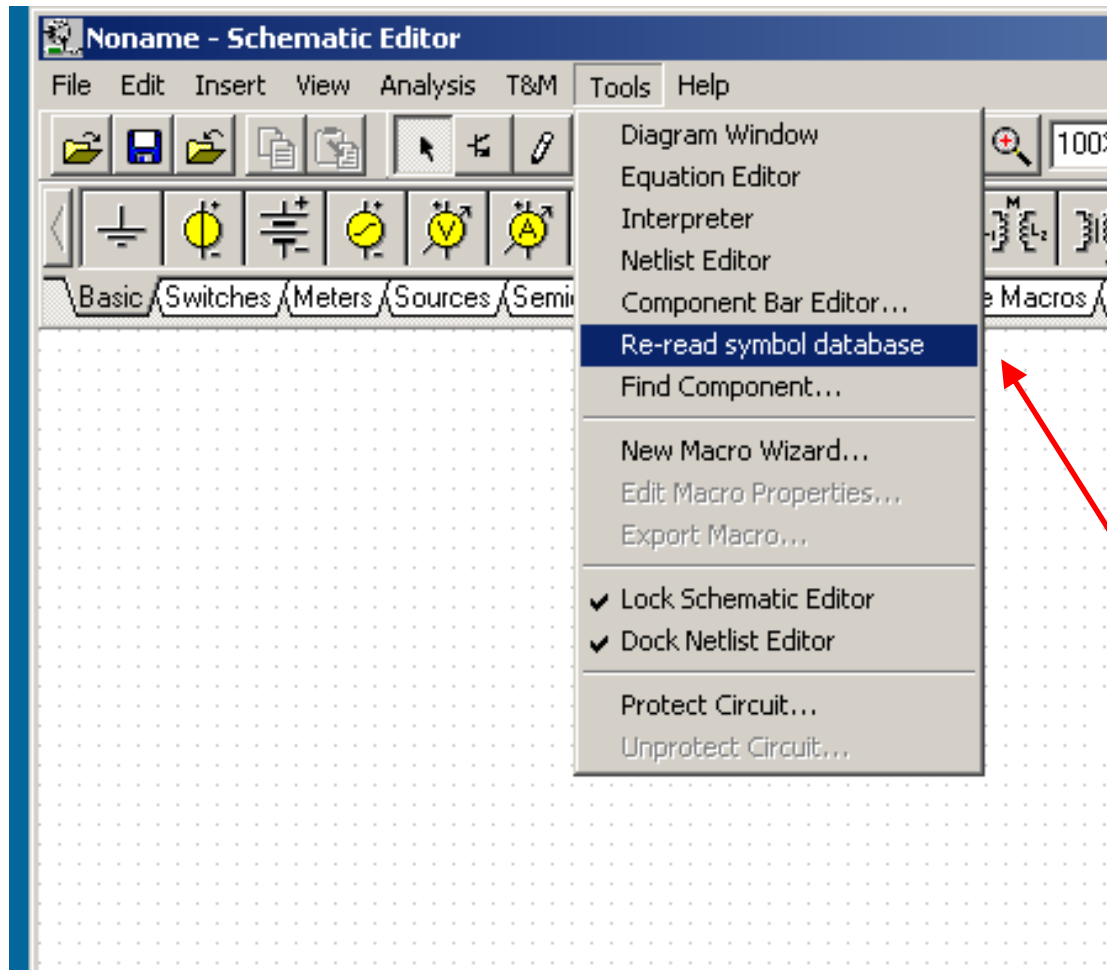
Macro-Model Creation



Either “Exit” or minimize the “Schematic Symbol Editor”

Part 2: Schematic Macro-Model Creation

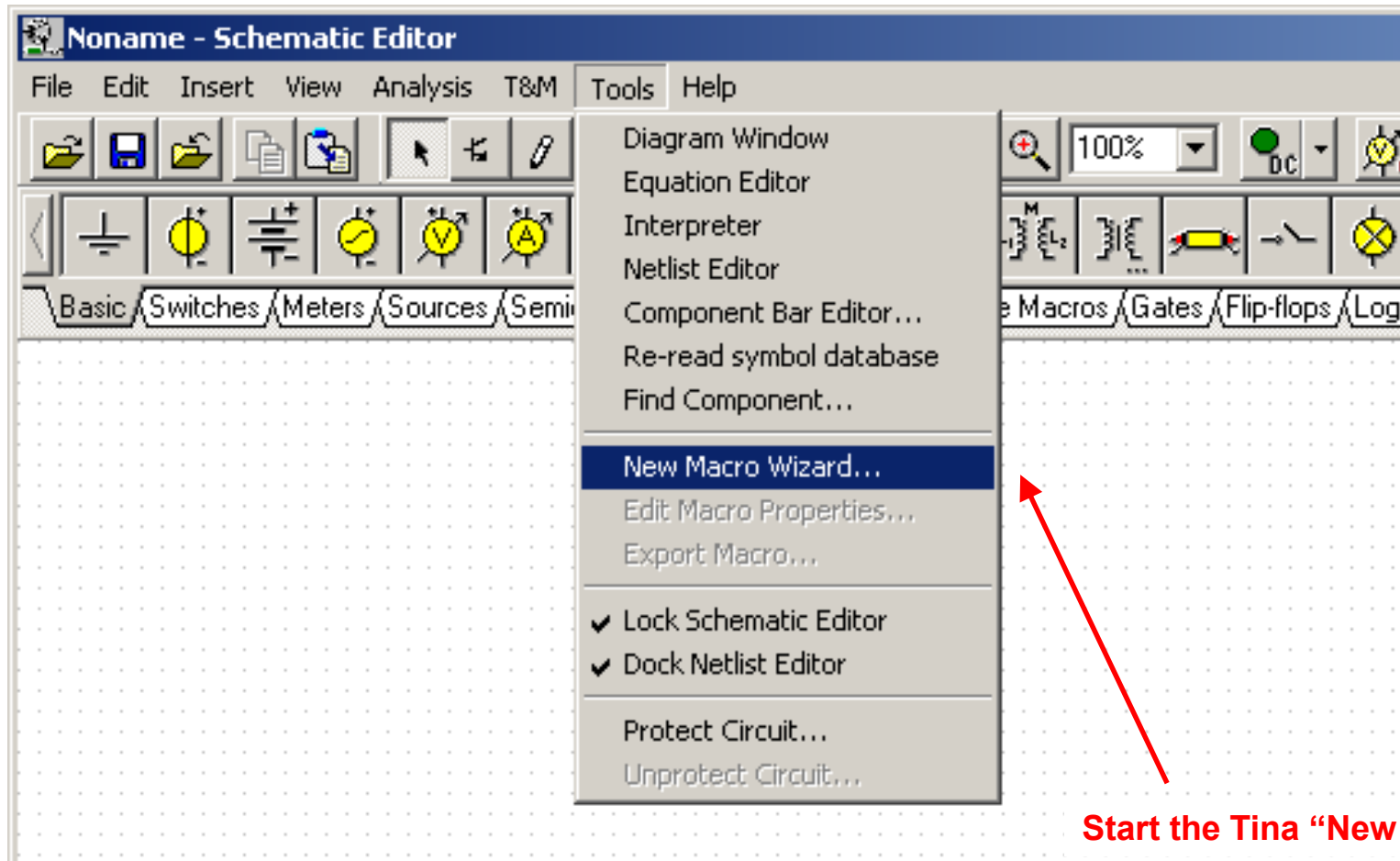
Macro-Model Creation



Select "Re-read symbol database"

Part 2: Schematic Macro-Model Creation

Macro-Model Creation



Start the Tina "New Macro Wizard"

Part 2: Schematic Macro-Model Creation

Macro-Model Creation

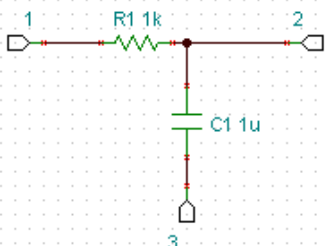
Low Pass - Schematic Editor

File Edit Insert View Analysis T&M Tools Help

100%

Basic Switches Meters Sources Semiconductors Optoelectronics Spice Macros Gates Flip-flops Logic ICs AD/DA-555 RF/An

Desired Macro-Model



New Macro Wizard

Name: LPF_SCH

Content: <Current>

Shape:

Defaults

Label:

Parameters:

☐ Empty circuit

☒ Current circuit

☐ Auto-generated

Vhdl

☐ Generate VHDL component

OK Cancel Help

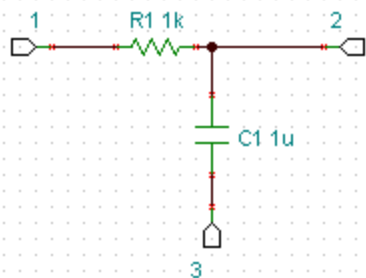
Open the schematic where the Schematic SubCircuit is created.

Enter a name for the Macro-Model under "Name".

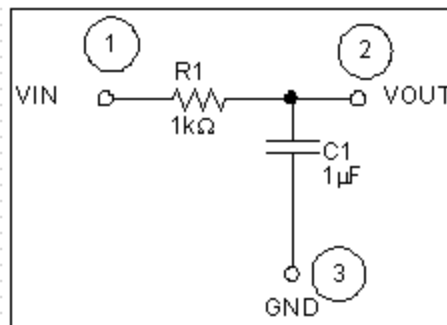
Check "Current circuit".

Part 2: Schematic Macro-Model Creation

Macro-Model Creation



Desired Macro-Model



New Macro Wizard

Name:

Content: ...

Shape: ...

Defaults

Label:

Parameters:

☐ Empty circuit

☒ Current circuit

☐ Auto-generated

Vhdl

☐ Generate VHDL component

OK Cancel Help

Macro Shapes

TL431

ShuntReg

<LPF>

VIN VOUT

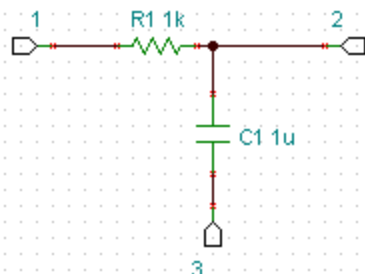
GND

OK Cancel

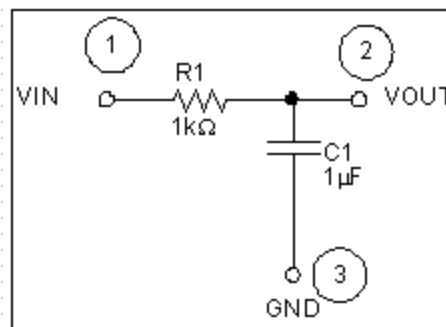
Click on
"Shape". The
ONLY symbol
choices that
will appear at
the right are
ones that
contain pin
numbers and
quantities that
EXACTLY
match the Text
SubCircuit.
(i.e., 3 pins and
pins numbered
1, 2, 3
corresponding
to the Nodes in
the Schematic
Subcircuit of
1, 2, 3)

Part 2: Schematic Macro-Model Creation

Macro-Model Creation



Desired Macro-Model



New Macro Wizard

Name:

Content:

Shape:

Defaults

Label:

Parameters:

☐ Empty circuit

☒ Current circuit

☐ Auto-generated

Vhdl

☐ Generate VHDL component

OK Cancel Help

Macro Shapes

TL431

ShuntReg

<LPF>

VIN VOUT

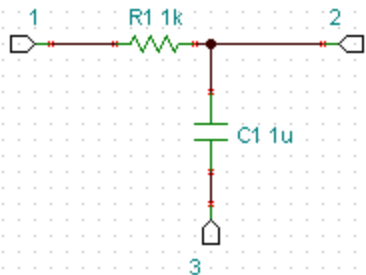
GND

OK Cancel

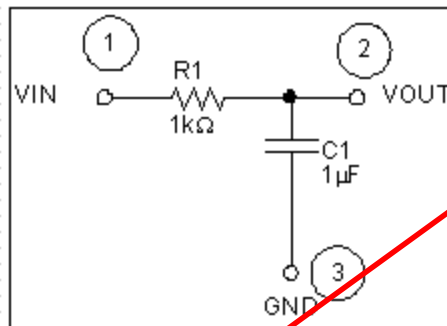
Highlight desired "Shape" at the right and click "OK" to enter it into the "Shape" dialog box.

Part 2: Schematic Macro-Model Creation

Macro-Model Creation



Desired Macro-Model



If you want a label to appear next to the Ref Designator when you place the Macro-Model on a schematic enter it here under “Label”

New Macro Wizard

Name:

Content: ☐ Empty circuit ☒ Current circuit

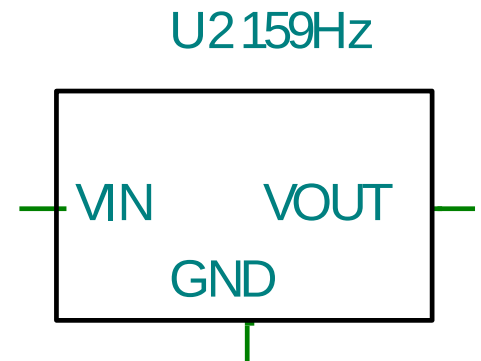
Shape: ☐ Auto-generated

Defaults

Label: ☐ Generate VHDL component

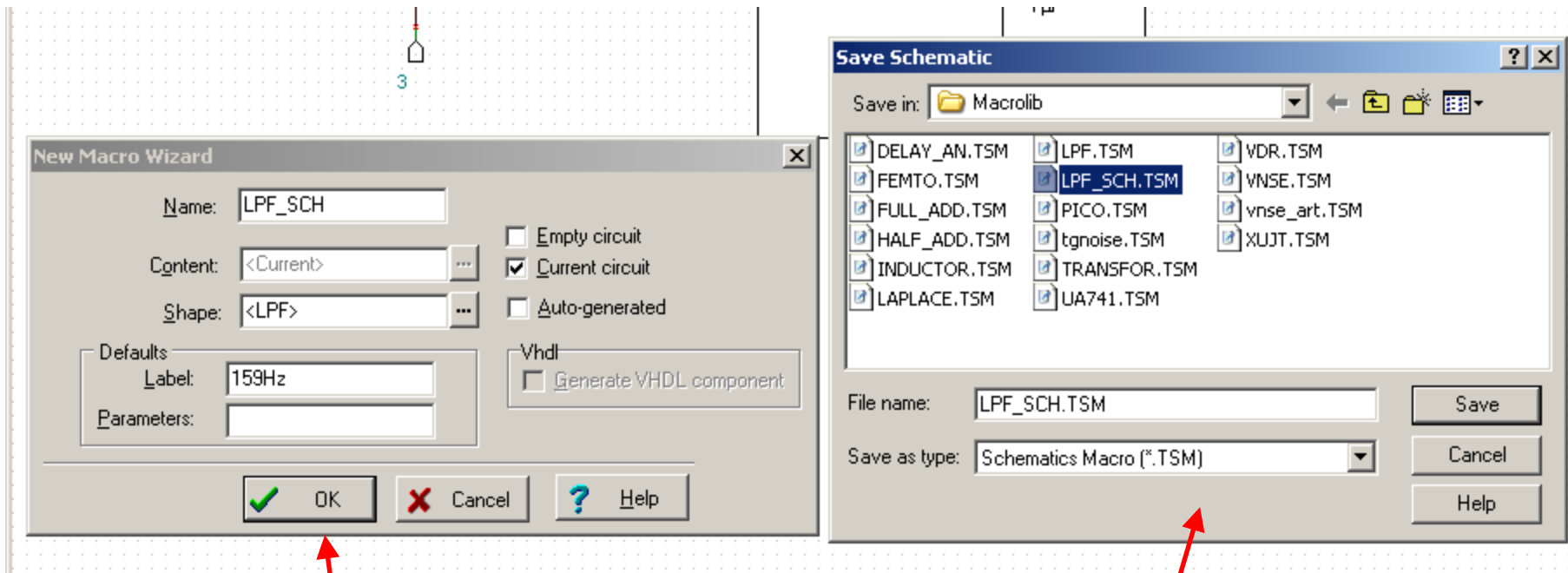
Parameters:

☒ OK ☐ Cancel



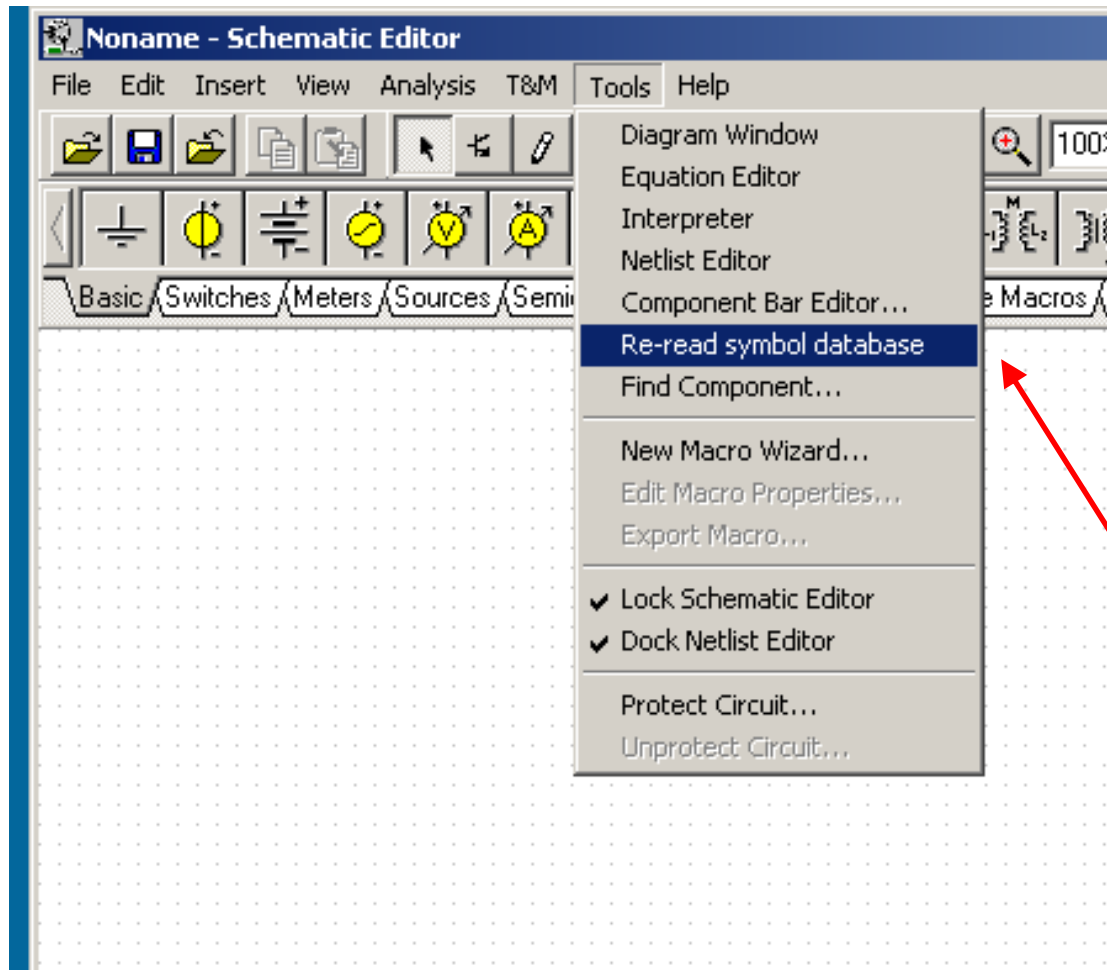
Part 2: Schematic Macro-Model Creation

Macro-Model Creation



Finish Macro-Model creation and Save the results. Click "OK" and popup window on right will appear. Tina uses ".TSM" as Macro Schematics file extension. After creating or using default File name and choosing the folder to store the Macro-Model in click "Save"

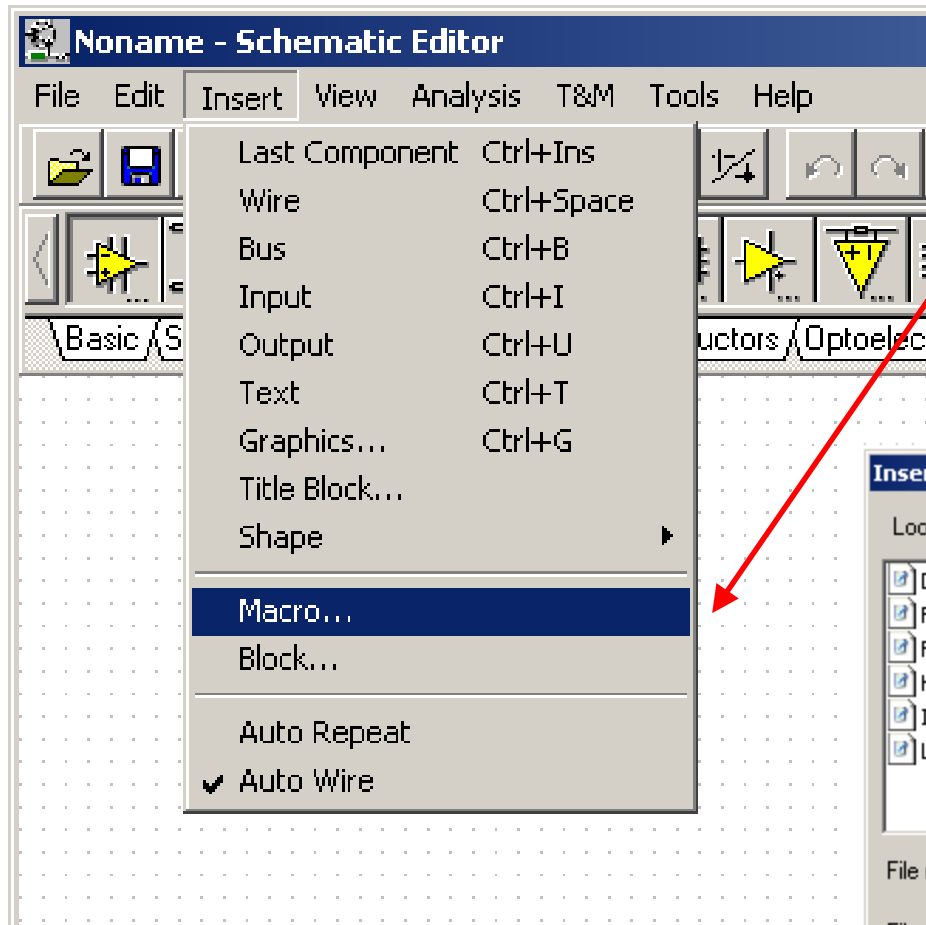
Part 2: Schematic Macro-Model Creation Macro-Model Use



Select "Re-read symbol database"

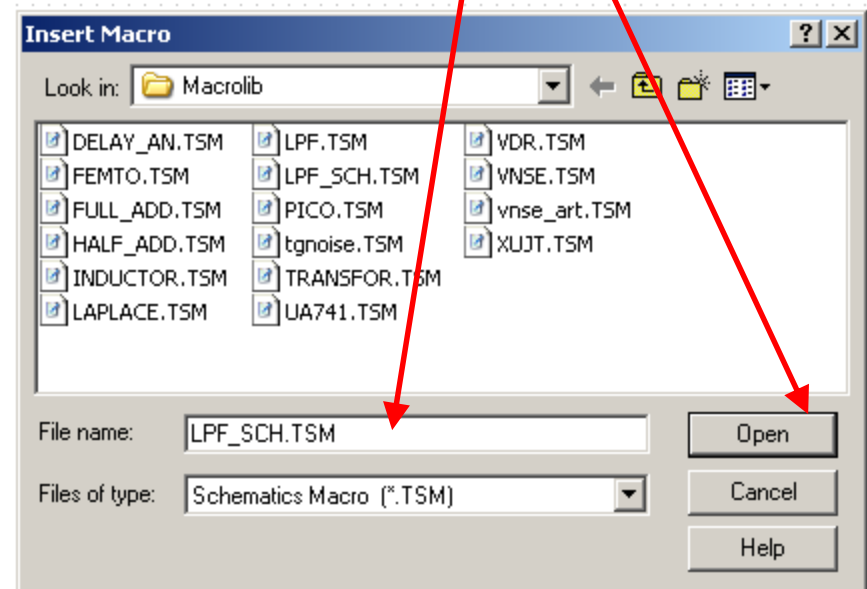
Part 2: Schematic Macro-Model Creation

Macro-Model Use



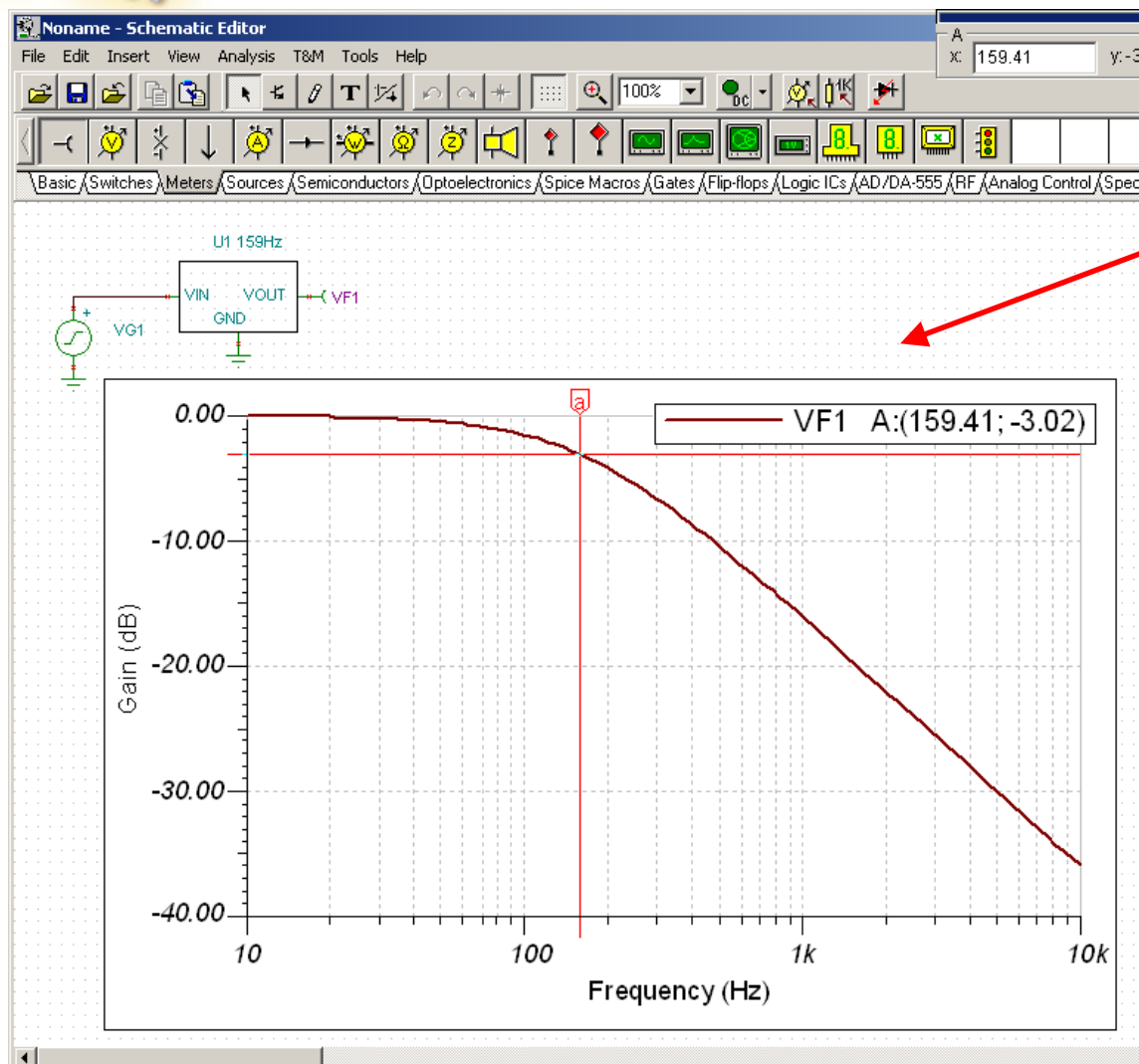
In the Tina “Schematic Editor” choose “Insert”, “Macro”.

From the desired folder select the Macro for insertion into the Schematic Editor and click “Open”



Part 2: Schematic Macro-Model Creation

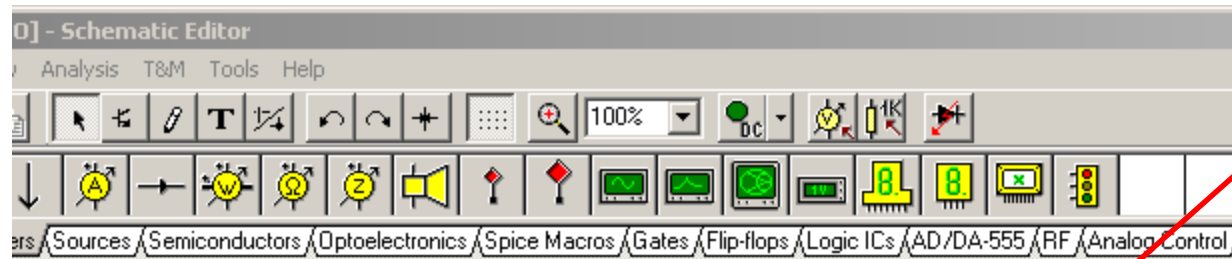
Macro-Model Use



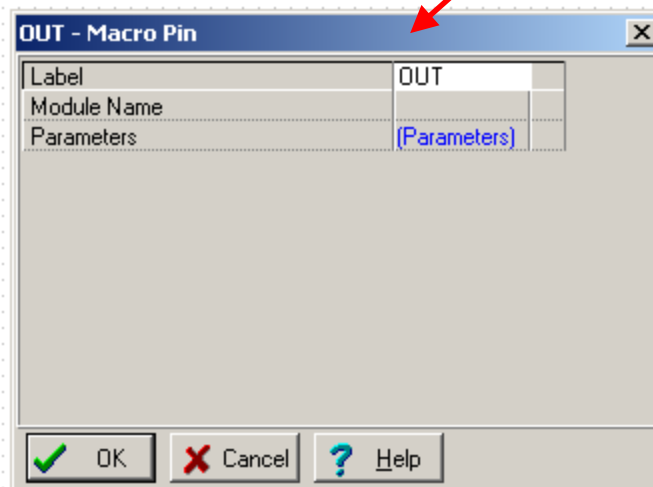
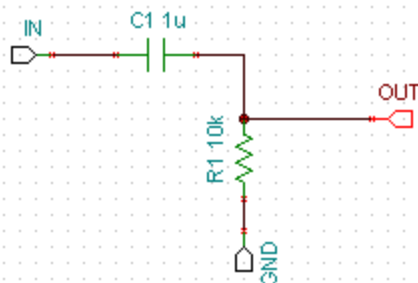
Now the Macro-Model may be connected and used in the Schematic Editor as a normal Macro-Model.

Part 2: Schematic Macro-Model Creation

Auto-Generated Shape

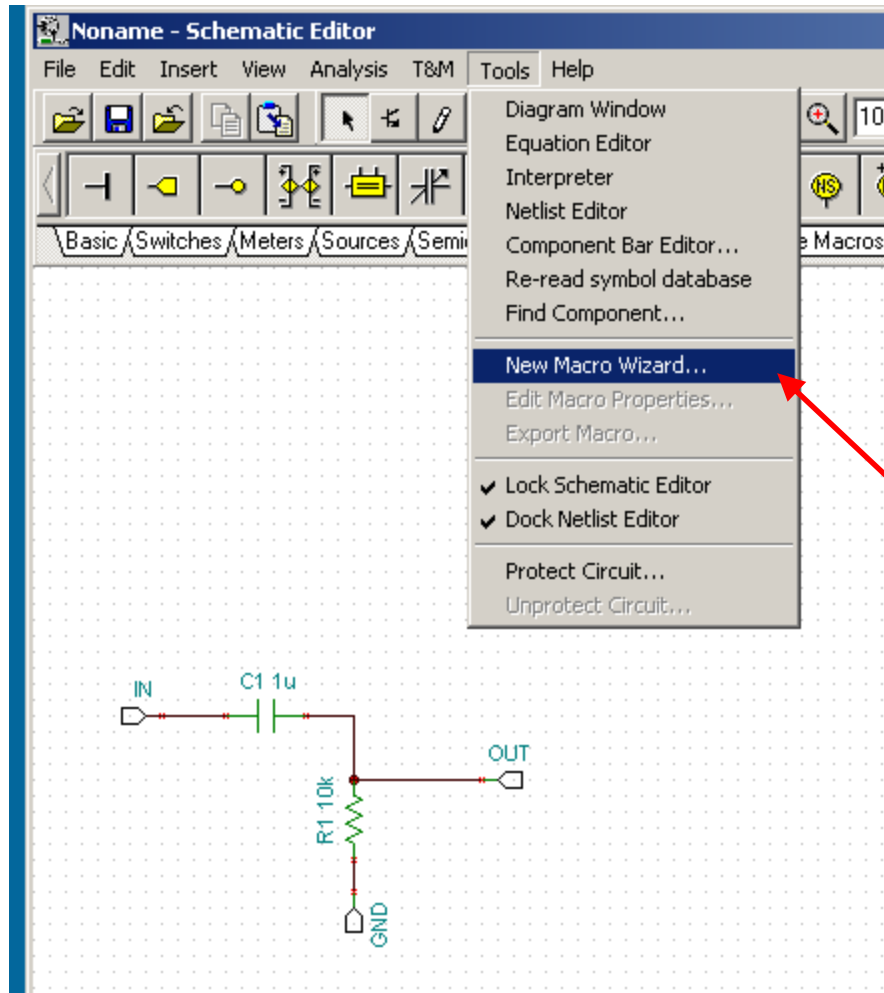


Create subcircuit in the Schematic Editor. Connect a "Macro Pin" to each I/O of the subcircuit.



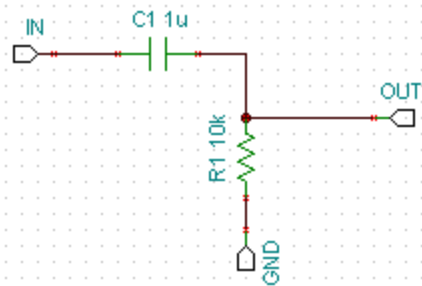
Double-Click on each "Macro Pin" and assign it a desired name in the "Label" dialog box. Click "OK" to save the pin name.

Part 2: Schematic Macro-Model Creation Auto-Generated Shape



Start the Tina “New Macro Wizard”

Part 2: Schematic Macro-Model Creation Auto-Generated Shape



New Macro Wizard

Name:

Content:

Shape:

Defaults

Label:

Parameters:

☐ Empty circuit

☒ Current circuit

☒ Auto-generated

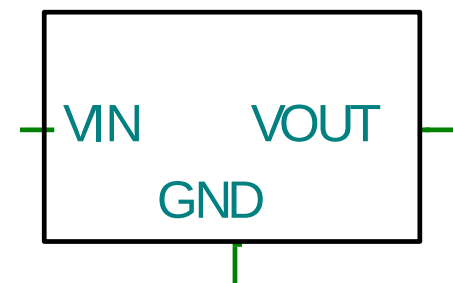
Vhdl

☐ Generate VHDL component

☒ OK ☐ Cancel ☐ Help

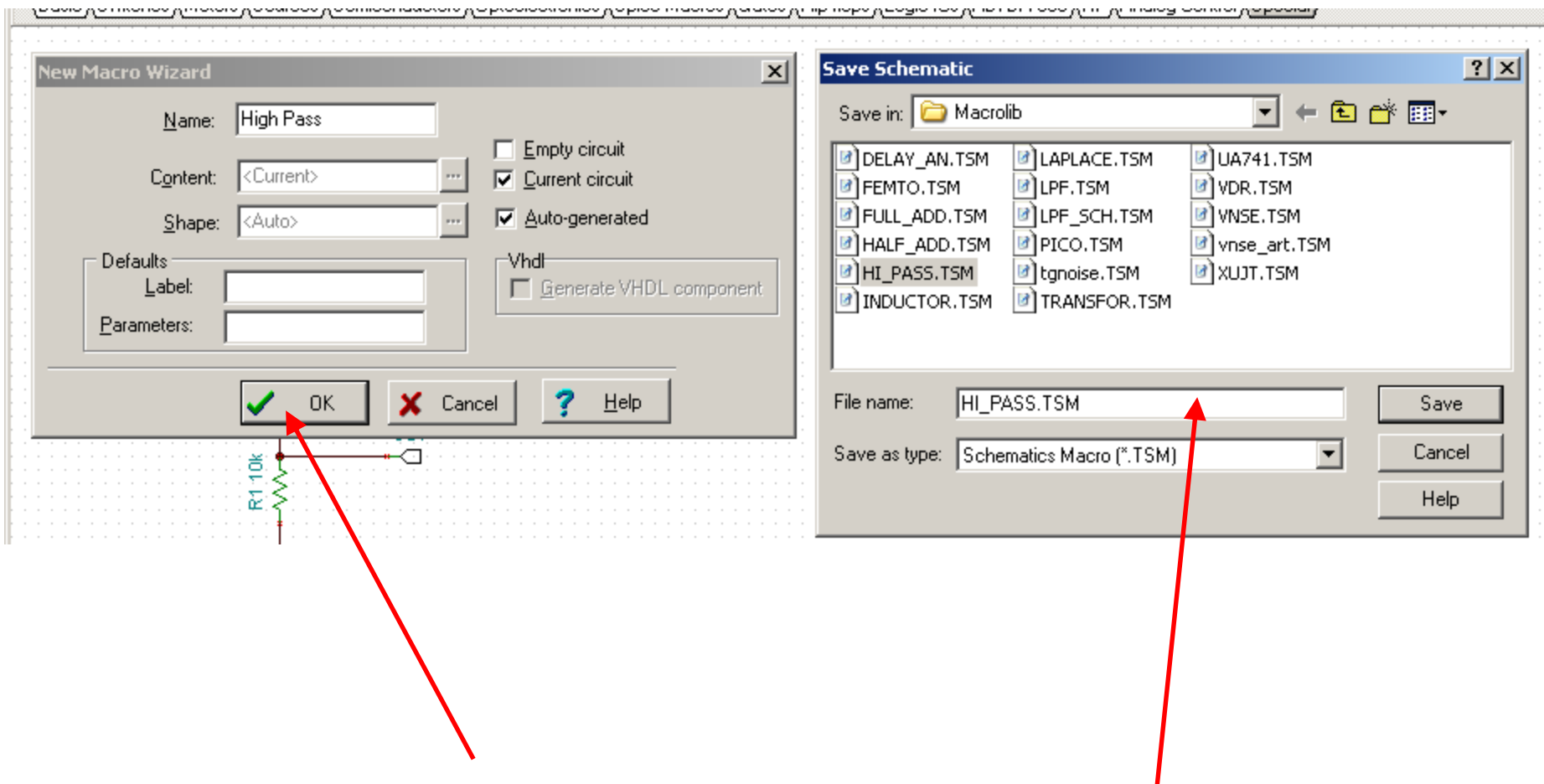
If you want a label to appear next to the Ref Designator when you place the Macro-Model on a schematic enter it here under "Label"

U2 159Hz



Check the box next to the "Content" dialog as "Current circuit" since we are turning the current schematic into a Macro-Model. Check the box next to the "Shape" dialog as "Auto-generated" to have Tina create a Macro-Model symbol automatically for you. Then click "OK".

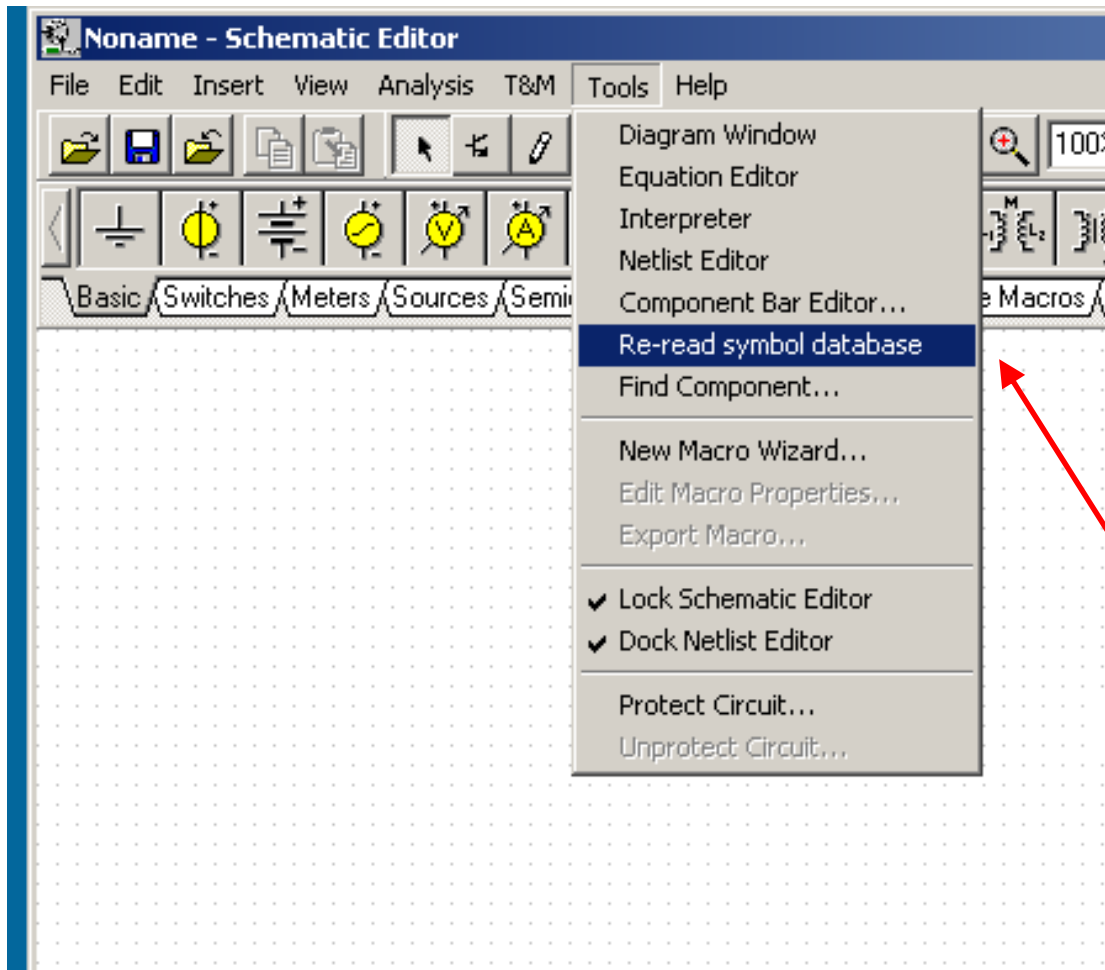
Part 2: Schematic Macro-Model Creation Auto-Generated Shape



Finish Macro-Model creation and Save the results. Click “OK” and popup window on right will appear. Tina uses “.TSM” as Macro Schematics file extension. After creating or using default File name and choosing the folder to store the Macro-Model in click “Save”

Part 2: Schematic Macro-Model Creation

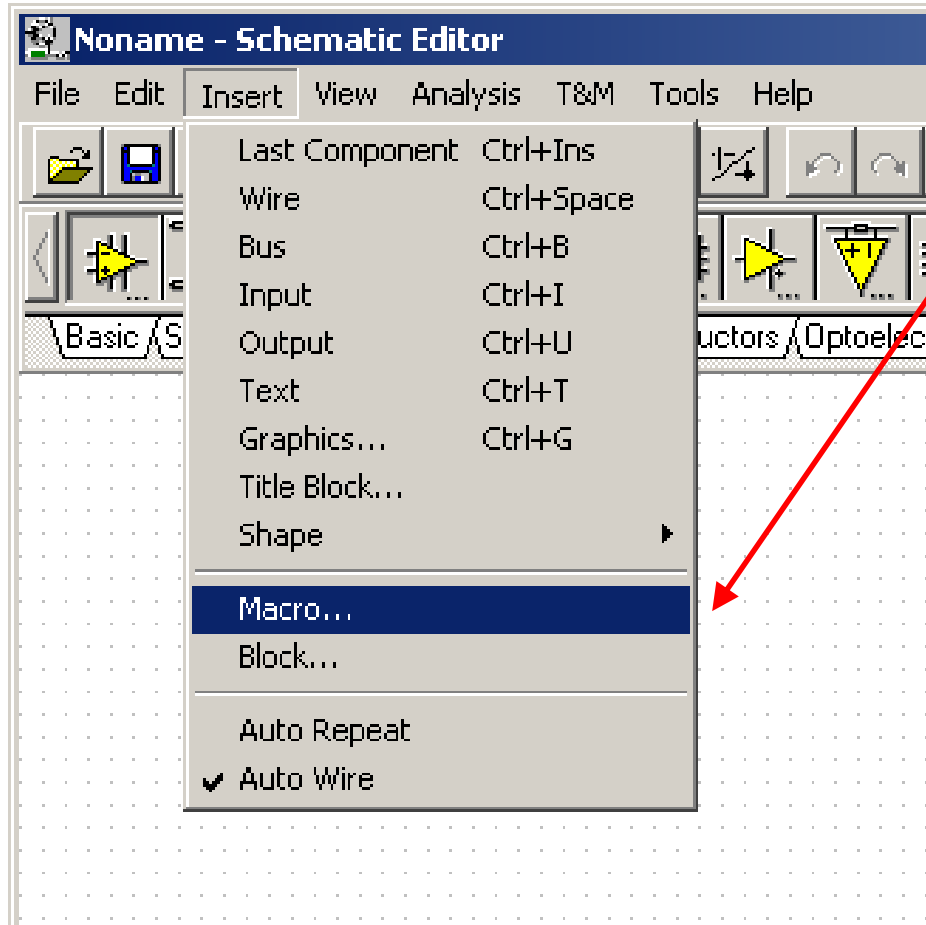
Auto-Generated Shape: Macro-Model Use



Select "Re-read symbol database"

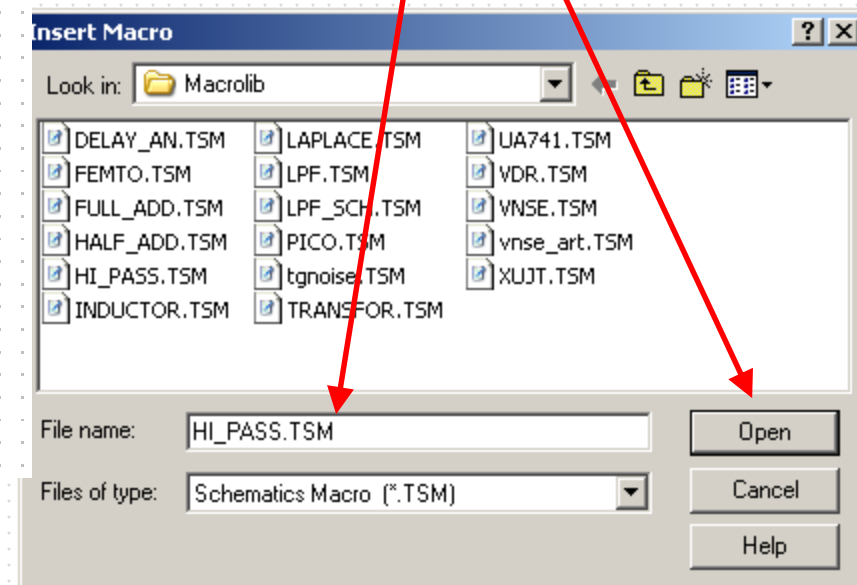
Part 2: Schematic Macro-Model Creation

Auto-Generated Shape: Macro-Model Use



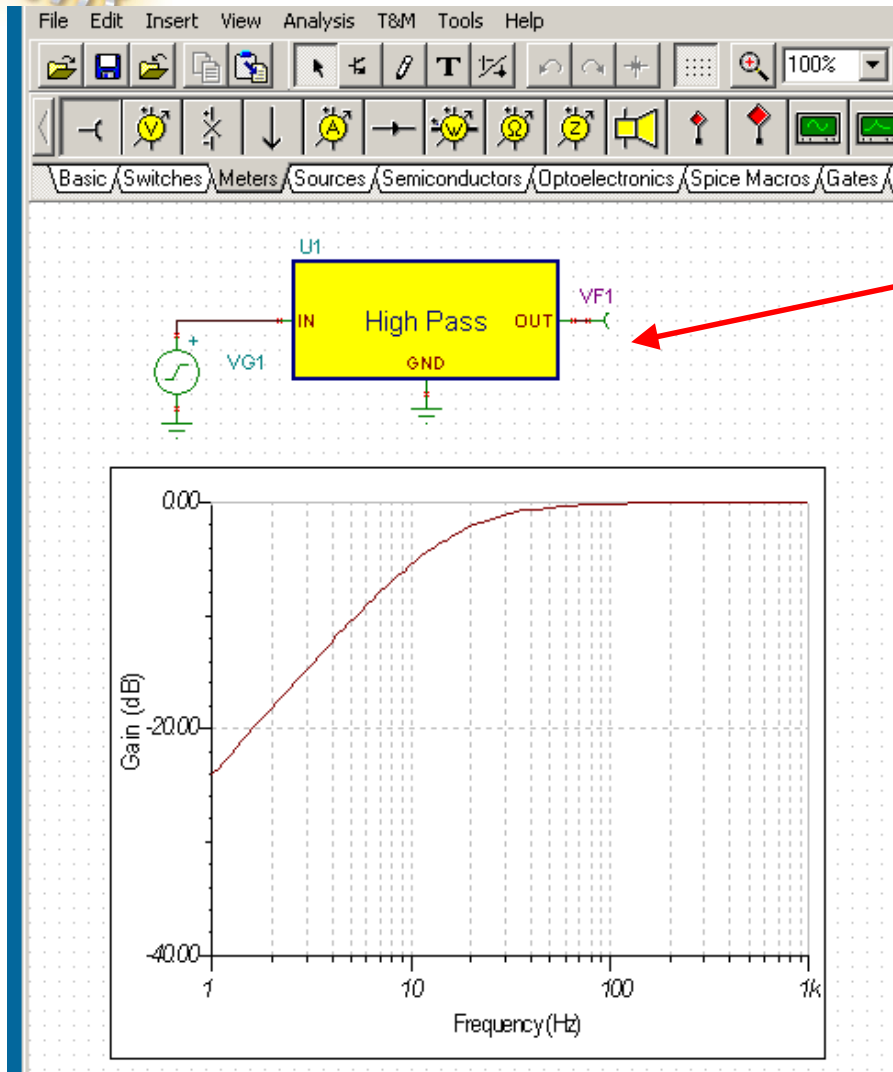
In the Tina "Schematic Editor" choose "Insert", "Macro".

From the desired folder select the Macro for insertion into the Schematic Editor and click "Open"



Part 2: Schematic Macro-Model Creation

Auto-Generated Shape: Macro-Model Use

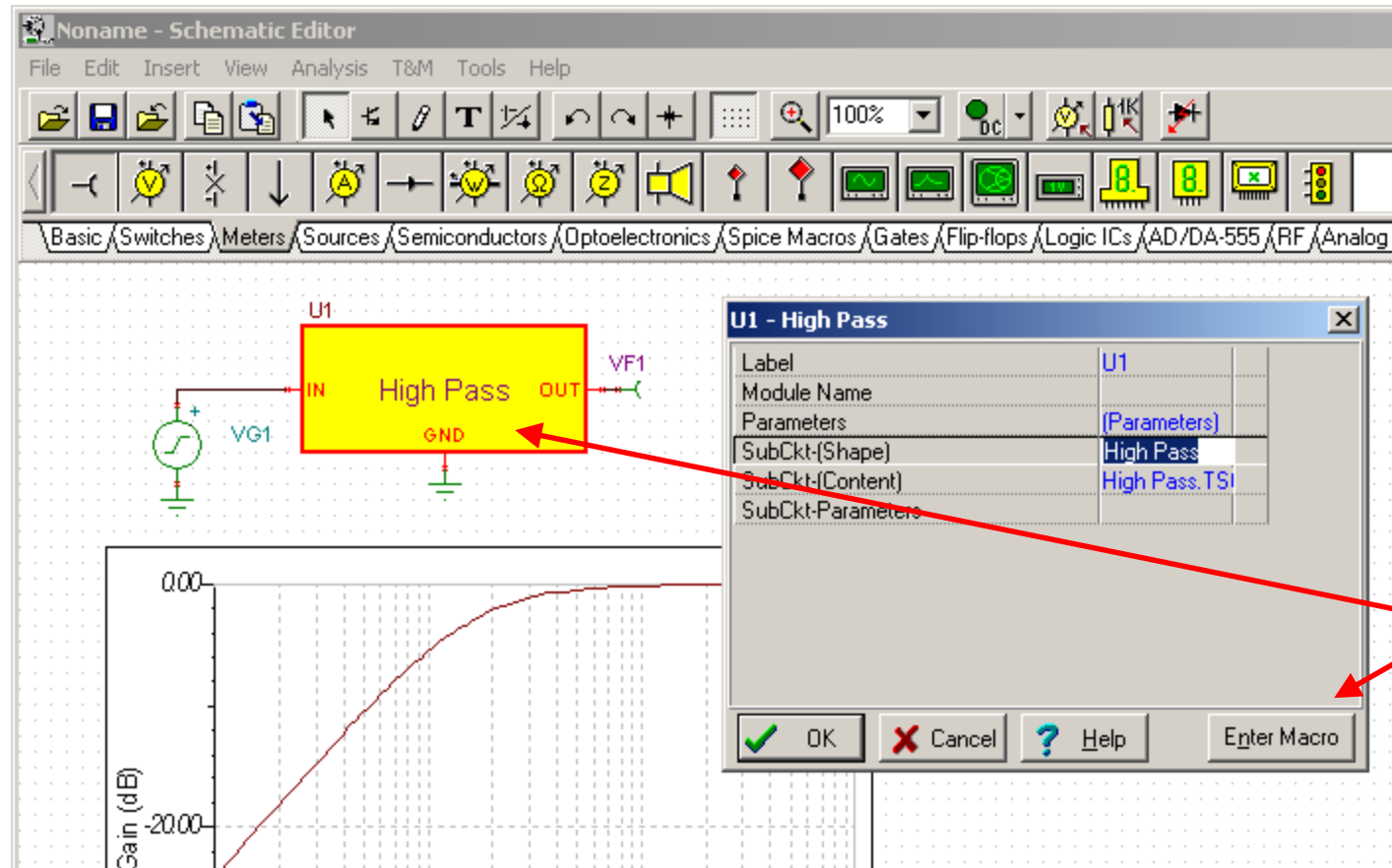


Now the Macro-Model may be connected and used in the Schematic Editor as a normal Macro-Model.

Notice how Tina created the box shape for the Macro-Model and automatically assigned the pins the names we used for each "Macro Pin" name when we created the Schematic SubCircuit

Part 2: Schematic Macro-Model Creation

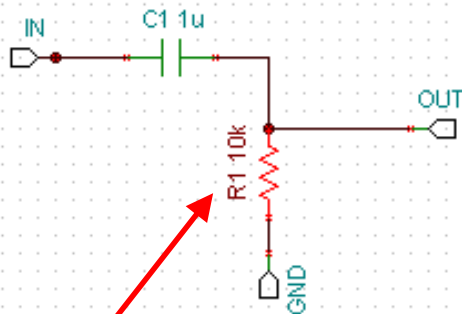
Auto-Generated Shape: Macro-Model Use



Double-click on the Macro-Model Shape and the popup Properties window will appear. Click on "Enter Macro"

Part 2: Schematic Macro-Model Creation

Auto-Generated Shape: Macro-Model Use



R1 - Resistor

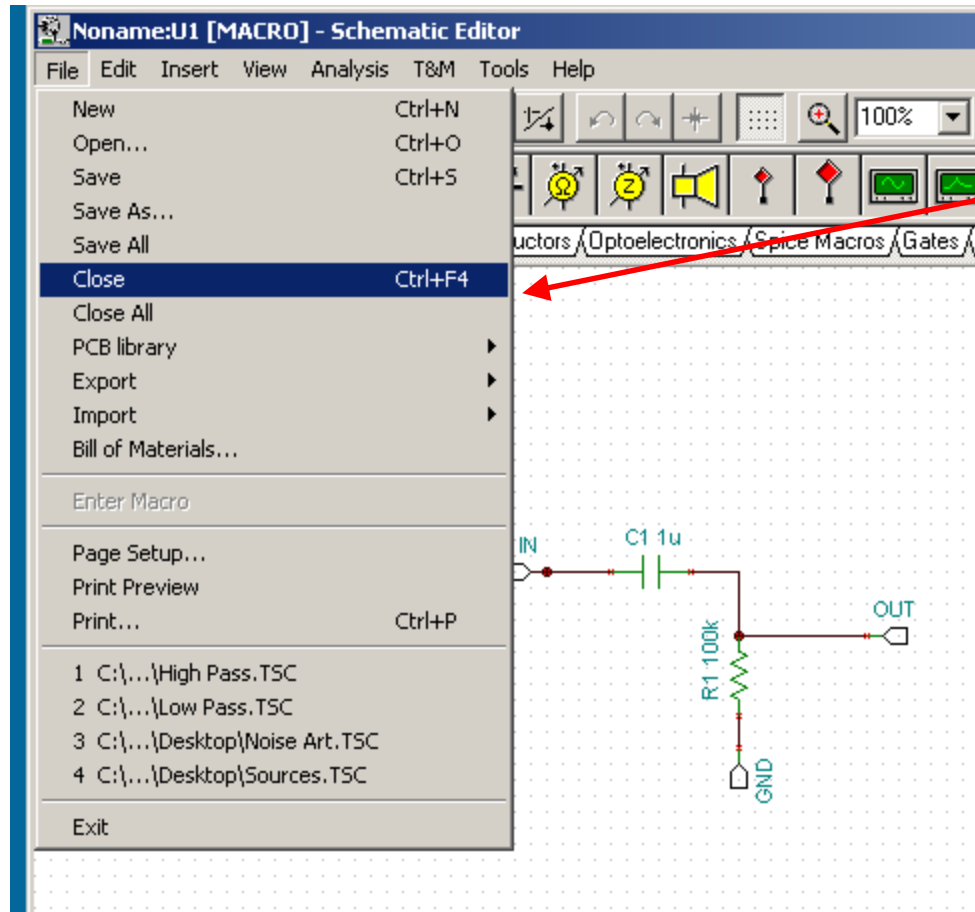
Label	R1
Module Name	
Parameters	(Parameters)
Resistance [Ohm]	100k
Power [W]	1
Temperature	Relative
Temperature [C]	0
Linear temp. coef. [1/C]	0
Quadratic temp. coef. [1/C ²]	0
Exponential temp. coef. [%/C]	0
Maximum voltage [V]	100
Fault	None

OK Cancel Help

The Schematic SubCircuit that was used to create the Macro-Model will appear in a separate window. Values in the SubCircuit may be modified as in the normal Schematic Editor.

Part 2: Schematic Macro-Model Creation

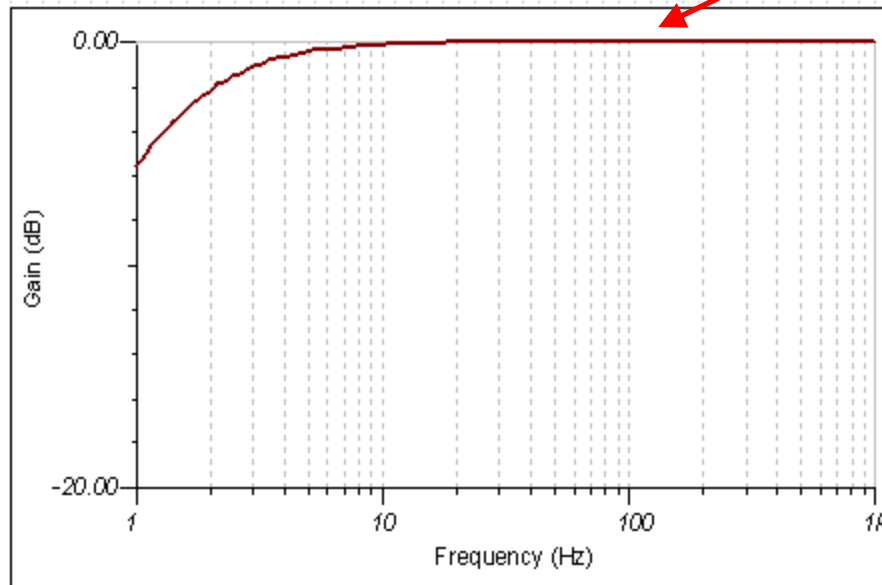
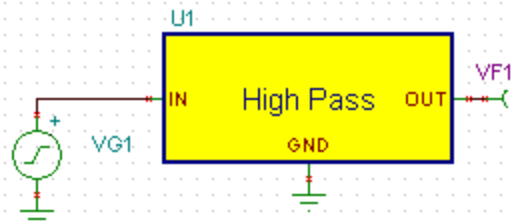
Auto-Generated Shape: Macro-Model Use



When the edits to the component values inside of the Schematic SubCircuit are completed close the window by selecting "File", "Close".

Part 2: Schematic Macro-Model Creation

Auto-Generated Shape: Macro-Model Use



The modified Schematic SubCircuit used for the Macro-Model was modified and the changes are now included in the Macro-Model. These changes stay wherever this instance of the Macro-Model are copied and placed in this or another schematic.