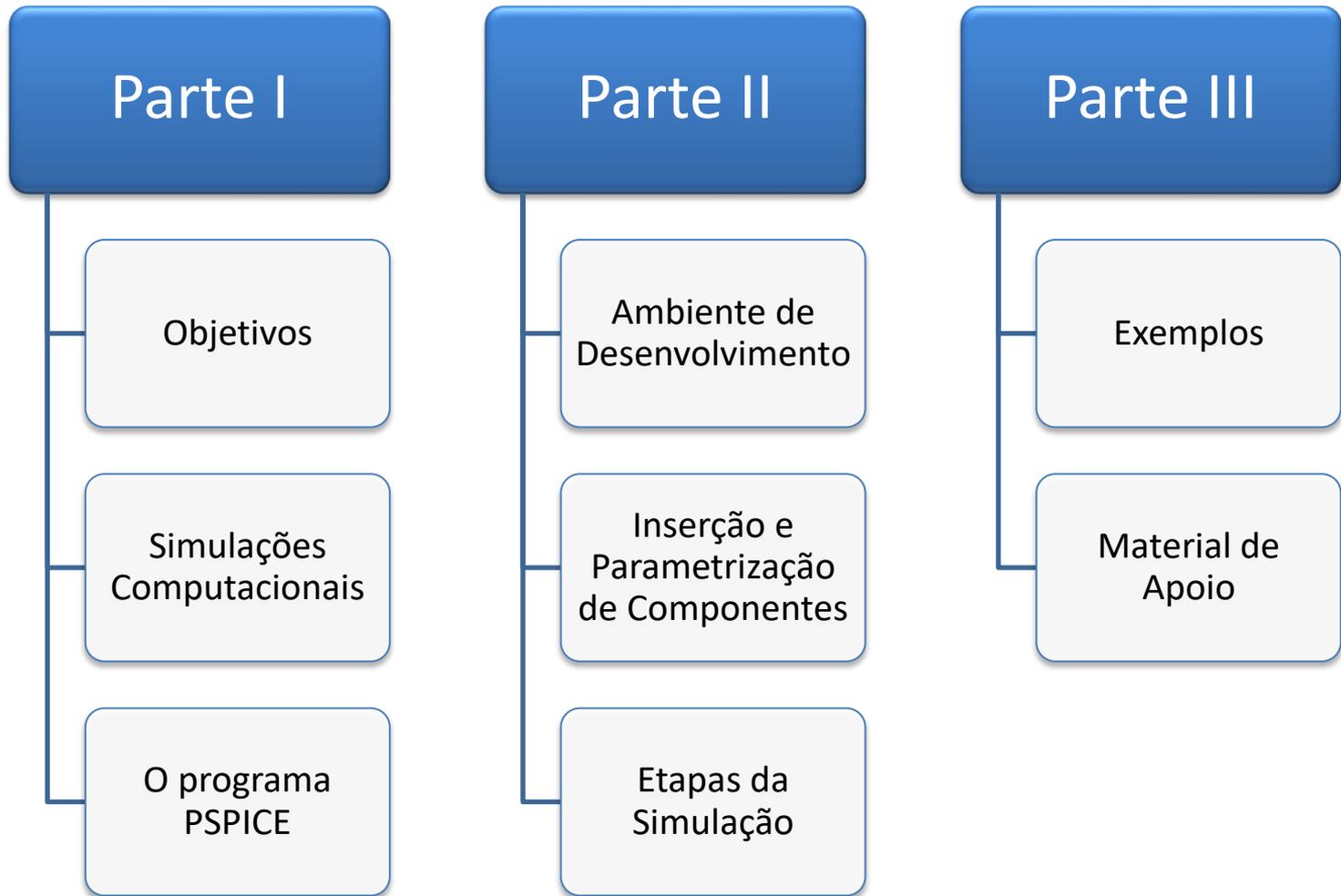


Introdução à Análise de Circuitos Elétricos Utilizando “PSPICE”

Disciplinas: *SEL0301 – Circuitos Elétricos I e SEL0602 – Circuitos Elétricos*
Docente: *Prof. Dr. Mário Oleskovicz*
Palestrante: *Moisés Junior Batista Borges Davi*

Organização da Apresentação



(Parte I) Objetivos



Familiarização com a ferramenta computacional para ampliação da capacidade de análise de circuitos (PSPICE)



Adquirir conhecimento mínimo necessário para solucionar os exercícios a serem repassados

- Serão 03 exercícios (solução analítica e por meio de simulação computacional)
- Nota total (1,5 ponto)

(Parte I) Simulações Computacionais

- ✓ Buscam representar matematicamente o mundo real (físico) por meio de modelos computacionais:



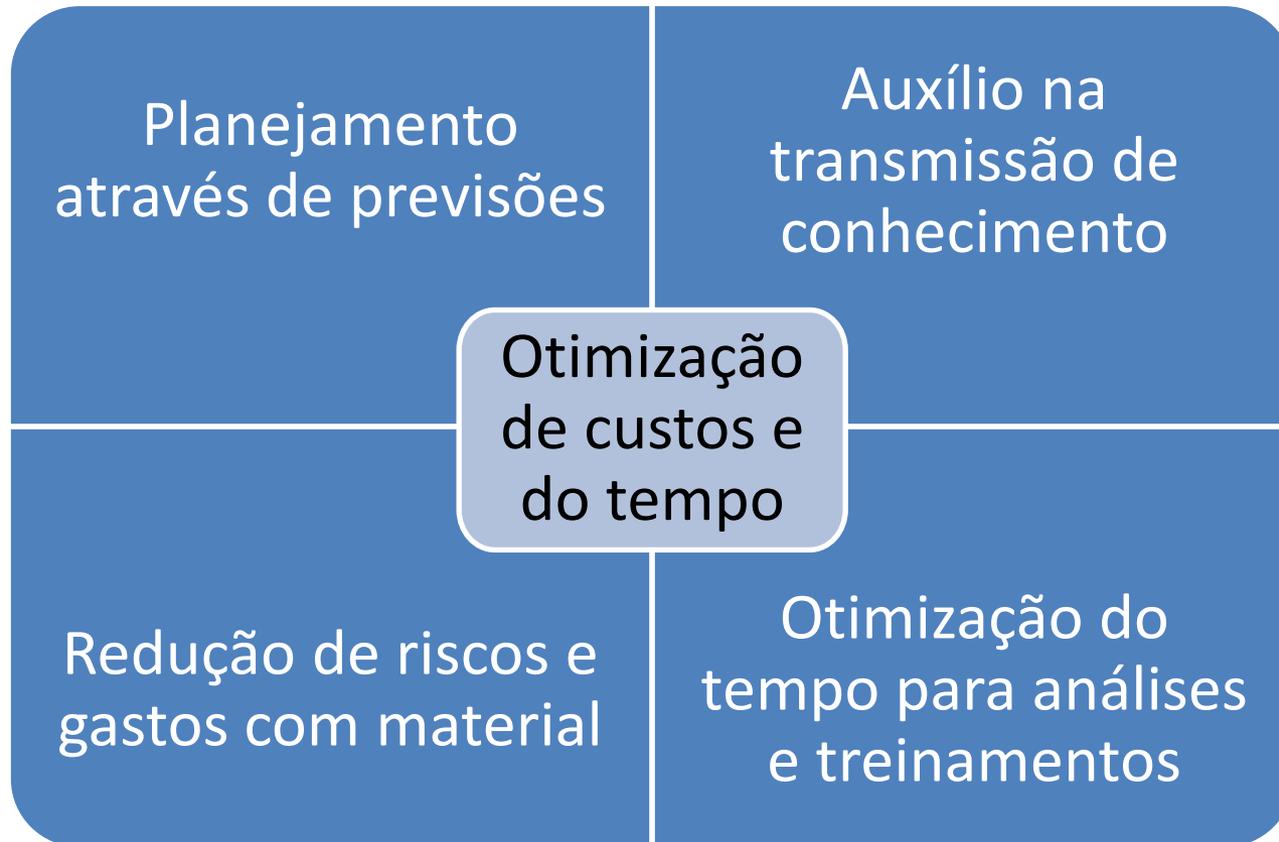
Previsões do clima

Previsões de crescimento econômico de empresas

Simulação de voo para treinamento de pilotos de avião

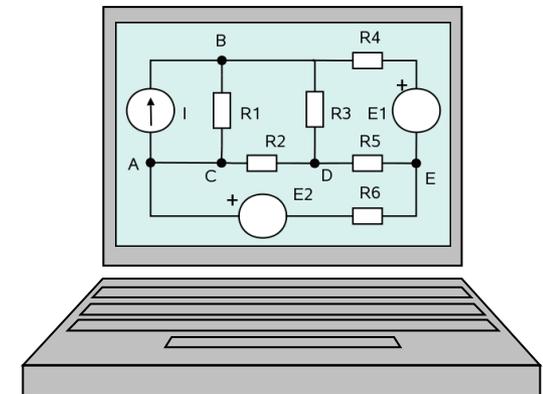
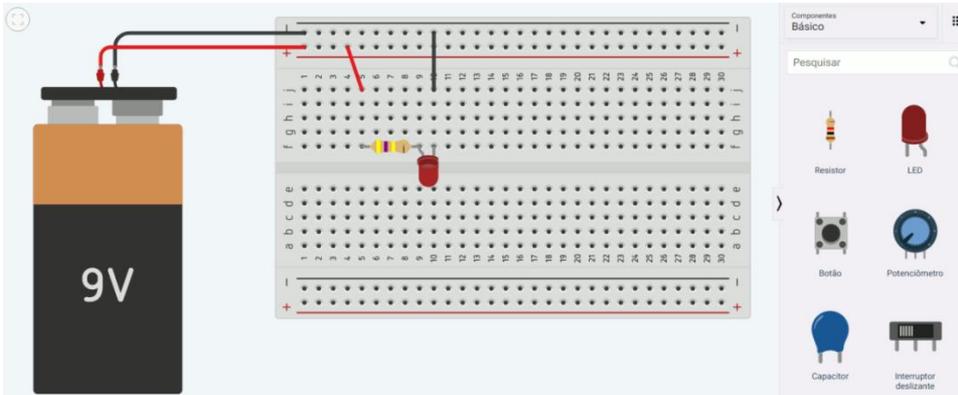
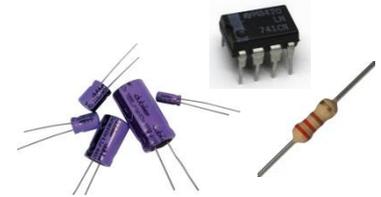
(Parte I) Simulações Computacionais

- ✓ Buscam representar matematicamente o mundo real (físico) por meio de modelos computacionais:



(Parte I) Simulações Computacionais

- ✓ Buscam representar matematicamente circuitos ou sistemas elétricos:
 - ✓ Menor tempo necessário para análises de circuitos complexos;
 - ✓ Possibilidade de experimentos:
 - Trocar valores dos componentes;
 - Alterar configurações e topologias do circuito;
 - Monitorar/prever o comportamento das grandezas elétricas nos testes.



(Parte I) O Programa PSPICE

- ✓ SPICE
 - *Simulation Program with Integrated Circuit Emphasis*
- ✓ Prof. Donald O. Pederson;
 - SPICE 1 (1972), 2 (1975) e 3 (1989)
- ✓ Dentre as várias versões para PC que foram disponibilizadas, a mais conhecida e que será utilizada no curso, é o PSPICE (Personal SPICE);
- ✓ Ferramenta estável;
- ✓ Farta documentação e exemplos;

Berkeley
UNIVERSITY OF CALIFORNIA



(Parte I) O Programa PSPICE

Fontes DC

- Ponto de operação (*V dos nós e I das fontes*)
- Análise de transitórios em sistemas com fonte DC

Fontes AC

- Análise de circuitos AC: levantamento de fasores de tensão e corrente

(Parte II) Ambiente de Desenvolvimento

- ✓ PSPICE (**P**ersonal **Spice**)

- ✓ PSPICE 9.1 Student Version
 - Algumas limitações:
 - Quantidade de componentes;
 - Biblioteca.

 - Disponível no portal edisciplinas;

 - Para modelagem e análise dos circuitos será utilizado o Pspice Schematics.

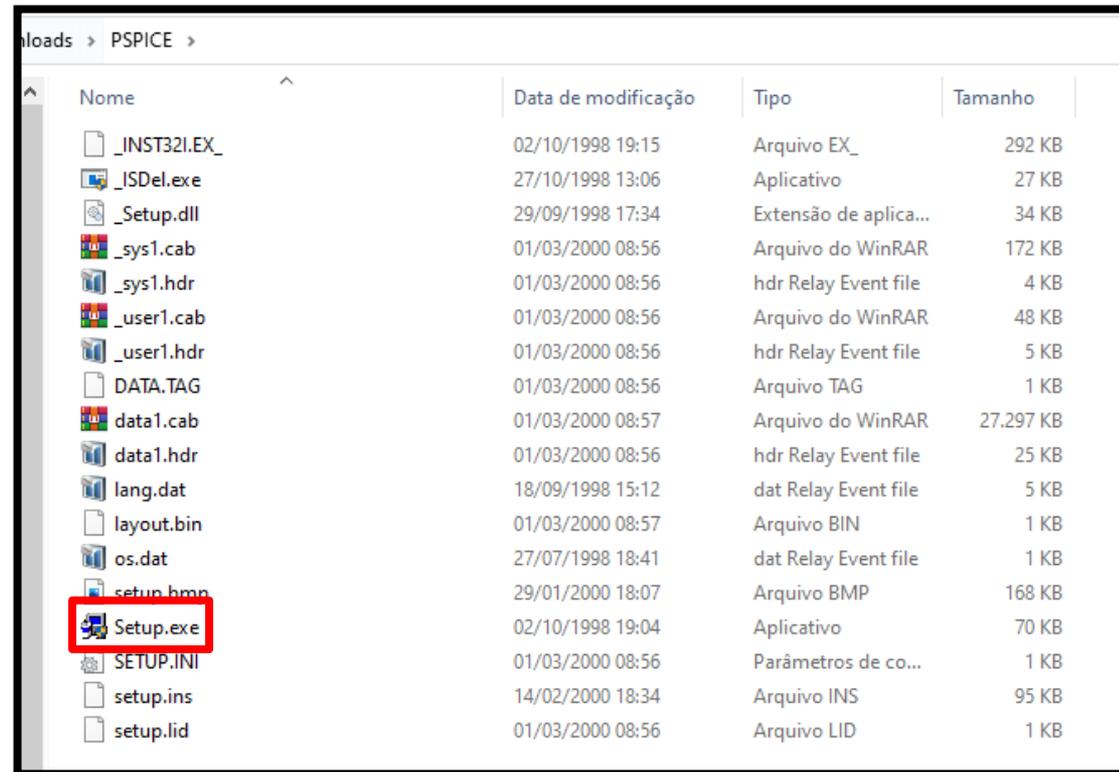
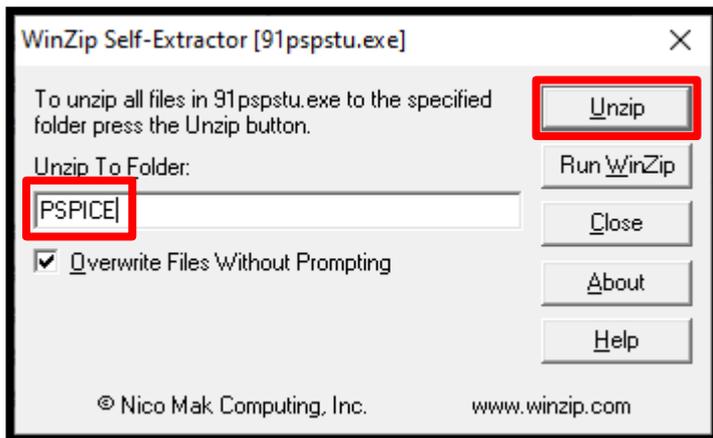


PSpice Schematics
Evaluation Version 9.1 - Web Update 1

(Parte II) Ambiente de Desenvolvimento

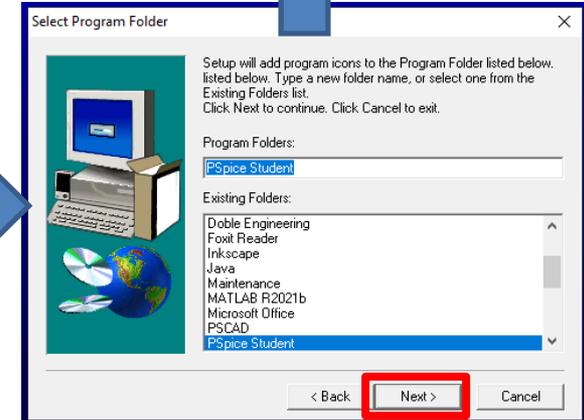
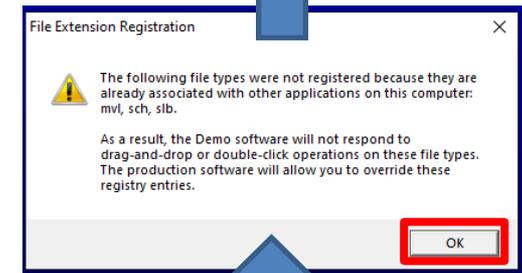
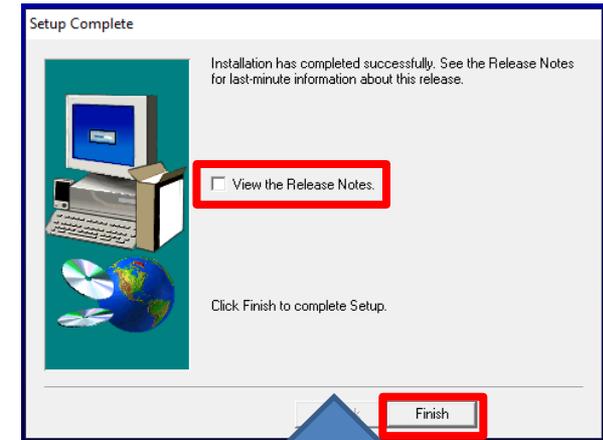
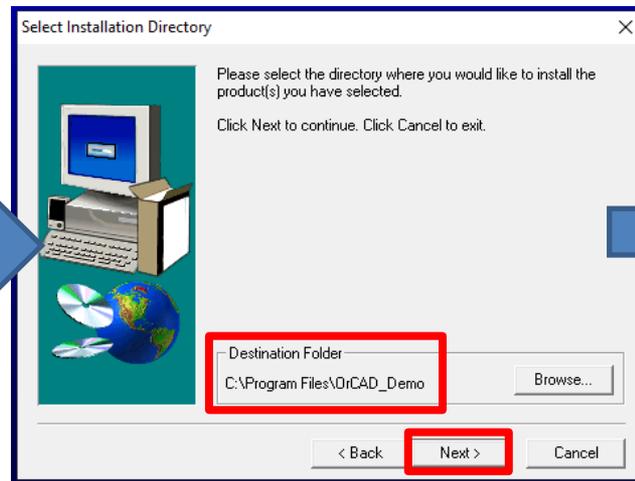
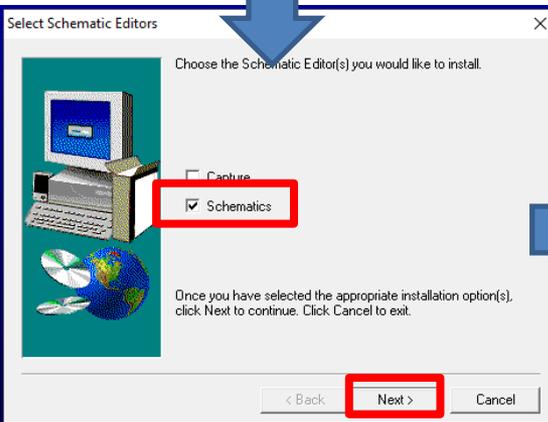
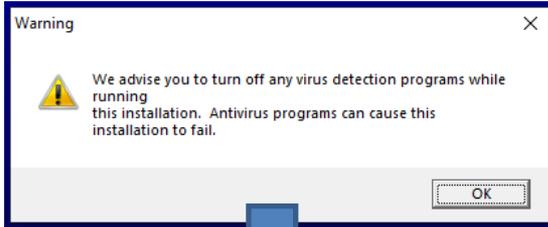
✓ PSPICE (Personal Spice)

✓ Instalação!



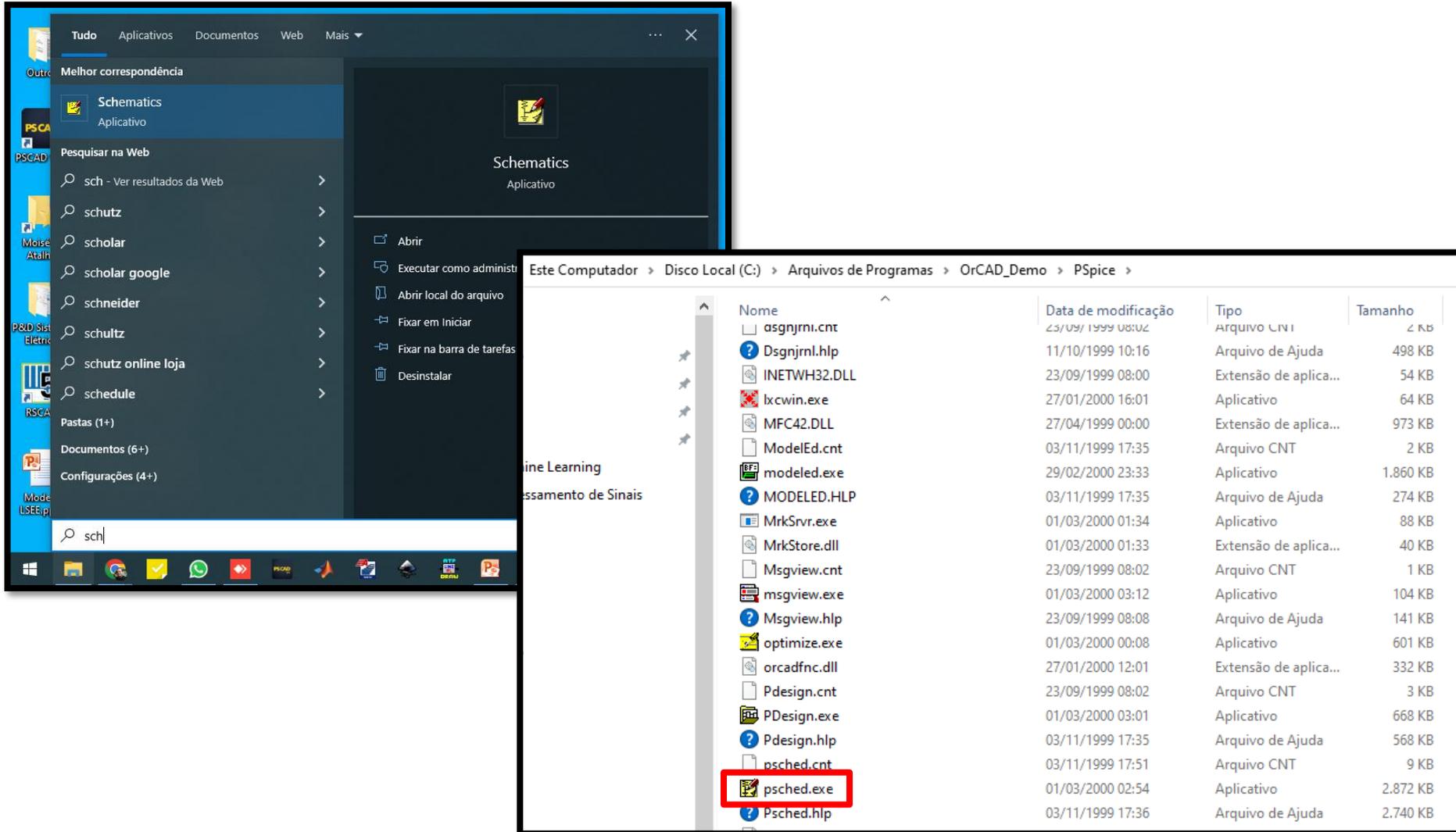
(Parte II) Ambiente de Desenvolvimento

✓ PSPICE (Personal Spice) – Instalação!



(Parte II) Ambiente de Desenvolvimento

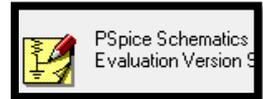
✓ PSPICE (Personal Spice) – Instalação!



The image shows two overlapping screenshots from a Windows operating system. The top-left screenshot displays a search window for 'Schematics', showing a list of search results with 'Schematics' as the top result. The bottom-right screenshot shows a file explorer window with the path 'Este Computador > Disco Local (C:) > Arquivos de Programas > OrCAD_Demo > PSpice'. The file list includes various files, with 'psched.exe' highlighted by a red box.

Nome	Data de modificação	Tipo	Tamanho
asgnjrni.cnt	23/09/1999 08:02	Arquivo CNT	2 KB
Dsgnrnl.hlp	11/10/1999 10:16	Arquivo de Ajuda	498 KB
INETWH32.DLL	23/09/1999 08:00	Extensão de aplica...	54 KB
lxcwin.exe	27/01/2000 16:01	Aplicativo	64 KB
MFC42.DLL	27/04/1999 00:00	Extensão de aplica...	973 KB
ModelEd.cnt	03/11/1999 17:35	Arquivo CNT	2 KB
modeled.exe	29/02/2000 23:33	Aplicativo	1.860 KB
MODELED.HLP	03/11/1999 17:35	Arquivo de Ajuda	274 KB
MrkSrvr.exe	01/03/2000 01:34	Aplicativo	88 KB
MrkStore.dll	01/03/2000 01:33	Extensão de aplica...	40 KB
Msgview.cnt	23/09/1999 08:02	Arquivo CNT	1 KB
msgview.exe	01/03/2000 03:12	Aplicativo	104 KB
Msgview.hlp	23/09/1999 08:08	Arquivo de Ajuda	141 KB
optimize.exe	01/03/2000 00:08	Aplicativo	601 KB
orcadfnc.dll	27/01/2000 12:01	Extensão de aplica...	332 KB
Pdesign.cnt	23/09/1999 08:02	Arquivo CNT	3 KB
PDesign.exe	01/03/2000 03:01	Aplicativo	668 KB
Pdesign.hlp	03/11/1999 17:35	Arquivo de Ajuda	568 KB
psched.cnt	03/11/1999 17:51	Arquivo CNT	9 KB
psched.exe	01/03/2000 02:54	Aplicativo	2.872 KB
Psched.hlp	03/11/1999 17:36	Arquivo de Ajuda	2.740 KB

(Parte II) Ambiente de Desenvolvimento

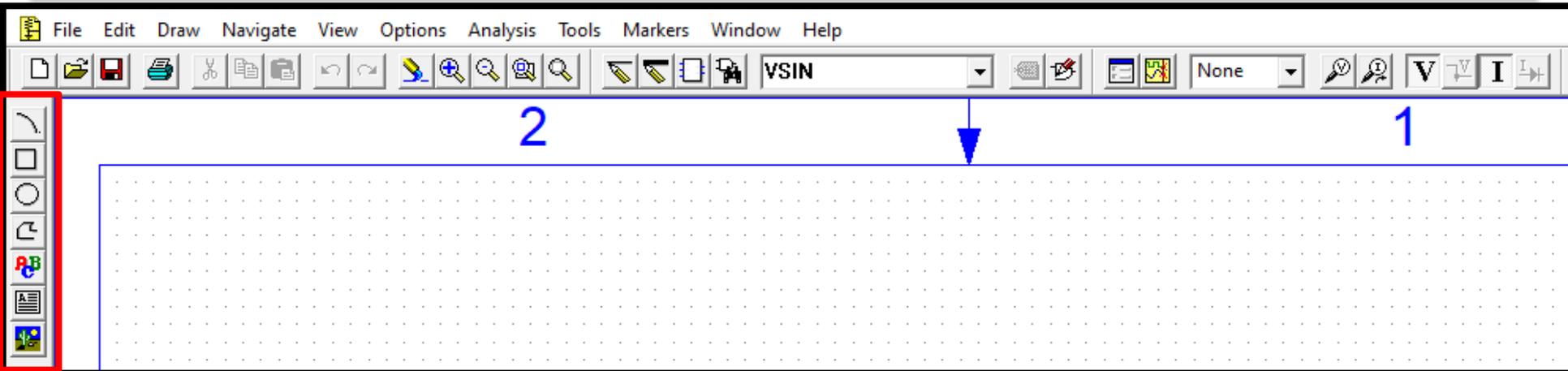


The screenshot displays the PSpice Schematics - [Schematic1.p.1] application window. The main workspace is a grid with a blue border. A blue arrow labeled '2' points down from the top border, and another labeled '1' points up from the bottom border. A blue arrow labeled 'B' points right from the left border, and another labeled 'A' points left from the right border. The workspace contains three windows:

- PSpice Design Manager**: A window with a menu bar (File, Workspace, View, Tools, Help) and a toolbar. The main area is empty. A status bar at the bottom says "For Help, press F1".
- PSpice Message Viewer - [Schematic1]**: A window with a menu bar (File, Edit, View, Tools, Window, Help) and a toolbar. It contains a table with columns: Severity, Origin, Time, and Message Text. The table is empty. A status bar at the bottom says "0 Messages: 0 Error, 0 Warning, 0 Info".
- PSpice Application...**: A small dialog box with a green circular icon, the text "Version 9.1 - Web Update 1", and two buttons: "Activity..." and "Close".

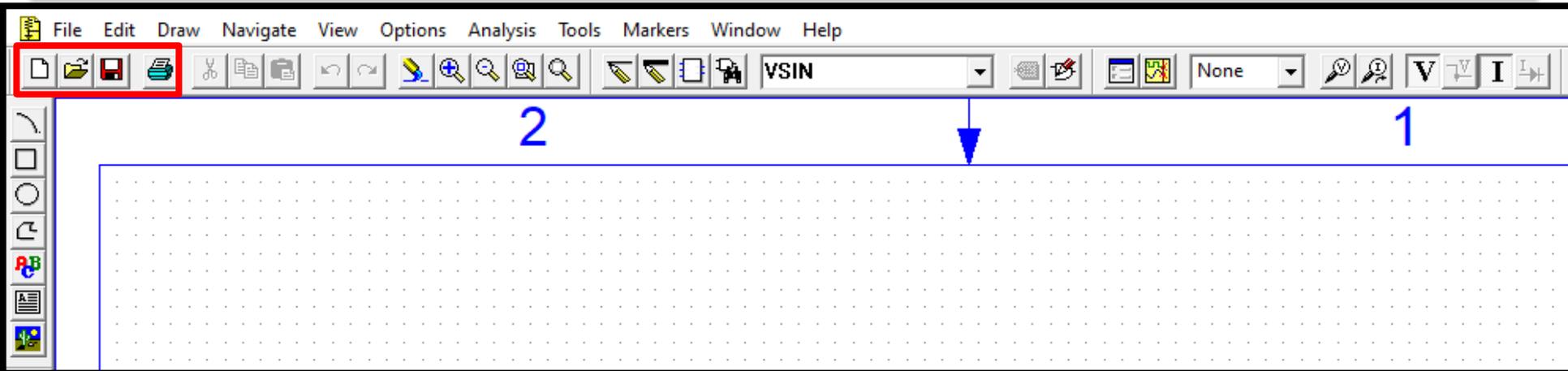
The status bar at the bottom left of the main window shows "1.74, 0.00".

(Parte II) Ambiente de Desenvolvimento



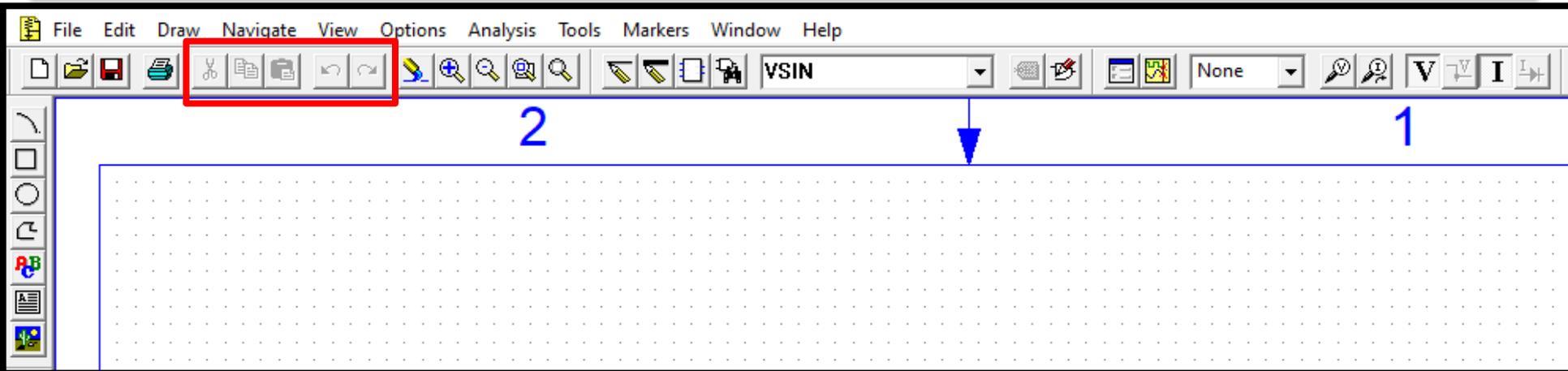
**Ferramentas
para desenho
no ambiente
PSPICE**

(Parte II) Ambiente de Desenvolvimento



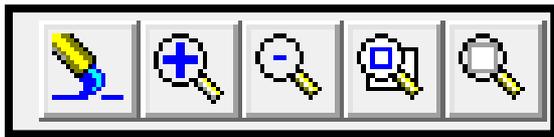
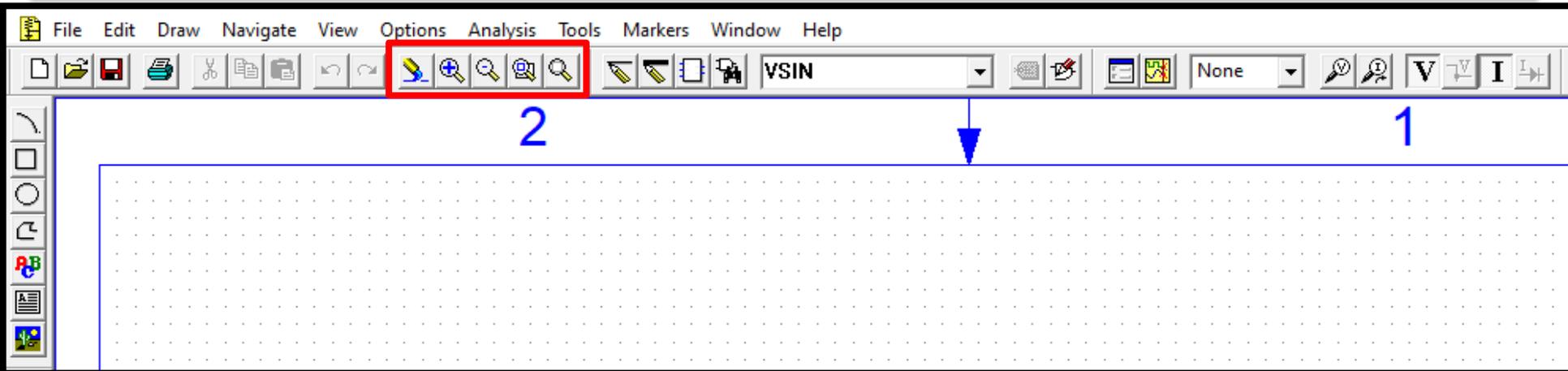
**Criar novo projeto,
abrir projeto
existente ou salvar
projeto atual**

(Parte II) Ambiente de Desenvolvimento



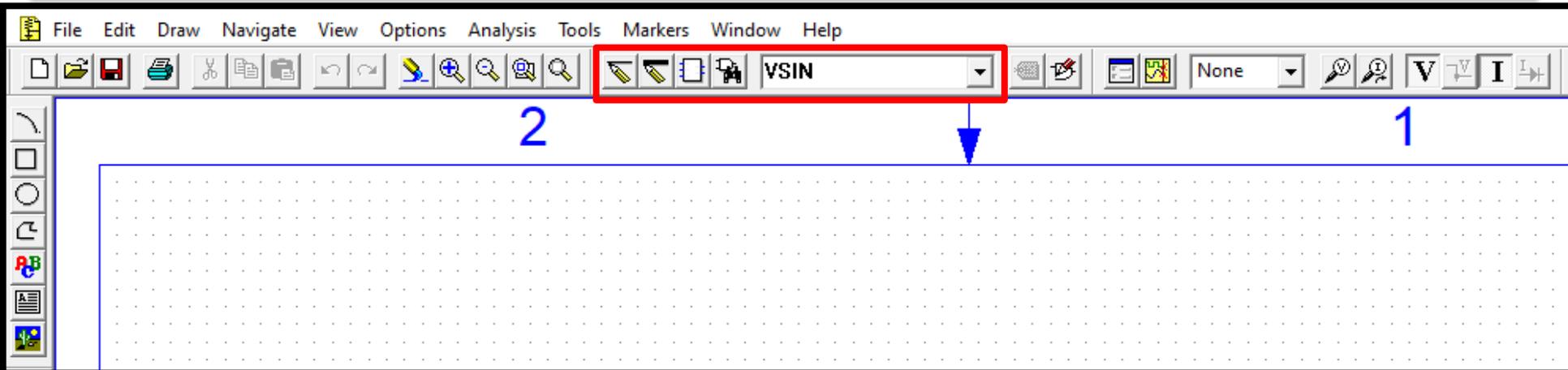
**Recortar, copiar,
colar, desfazer
e refazer**

(Parte II) Ambiente de Desenvolvimento



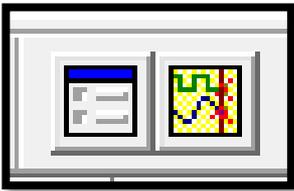
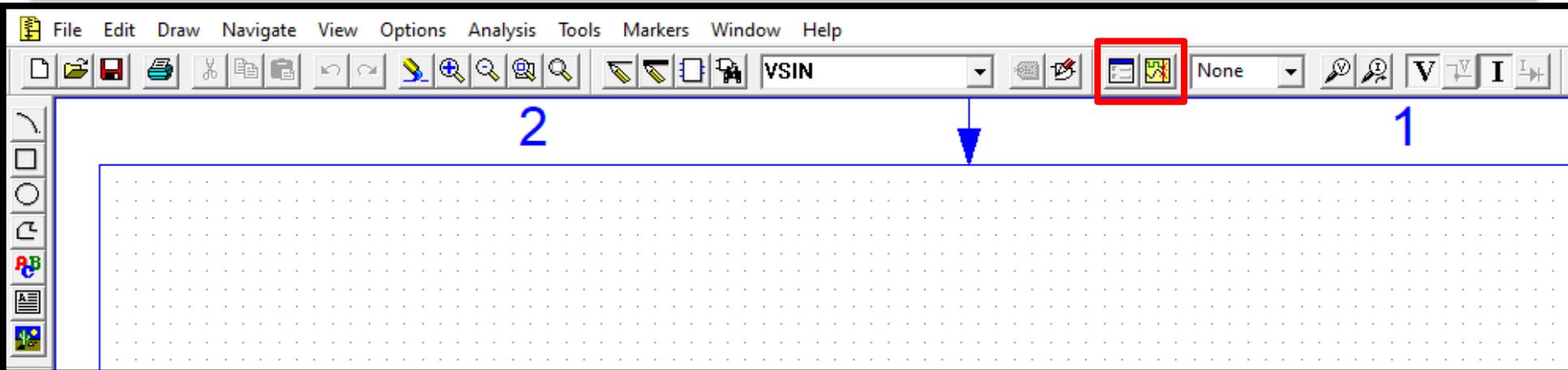
**Ferramentas
de zoom para
o ambiente**

(Parte II) Ambiente de Desenvolvimento



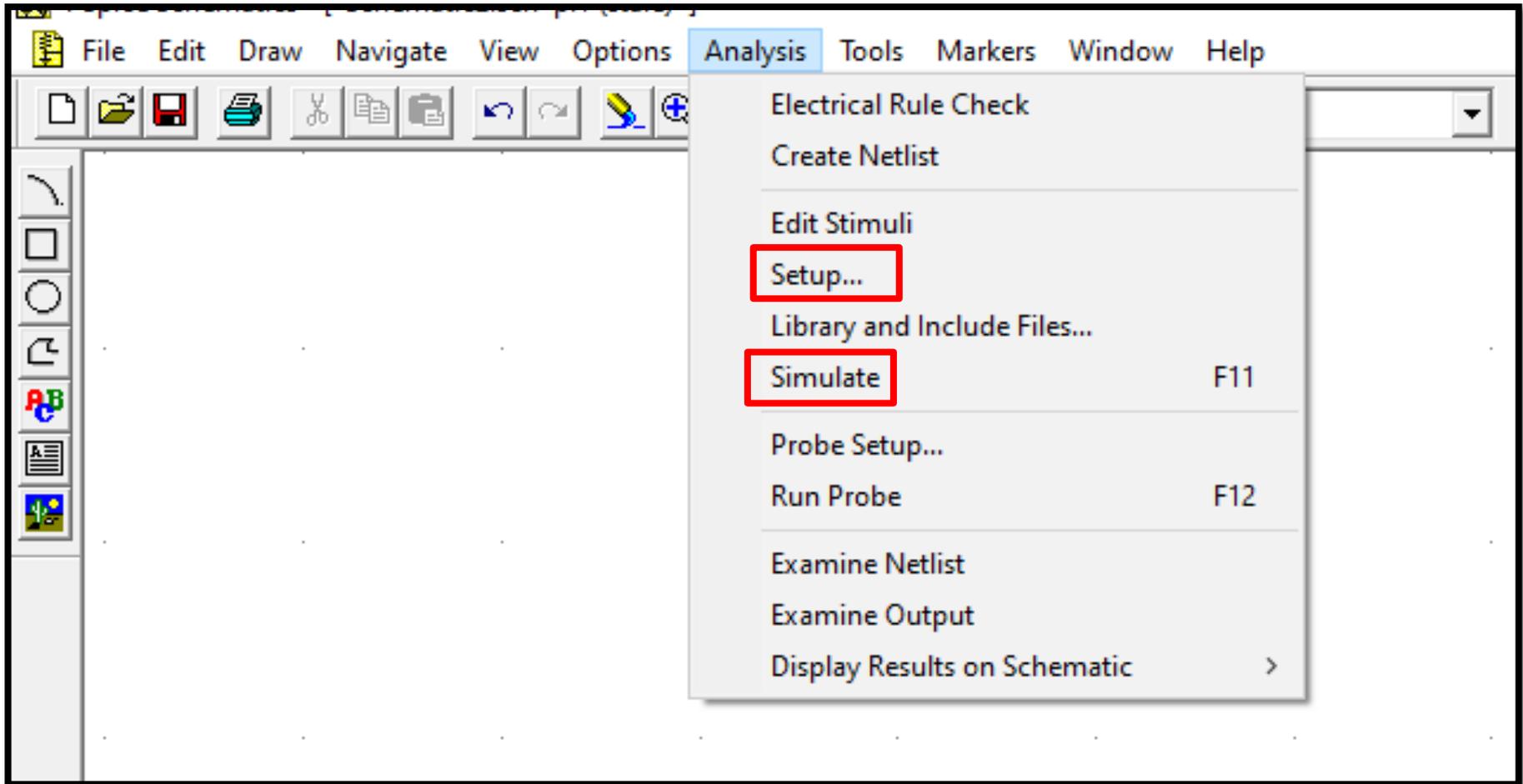
**Desenho de
linhas/barras e
inserção de
novos componentes**

(Parte II) Ambiente de Desenvolvimento

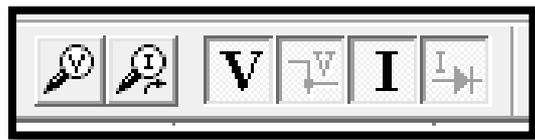
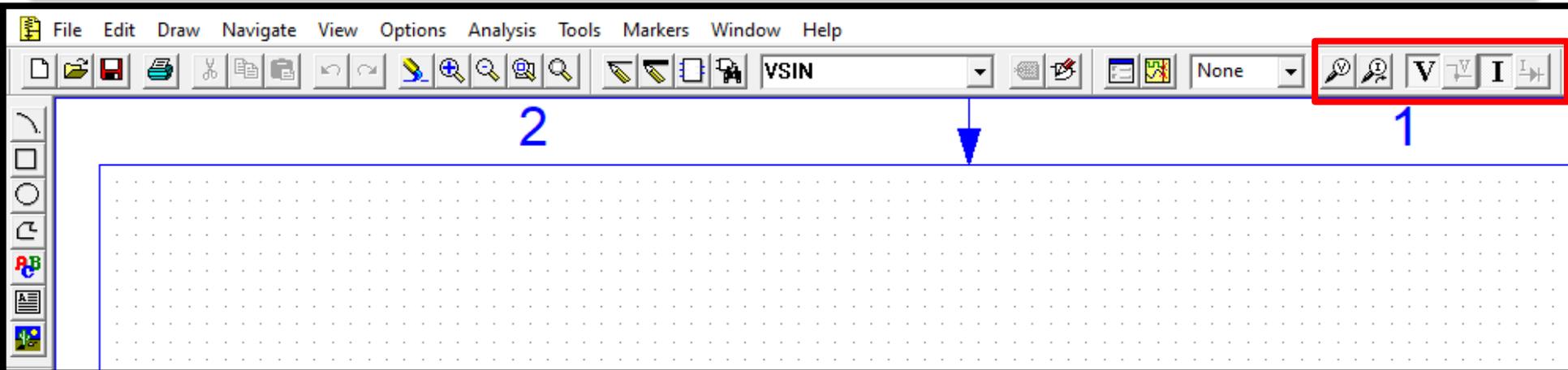


**Configurações
da simulação e
opção de Simular...**

(Parte II) Ambiente de Desenvolvimento



(Parte II) Ambiente de Desenvolvimento



**Inserção de medidores
e opção de mostrar
tensões e correntes
nos nós do circuito**

(Parte II) Regras para Inserção/Parametrização

- ✓ O PSPICE não é *case sensitive*
- ✓ É obrigatória a definição de um nó de referência (*ground*);
- ✓ O valor do elemento é dado em unidades do SI
- ✓ Valores fracionários fornecidos com **ponto decimal**
- ✓ Valor Exponencial: E3, E-3, E6, etc

Potencia de dez	Nome no SI	Sufixo do SPICE
-15	femto	f ou F
-12	pico	p ou P
-9	nano	n ou N
-6	micro	u ou U
-3	mili	m ou M
3	kilo	k ou K
6	mega	meg ou MEG
9	giga	g ou G
12	tera	t ou T

Unidade	Abreviatura
Volt	V
Ampére	A
Hertz	HZ
ohm (Ω)	OHM
Henry	H
Farad	F
graus	DEG

(Parte II) Inserção dos Elementos

2

1

Caso não apareça "Full List" digite * na primeira abertura

Part Name: [r]

Description: resistor

Description Search: []
Create New Part List [v]
Search

Library: C:\Program Files\ORCAD_Demo\PSpic

Close

Place

Place & Close

Help

Libraries...

23 << Basic

Full List

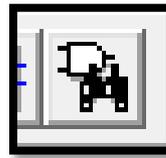
Partial List - *r

Edit Symbol

(Parte II) Código e Parametrização dos Elementos

✓ Cada elemento do circuito possui nome/termo único;

✓ Elementos Passivos:



Part Browser Advanced

Part Name: R
Description: resistor

Description Search: Create New Part List Search

Library: C:\Program Files\OrCAD_Demo\PSpic

R?
1k

Edit Symbol

Full List

Part Browser Advanced

Part Name: L
Description: inductor

Description Search: Create New Part List Search

Library: C:\Program Files\OrCAD_Demo\PSpic

L?
10uH

Edit Symbol

Full List

Part Browser Advanced

Part Name: C
Description: Capacitor

Description Search: Create New Part List Search

Library: C:\Program Files\OrCAD_Demo\PSpic

C?
1n

Edit Symbol

(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Elementos Passivos:

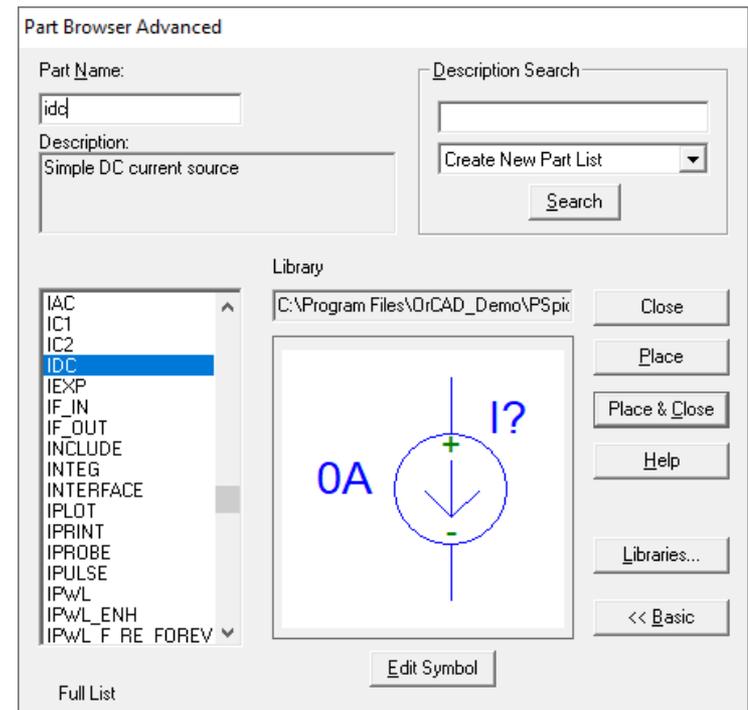
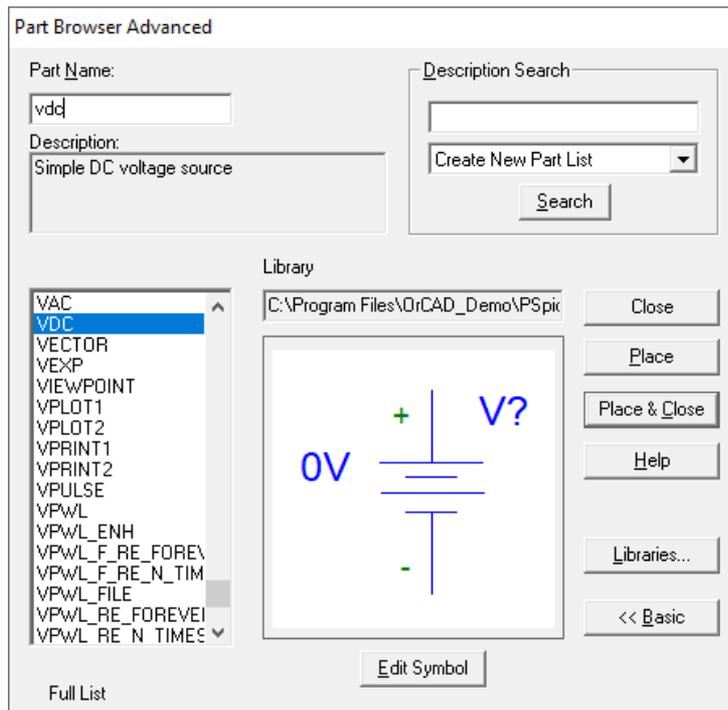
The image displays three overlapping dialog boxes from a circuit design software, each showing the configuration for a different passive component. The background shows a circuit schematic with a resistor, a capacitor, and an inductor.

- Resistor (R10):** The dialog box shows the component name 'R10' and 'PartName: r'. The 'Name' field contains 'VALUE' and the 'Value' field contains '= 2'. The 'Description' field contains 'VALUE=2' and 'TOLERANCE='.
- Capacitor (C1):** The dialog box shows the component name 'C1' and 'PartName: c'. The 'Name' field contains 'VALUE' and the 'Value' field contains '= 0.01'. The 'Description' field contains 'VALUE=0.01', 'IC=5.506', and 'TOLERANCE='.
- Inductor (L1):** The dialog box shows the component name 'L1' and 'PartName: L'. The 'Name' field contains 'VALUE' and the 'Value' field contains '= 0.01H'. The 'Description' field contains 'VALUE=0.01H', 'IC=2.065', and 'TOLERANCE='.

Each dialog box includes buttons for 'Save Attr', 'Change Display', 'Delete', 'OK', and 'Cancel'. The background circuit schematic shows a resistor labeled 'R?' with a value of '1k', a capacitor labeled 'C?' with a value of '1n', and an inductor labeled 'L?' with a value of '10uH'.

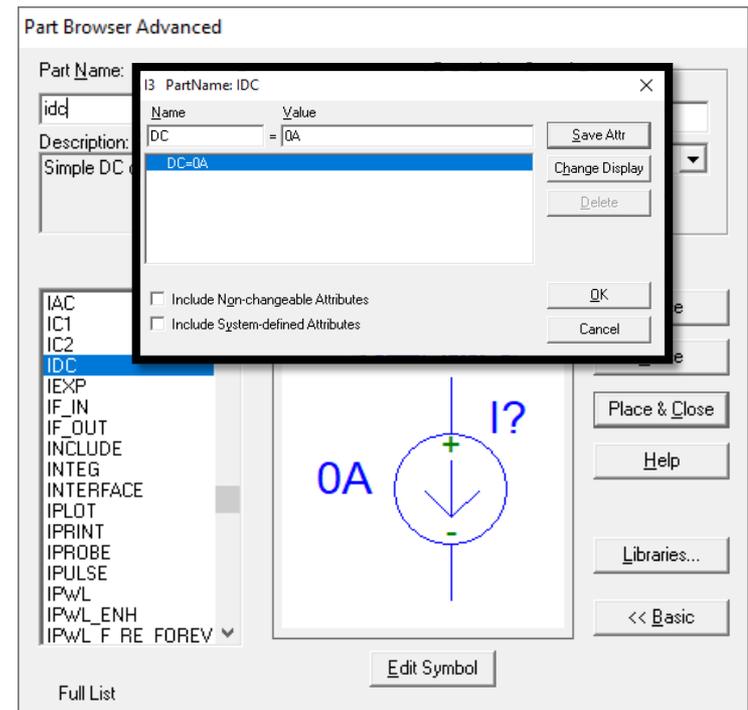
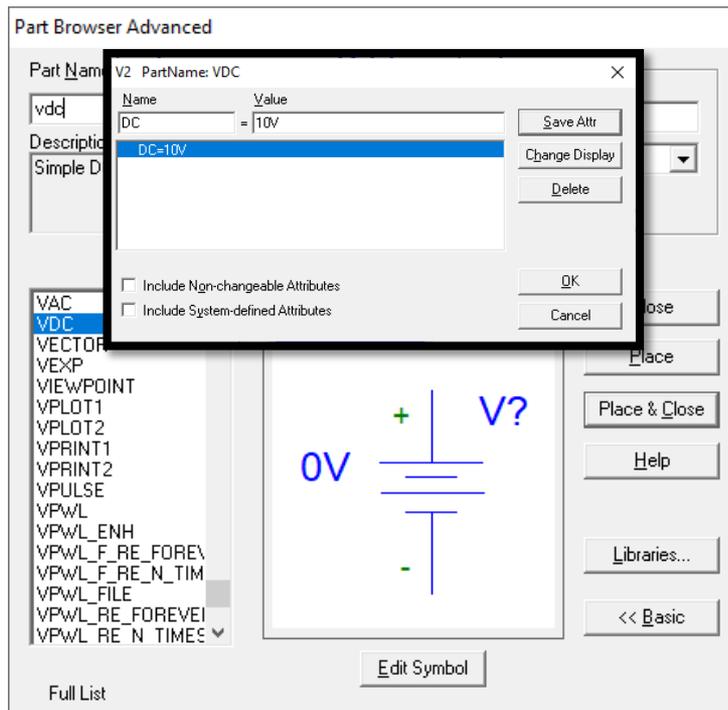
(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Fontes DC Independentes:



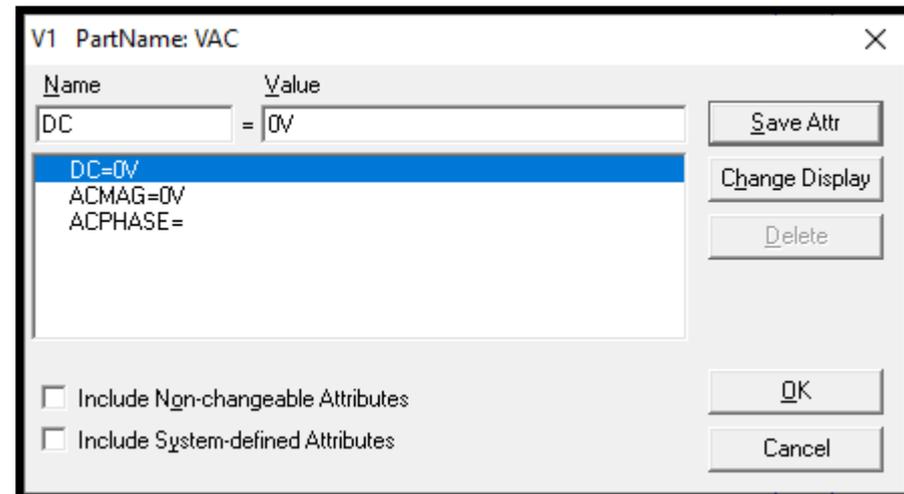
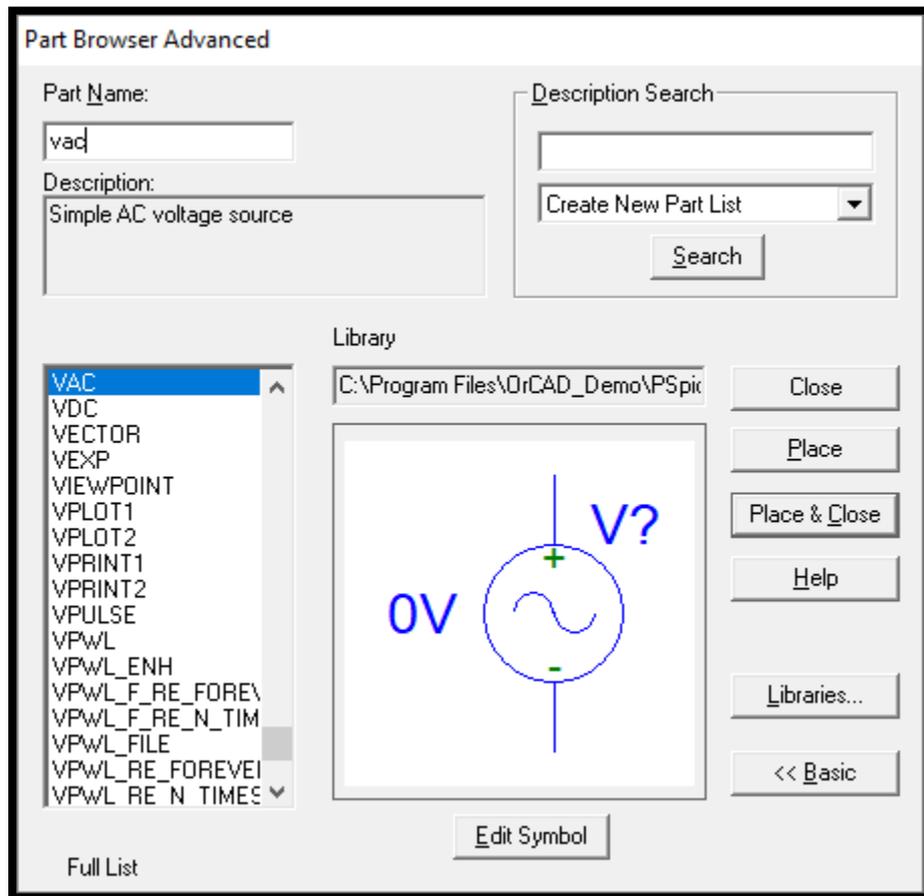
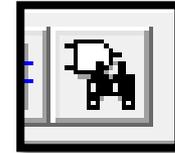
(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Fontes DC Independentes:



(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Fonte AC para Análise Fasorial do Circuito:



(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Fontes Dependentes/Controladas:

The image displays four instances of the 'Part Browser Advanced' dialog box, each showing a different dependent source component. A small circuit diagram with a dependent current source is shown in the center, with a red box highlighting the component being selected in the browser.

- Top Left:** Shows the 'FNOM' component. The 'Part Name' is 'F', the 'Description' is 'Current-controlled current source', and the symbol is a square with a downward arrow and a circle containing a plus sign.
- Top Right:** Shows the 'HNOM' component. The 'Part Name' is 'H', the 'Description' is 'Current-controlled voltage source', and the symbol is a square with a downward arrow and a circle containing a plus sign.
- Bottom Left:** Shows the 'GNOM' component. The 'Part Name' is 'G', the 'Description' is 'Voltage-controlled current source', and the symbol is a square with a plus sign and a circle containing a plus sign.
- Bottom Right:** Shows the 'ENOM' component. The 'Part Name' is 'E', the 'Description' is 'Voltage-controlled voltage source', and the symbol is a square with a plus sign and a circle containing a plus sign.

Each dialog box includes a 'Library' list on the left, a 'Description Search' field, a 'Create New Part List' dropdown, and a 'Search' button. The 'Library' list for each instance shows a scrollable list of components, with the selected component highlighted.

(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Fontes Dependentes:

The image displays a software interface for defining circuit components. It features three 'Part Browser Advanced' windows and a central dialog box for parameterizing a dependent current source.

Part Browser Advanced (Left): Shows a search for 'Current-controlled current source'. The library list includes components like FileStim, FPOLY, FTABLE, G, GAIN, GFREQ, GLAPLACE, GLIMIT, GLOBAL, GMULT, GND_ANALOG, GND_EARTH, GPOLY, GSUM, GTABLE, and GVALUE. A preview window shows a dependent current source symbol labeled 'FNOM' with a gain parameter 'F?'.

Part Browser Advanced (Top Center): Shows a search for 'Current-controlled voltage source'. A preview window shows a dependent voltage source symbol labeled 'GNOM' with a gain parameter 'G?'.

Part Browser Advanced (Right): Shows a search for 'Current-controlled voltage source'. A preview window shows a dependent voltage source symbol labeled 'ENOM' with a gain parameter 'E?'.

Central Dialog Box (G1 PartName: G): This dialog is used to define the gain parameter for the dependent current source. It contains a table with the following data:

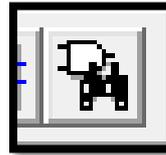
Name	Value
GAIN	= 2.5
GAIN=2.5	

Buttons in the dialog include 'Save Attr', 'Change Display', 'Delete', 'OK', and 'Cancel'. There are also checkboxes for 'Include Non-changeable Attributes' and 'Include System-defined Attributes', both of which are currently unchecked.

(Parte II) Código e Parametrização dos Elementos

✓ Cada elemento do circuito possui nome/termo único;

✓ Chaves:



Part Browser Advanced

Part Name: Sw_tClose

Description: Switch: closes at tClose=?

Description Search: Create New Part List

Search

Library: C:\Program Files\OrCAD_Demo\PSpic

Close

Place

Place & Close

Help

Libraries...

<< Basic

Edit Symbol

Full List

Diagram: tClose=0, 1, 2, U?

The diagram shows a switch symbol with two terminals labeled 1 and 2. A green arrow points from terminal 1 to terminal 2. The text "tClose=0" is written above the terminals, and "U?" is written below them.

Part Browser Advanced

Part Name: Sw_tOpen

Description: Switch: opens at tsw=?

Description Search: Create New Part List

Search

Library: C:\Program Files\OrCAD_Demo\PSpic

Close

Place

Place & Close

Help

Libraries...

<< Basic

Edit Symbol

Full List

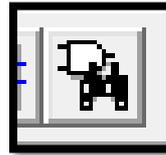
Diagram: tOpen=0, 1, 2, U?

The diagram shows a switch symbol with two terminals labeled 1 and 2. A green arrow points from terminal 1 to terminal 2. The text "tOpen=0" is written above the terminals, and "U?" is written below them.

(Parte II) Código e Parametrização dos Elementos

✓ Cada elemento do circuito possui nome/termo único;

✓ Chaves:



Part Browser Advanced

U2 PartName: Sw_tClose

Name	Value
tClose	= 10

tClose=10
ttran=1u
Rclosed=0.01
Ropen=1Meg

Include Non-changeable Attributes
 Include System-defined Attributes

OK
Cancel

Full List

Sw_tClose

tClose=0

1 2

U?

Edit Symbol

Part Browser Advanced

U3 PartName: Sw_tOpen

Name	Value
tOpen	= 0

tOpen=0
ttran=1u
Rclosed=0.01
Ropen=1Meg

Include Non-changeable Attributes
 Include System-defined Attributes

OK
Cancel

Full List

Sw_tOpen

tOpen=0

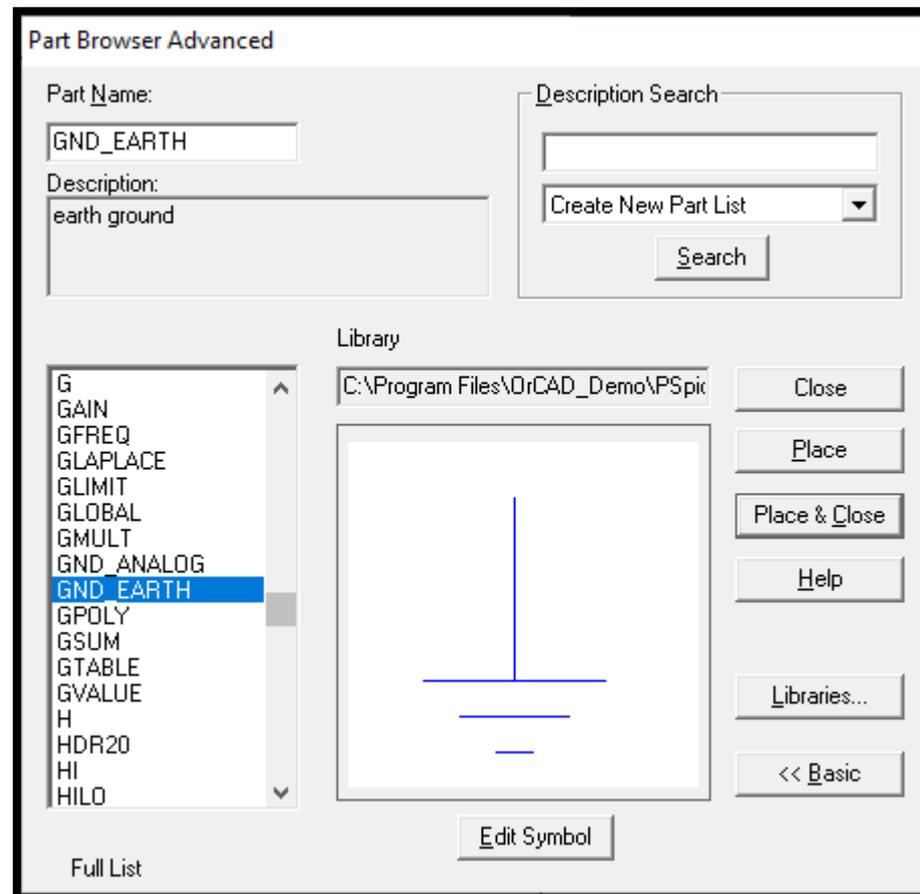
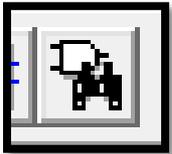
1 2

U?

Edit Symbol

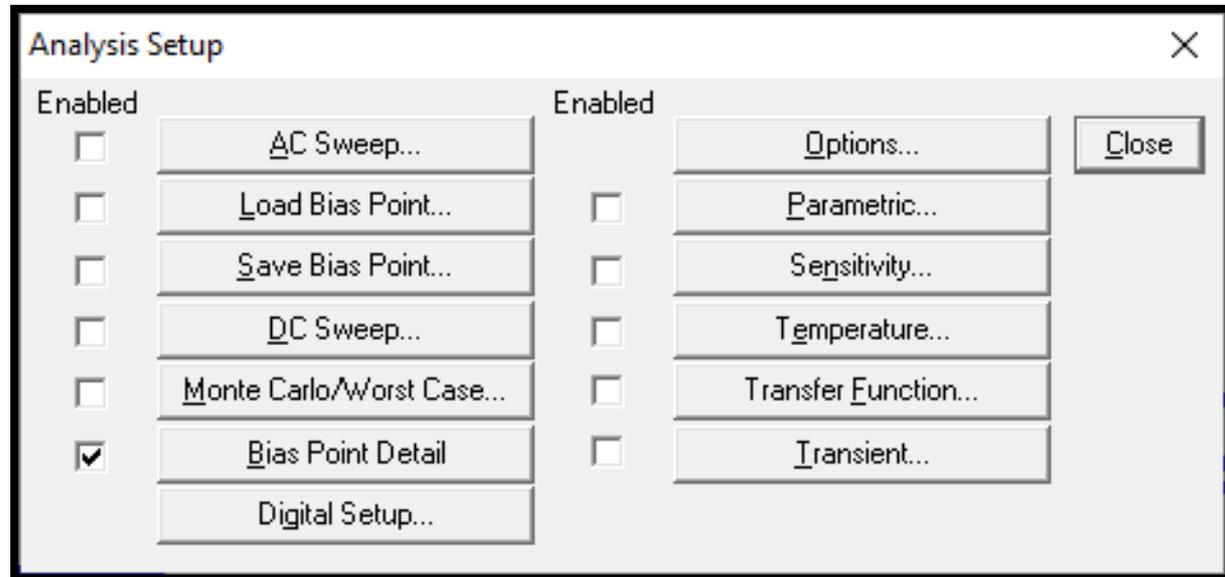
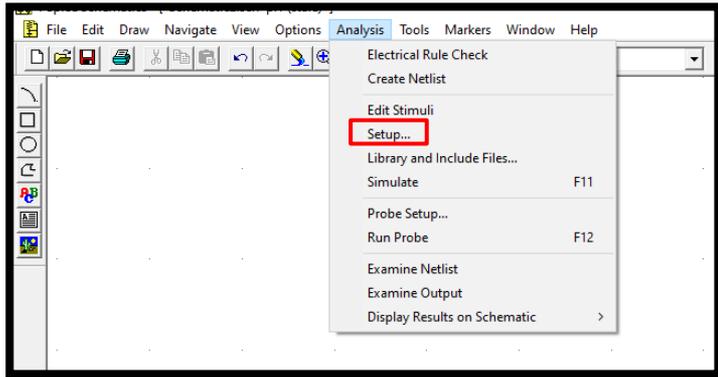
(Parte II) Código e Parametrização dos Elementos

- ✓ Cada elemento do circuito possui nome/termo único;
- ✓ Nó de referência do circuito:



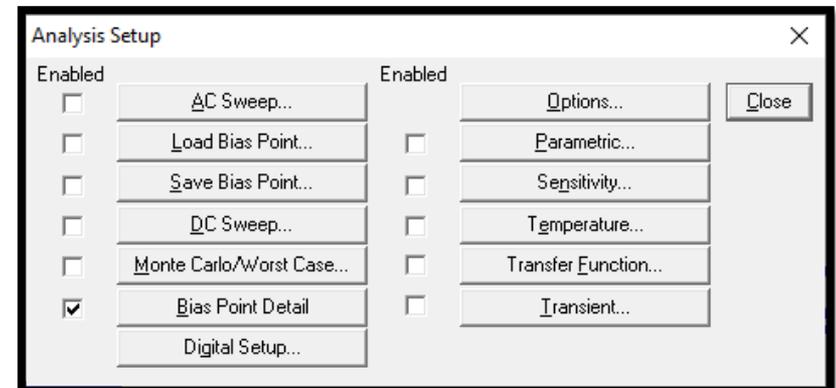
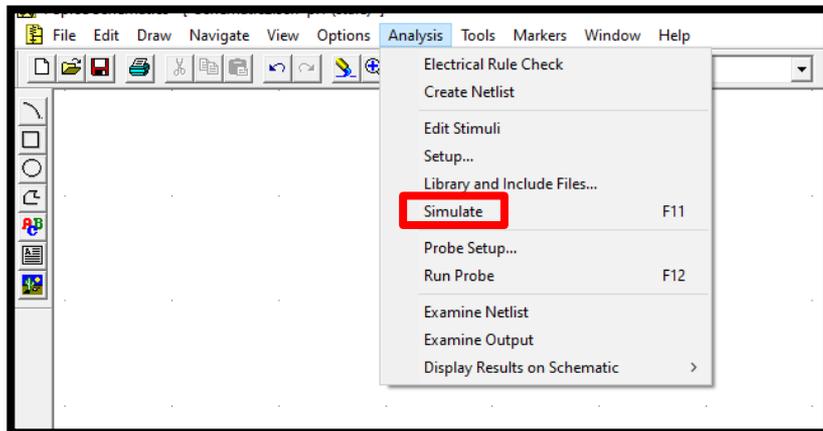
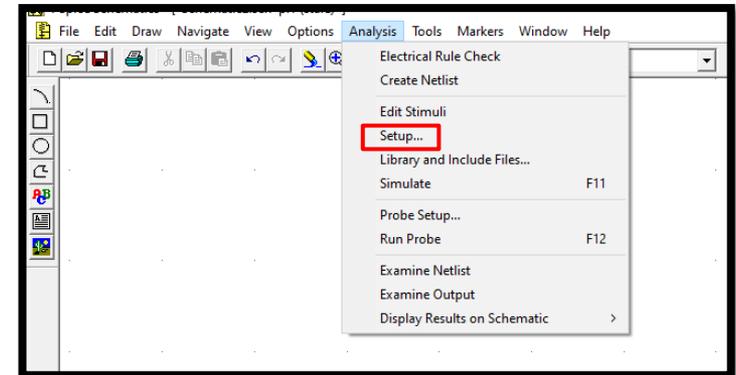
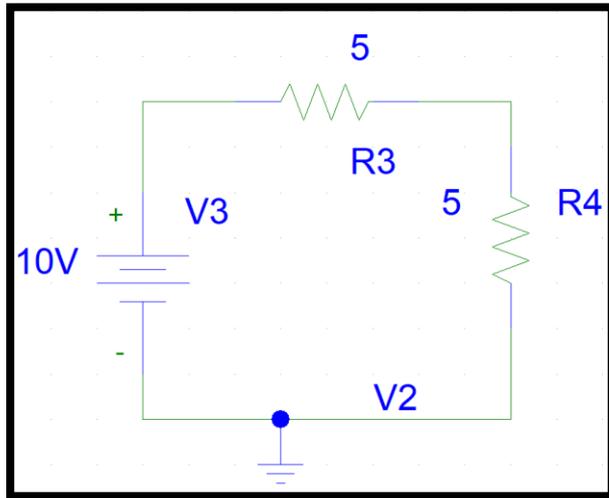
(Parte II) Tipos de Análises

- ✓ **Ponto de operação (*V dos nós e I das fontes*)**
- ✓ Chamada pelo PSPICE de “Bias Point Detail”



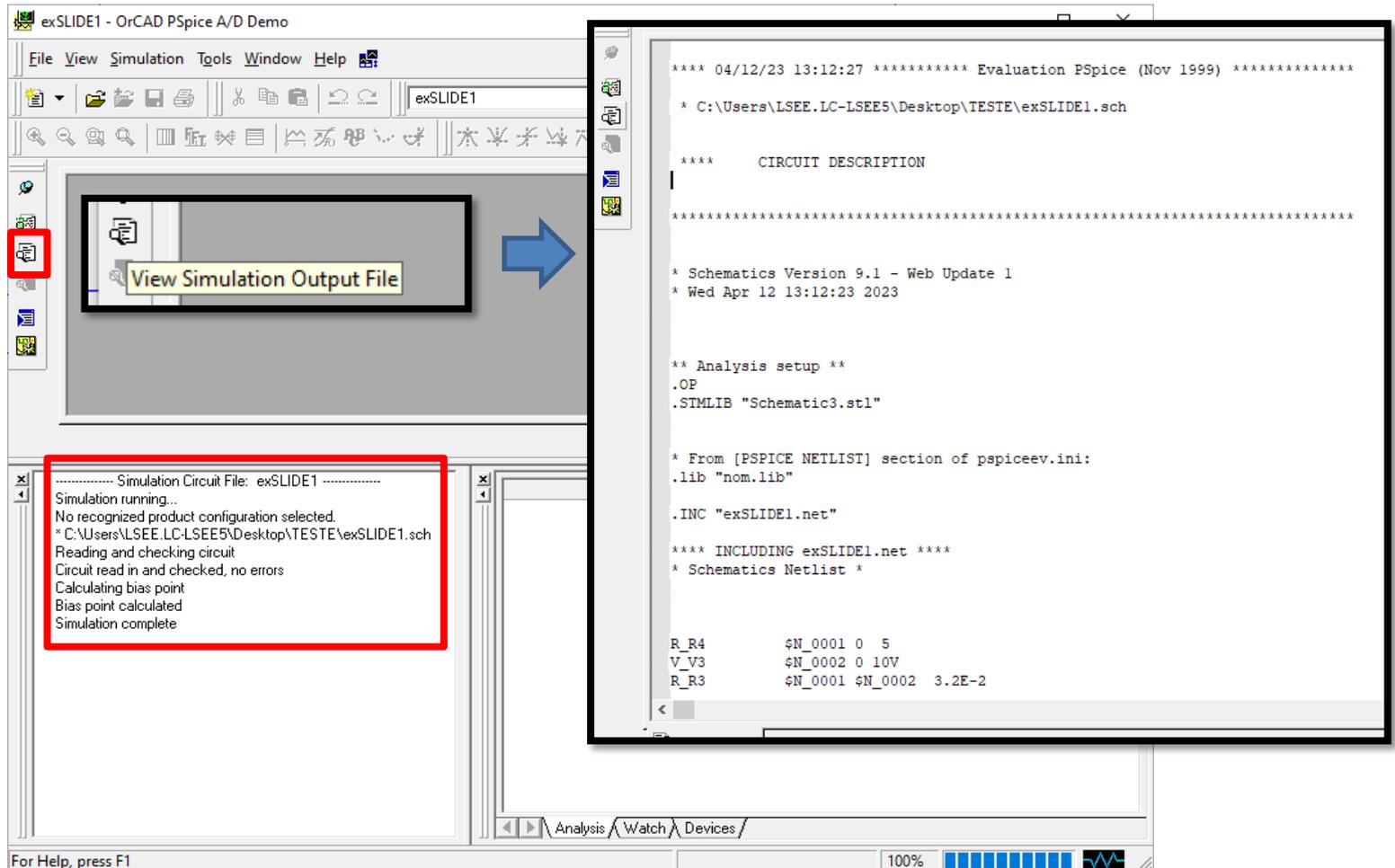
(Parte II) Tipos de Análises

- ✓ **Ponto de operação (*V dos nós e I das fontes*)**
- ✓ Chamada pelo PSPICE de “Bias Point Detail”



(Parte II) Tipos de Análises

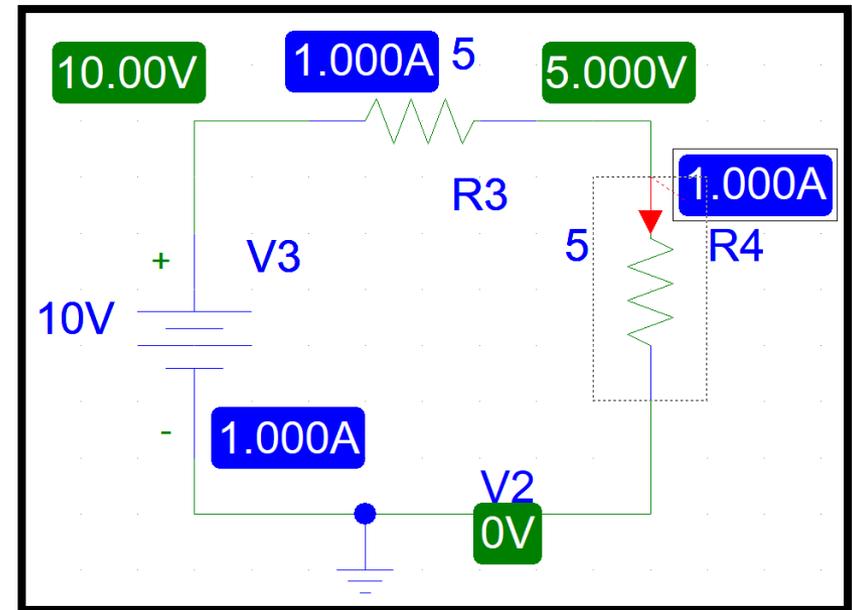
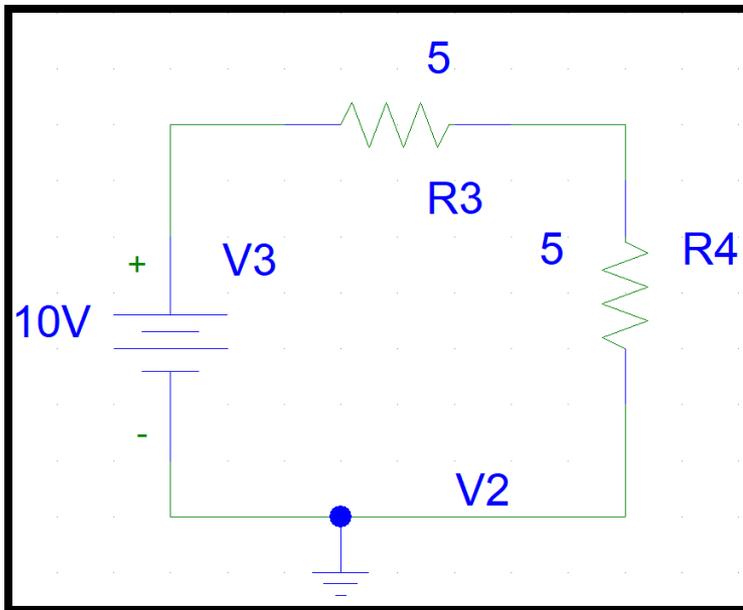
- ✓ Ponto de operação (*V dos nós e I das fontes*)
- ✓ Chamada pelo PSPICE de “Bias Point Detail”



(Parte II) Tipos de Análises

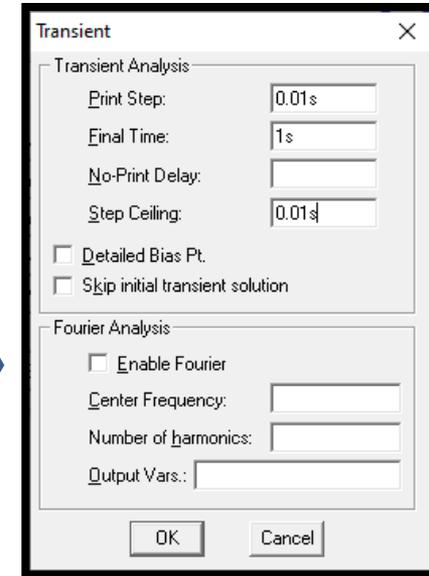
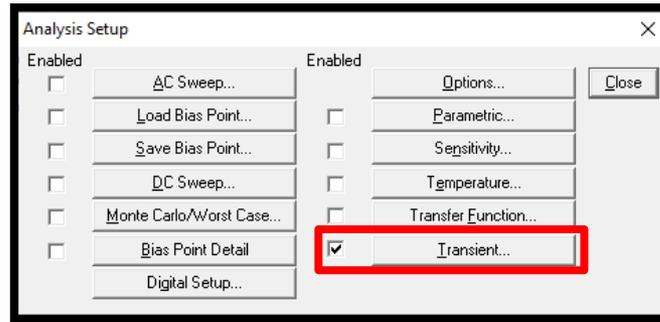
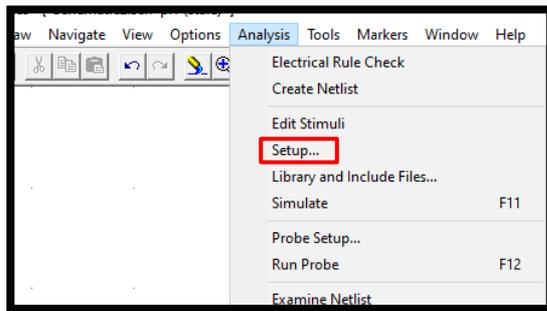
- ✓ Ponto de operação (*V dos nós e I das fontes*)
- ✓ Chamada pelo PSPICE de “Bias Point Detail”

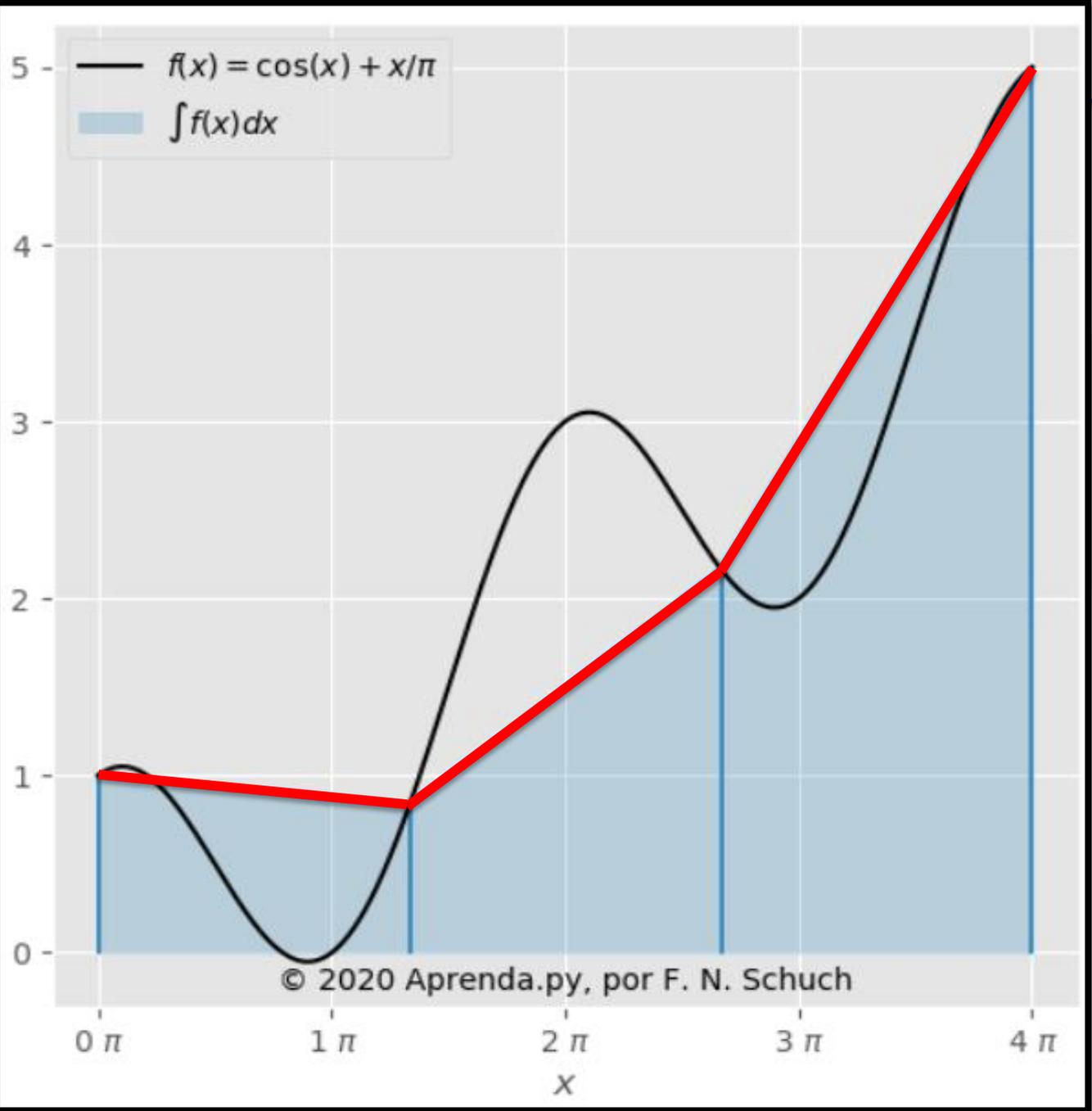
??? Exercício 01 ???



(Parte II) Tipos de Análises

- ✓ Análises de Transitórios
- ✓ Análises onde verificar as oscilografias é importante.
- ✓ Chamada pelo PSPICE de “Transient...”





Transient ✕

Transient Analysis

Print Step:

Final Time:

No-Print Delay:

Step Ceiling:

Detailed Bias Pt.

Skip initial transient solution

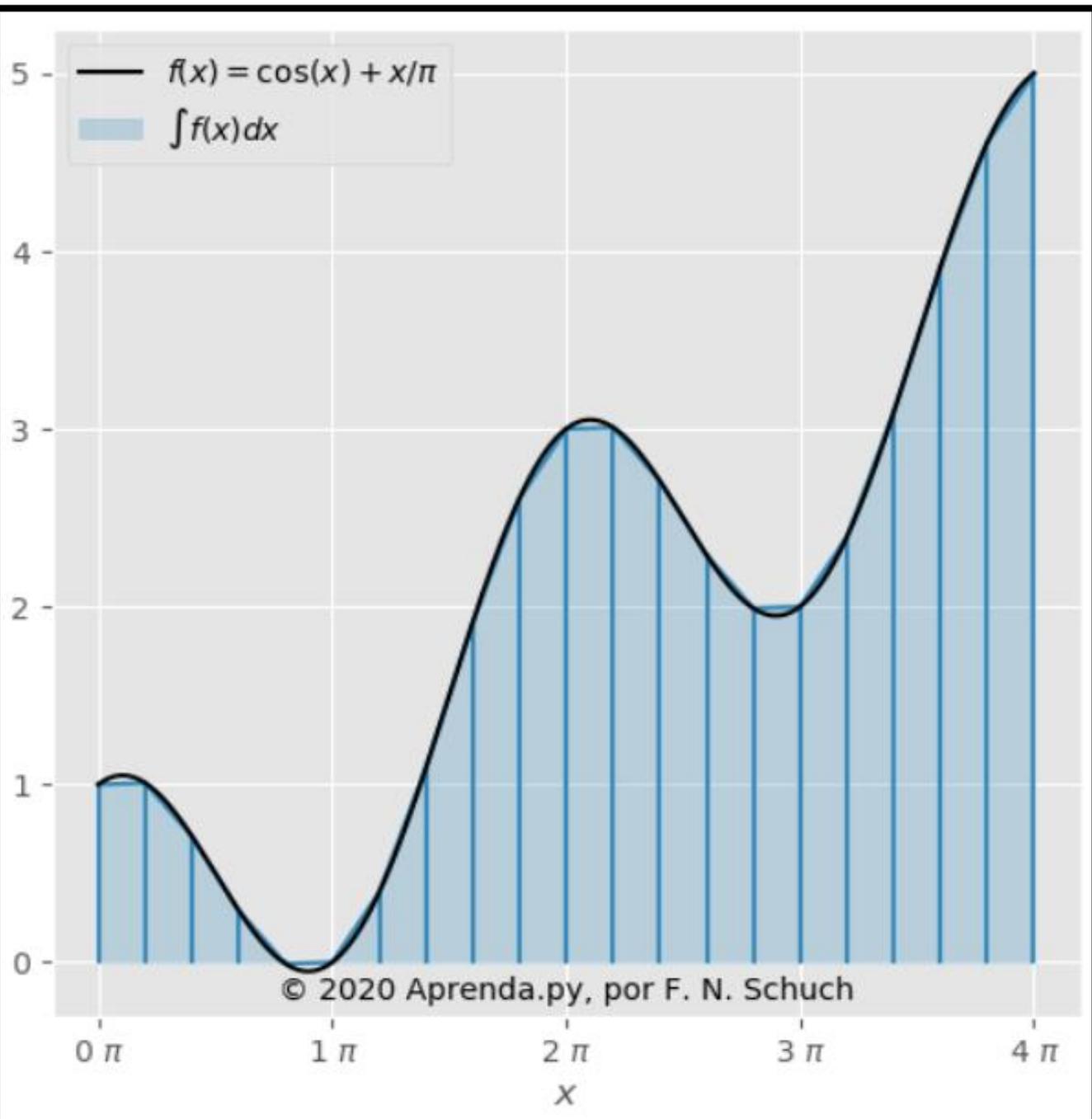
Fourier Analysis

Enable Fourier

Center Frequency:

Number of harmonics:

Output Vars.:



Transient

Transient Analysis

Print Step: 0.01s

Final Time: 1s

No-Print Delay:

Step Ceiling: 0.01s

Detailed Bias Pt.

Skip initial transient solution

Fourier Analysis

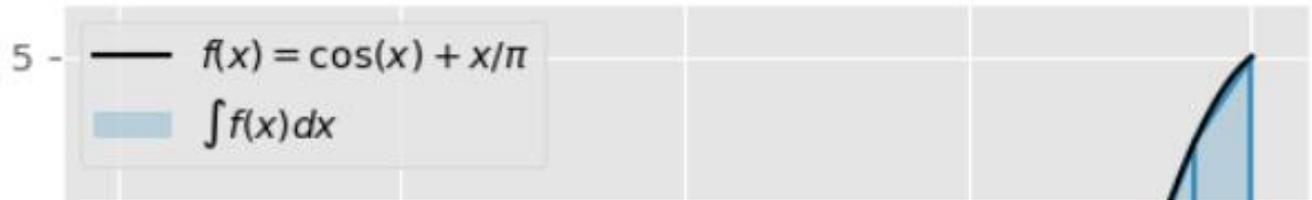
Enable Fourier

Center Frequency:

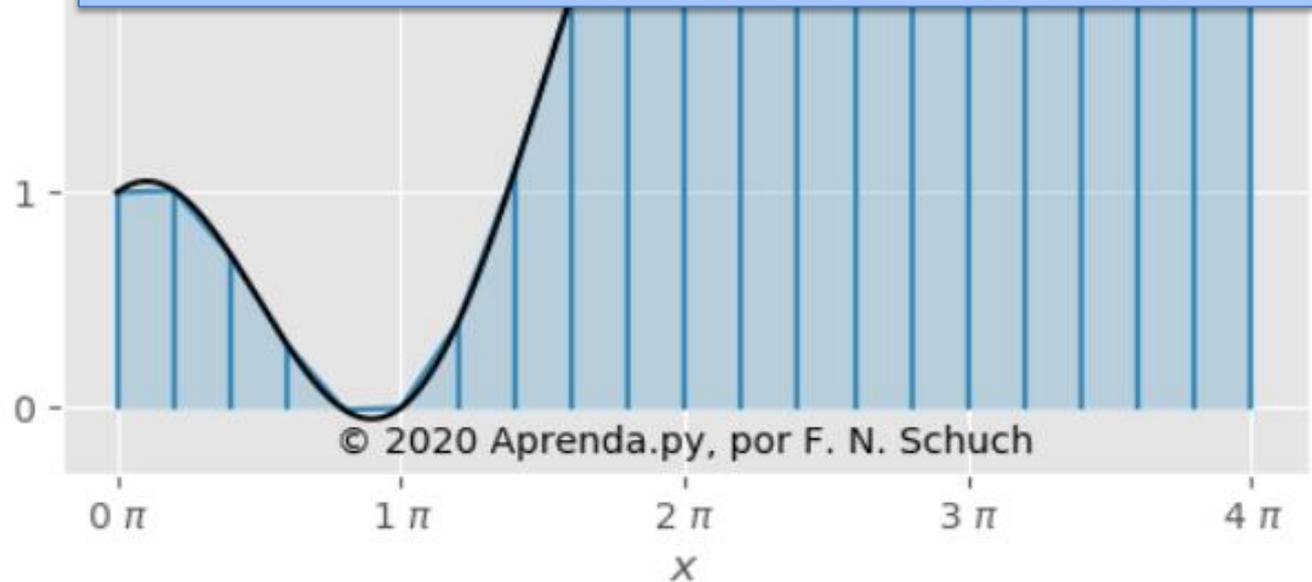
Number of harmonics:

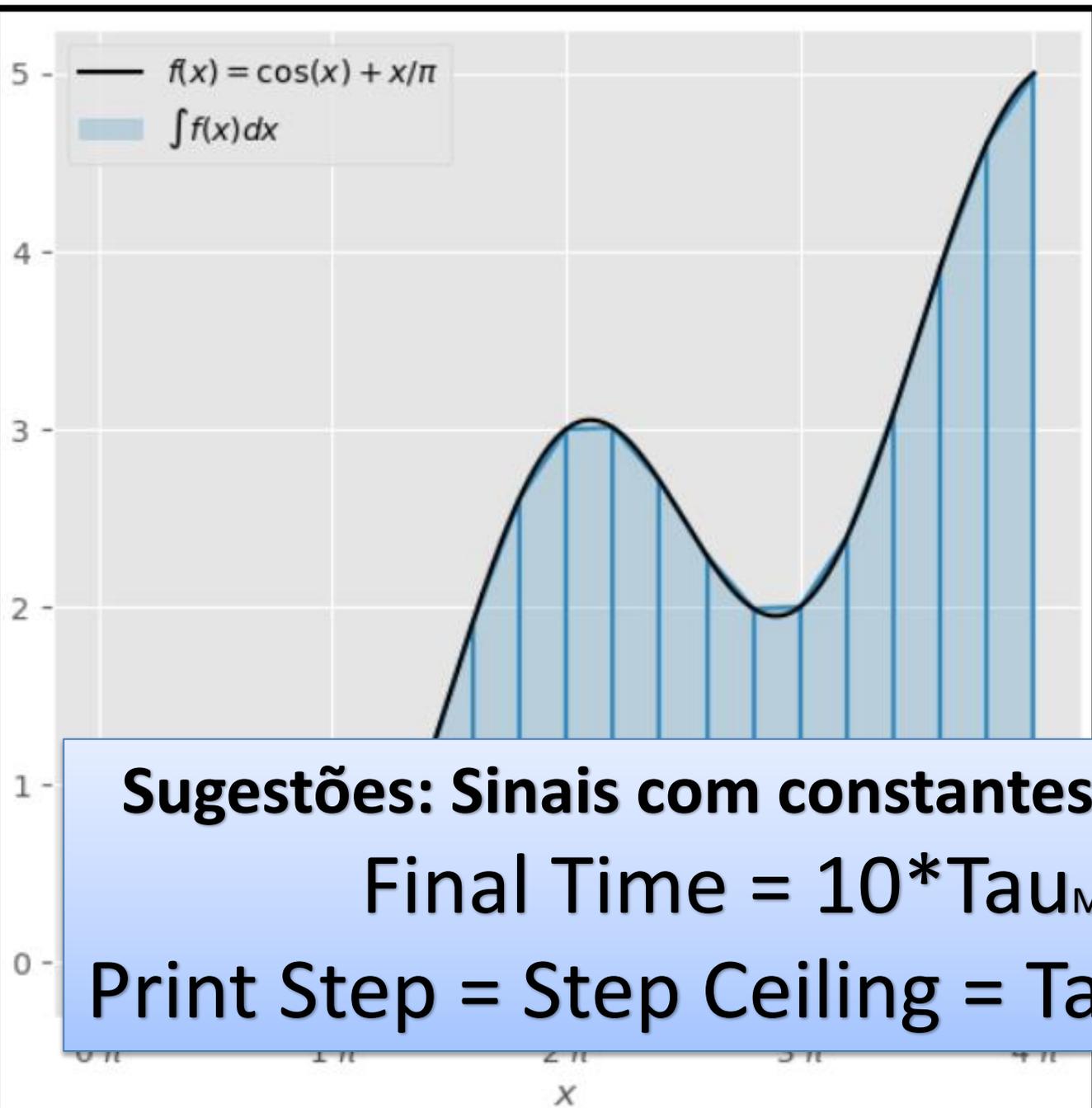
Output Vars.:

OK Cancel



**Quanto menor o passo de integração melhor representado estará o sinal!
Entretanto, passos muito reduzidos resultam em simulações demoradas!**





Transient

Transient Analysis

Print Step: 0.01 s

Final Time: 1 s

No-Print Delay:

Step Ceiling: 0.01 s

Detailed Bias Pt.

Skip initial transient solution

Fourier Analysis

Enable Fourier

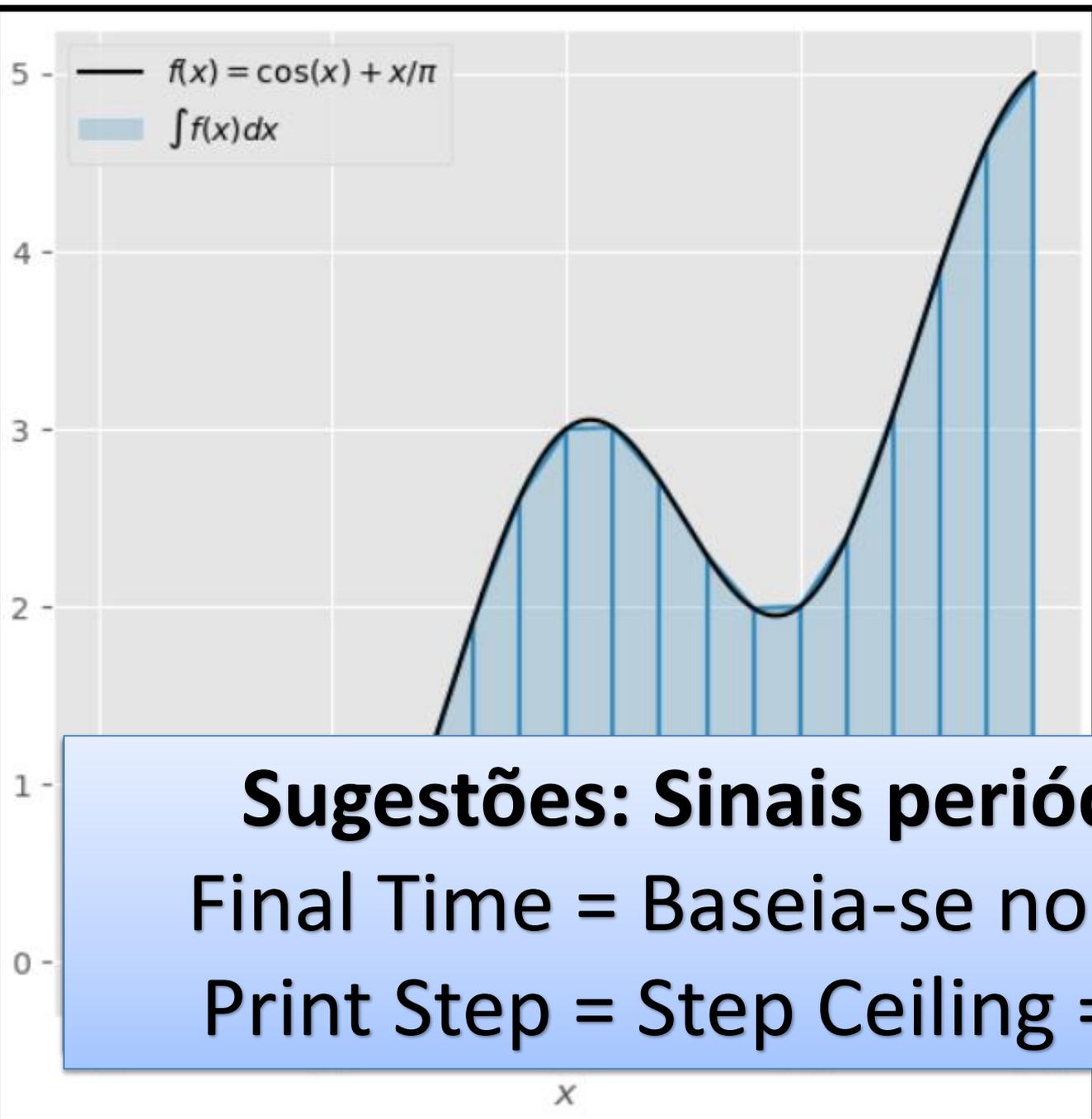
Center Frequency:

Number of harmonics:

Output Vars.:

OK Cancel

Sugestões: Sinais com constantes de tempo?
Final Time = 10* τ_{MAIOR}
Print Step = Step Ceiling = $\tau_{\text{MENOR}}/100$



Transient

Transient Analysis

Print Step: 0.01s

Final Time: 1s

No-Print Delay:

Step Ceiling: 0.01s

Detailed Bias Pt.

Skip initial transient solution

Fourier Analysis

Enable Fourier

Center Frequency:

Number of harmonics:

Output Vars.:

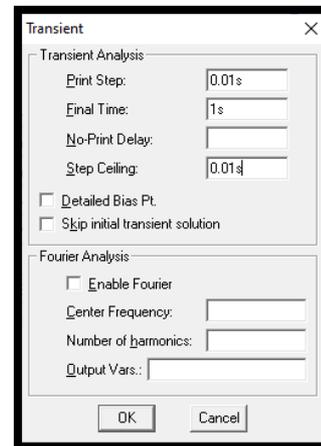
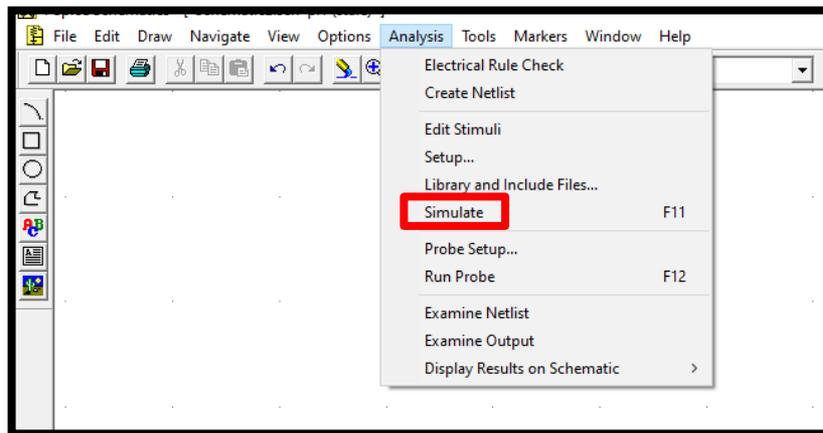
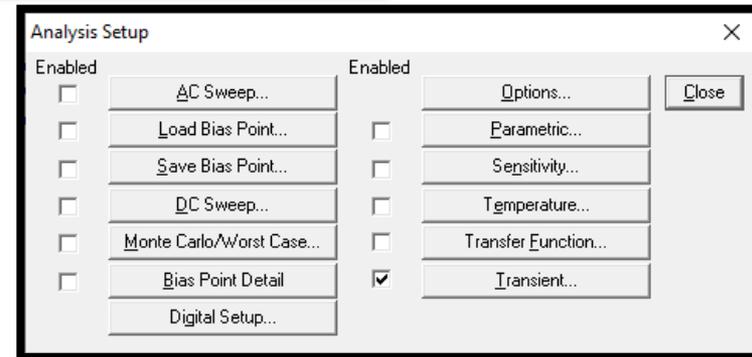
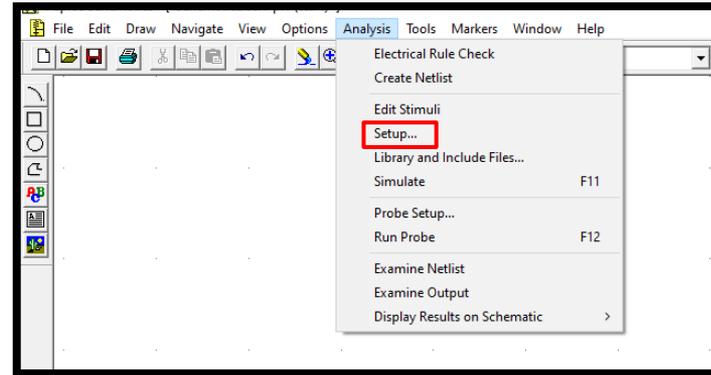
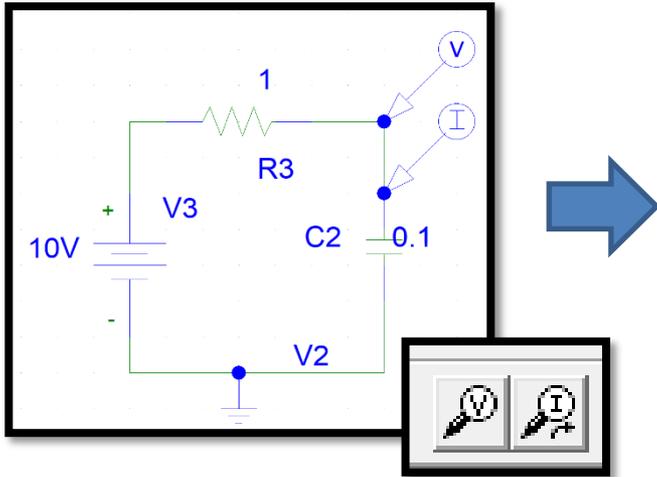
OK Cancel

Sugestões: Sinais periódicos?

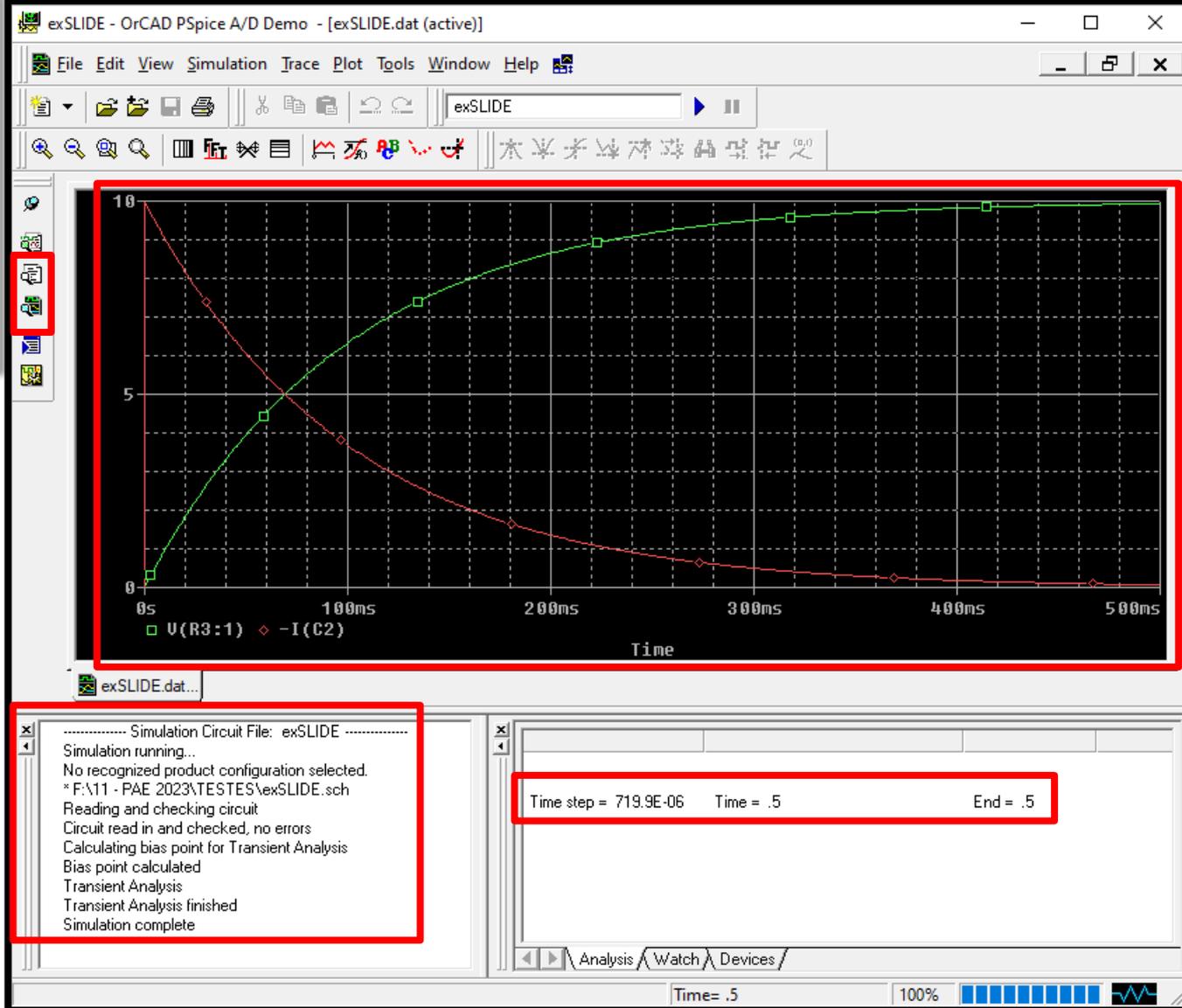
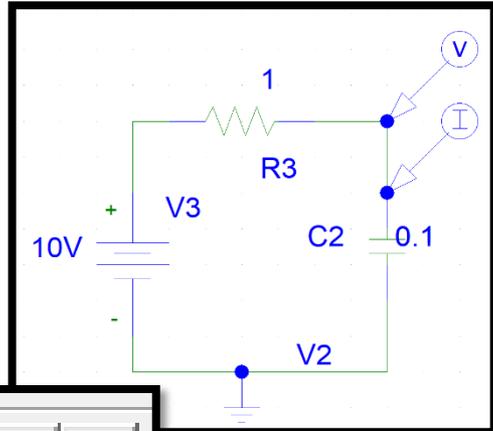
Final Time = Baseia-se no T e Tau

Print Step = Step Ceiling = T/100

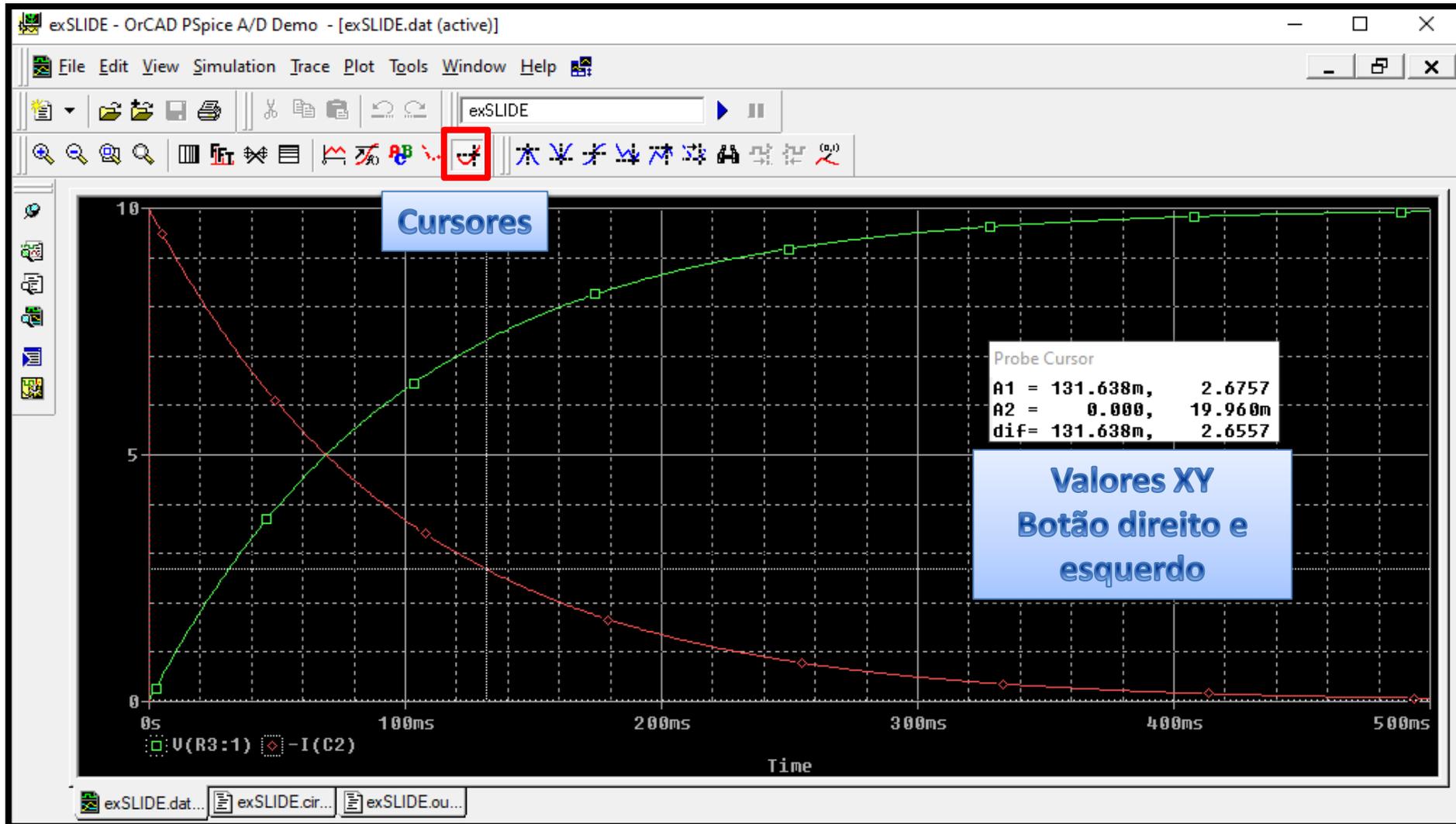
(Parte II) Tipos de Análises



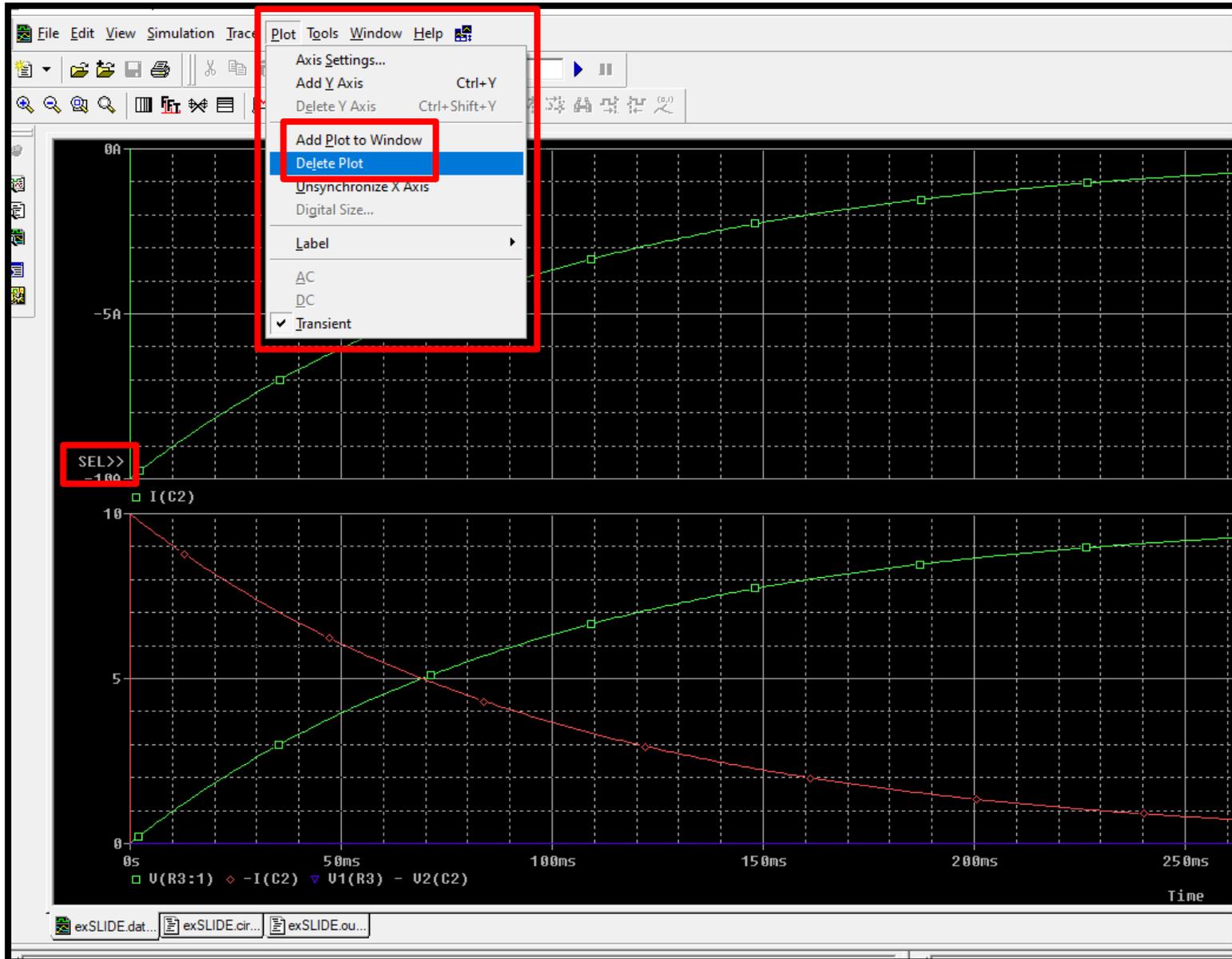
(Parte II) Tipos de Análises



(Parte II) Tipos de Análises



(Parte II) Tipos de Análises



(Parte II) Tipos de Análises

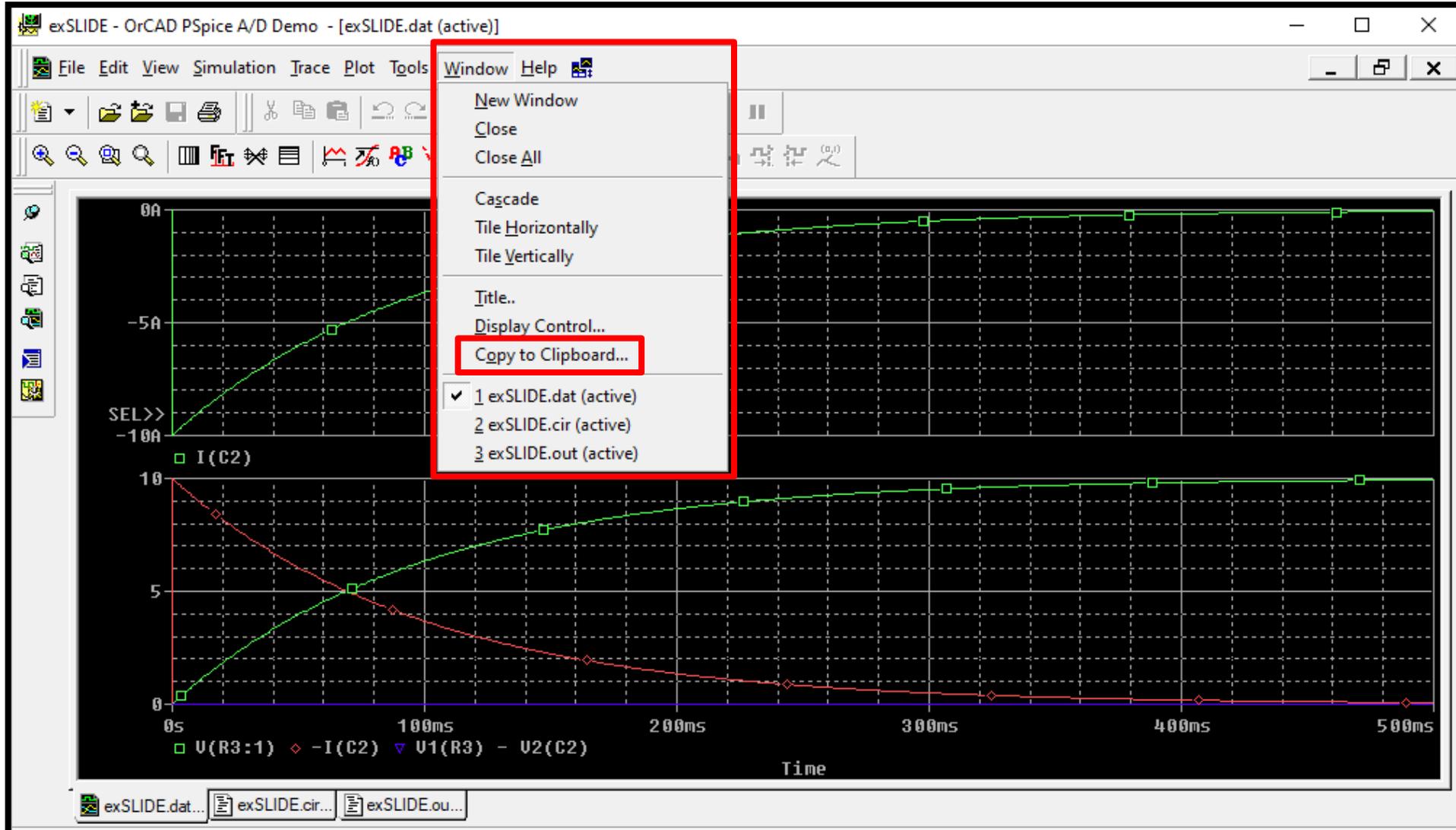
The image shows a screenshot of the OrCAD PSpice A/D Demo software interface. The main window displays a simulation plot with a blue box containing the text "Adicionar novas curvas" (Add new curves). The plot shows current $I(C2)$ and voltage $U(R3:1)$ over time. The $I(C2)$ trace is a green line starting at 0 and increasing linearly. The $U(R3:1)$ trace is a red line starting at 10 and decreasing linearly. The x-axis is labeled "Time" and has markers at 0s, 100ms, and 500ms. The y-axis has markers at 0, 5, 10, -5, and 0A.

The "Add Traces" dialog box is open, showing a list of simulation output variables. The variable $I(C2)$ is selected. The dialog box has the following sections:

- Simulation Output Variables:** A list of variables including $I(C2)$, $I(R3)$, $I(V3)$, Time, $V(\$N_0001)$, $V(\$N_0002)$, $V(0)$, $V(C2:1)$, $V(C2:2)$, $V(R3:1)$, $V(R3:2)$, $V(V2)$, $V(V3+)$, $V(V3-)$, $V1(R3)$, $V1(V3)$, $V2(C2)$, and $V2(R3)$.
- Functions or Macros:** A list of mathematical functions including $\#$, $()$, $*$, $+$, $-$, $/$, $@$, $ABS()$, $ARCTAN()$, $ATAN()$, $AVG()$, $AVGX(.)$, $AVGY(.)$, $COS()$, $D()$, $DB()$, $ENVMAX(.)$, $ENVMIN(.)$, $EXP()$, $G()$, $IMG()$, $LOG()$, $LOG10()$, $M()$, and $MAX()$.
- Checkboxes:** Analog, Digital, Voltages, Currents, Noise (V^2/Hz), Alias Names, Subcircuit Nodes.
- Trace Expression:** A text field for entering a custom expression.

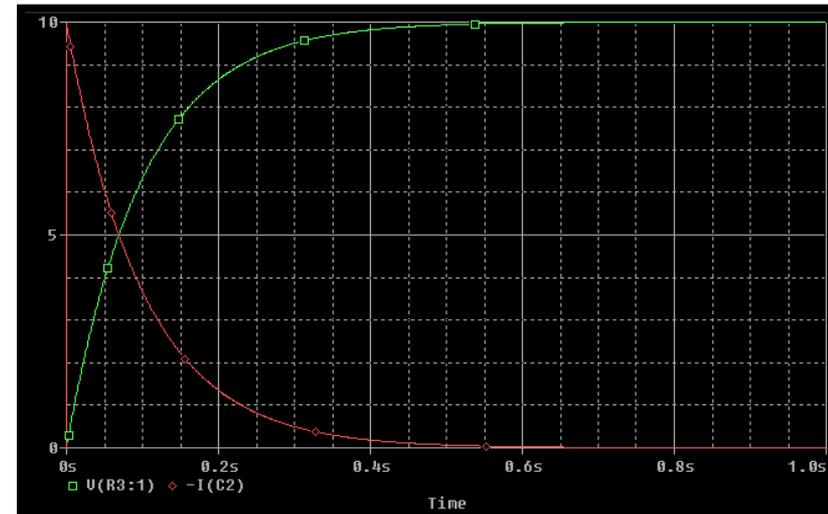
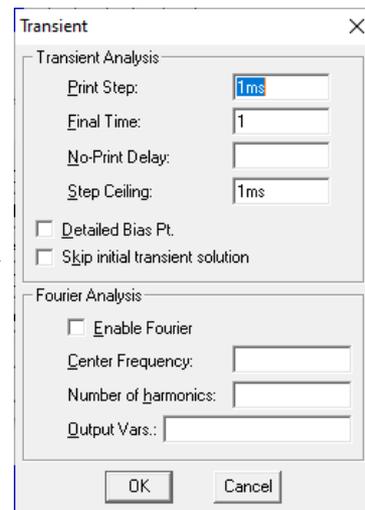
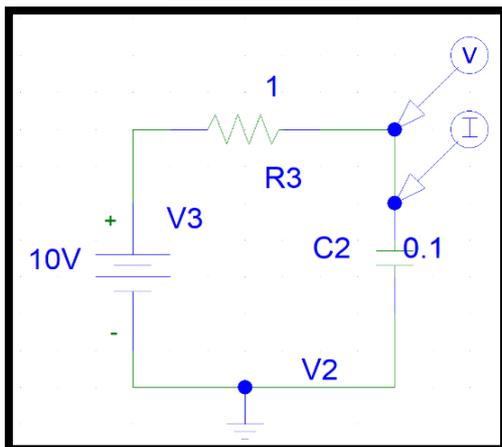
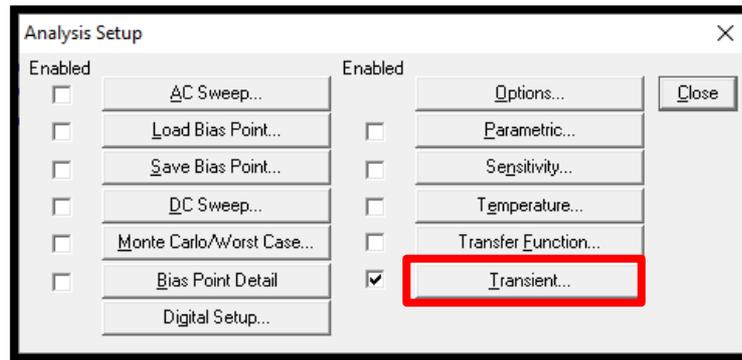
The status bar at the bottom shows the file names: exSLIDE.dat..., exSLIDE.cir..., and exSLIDE.ou...

(Parte II) Tipos de Análises



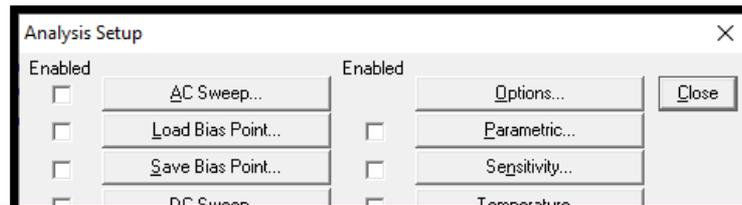
(Parte II) Tipos de Análises

- ✓ Análises de Transitórios
- ✓ Análises onde verificar as oscilografias é importante.
- ✓ Chamada pelo PSPICE de “Transient...”

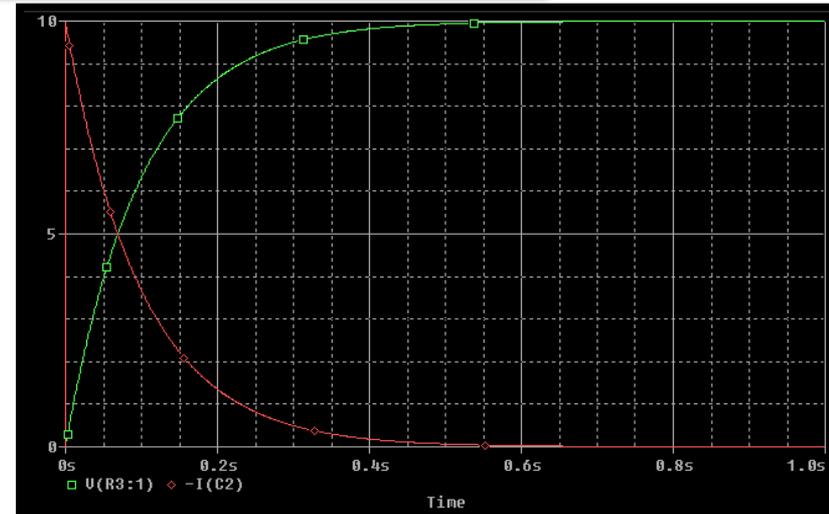
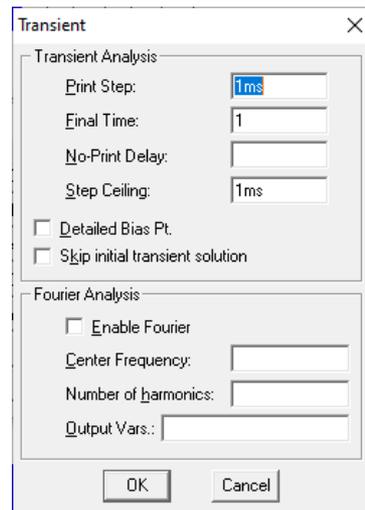
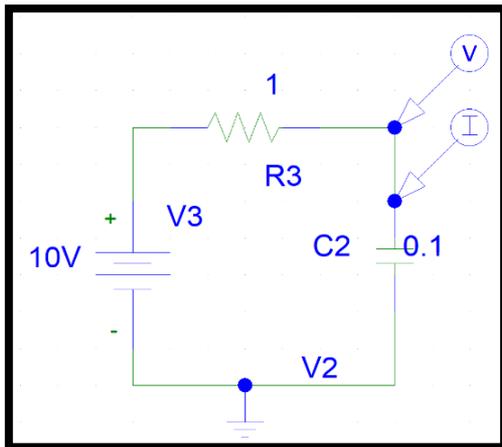


(Parte II) Tipos de Análises

- ✓ Análises de Transitórios
- ✓ Análises onde verificar as oscilografias é importante.
- ✓ Chamada pelo PSPICE de “Transient...”

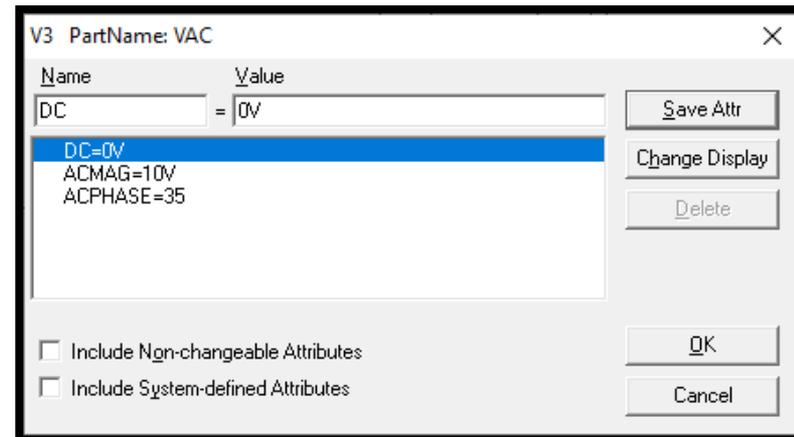
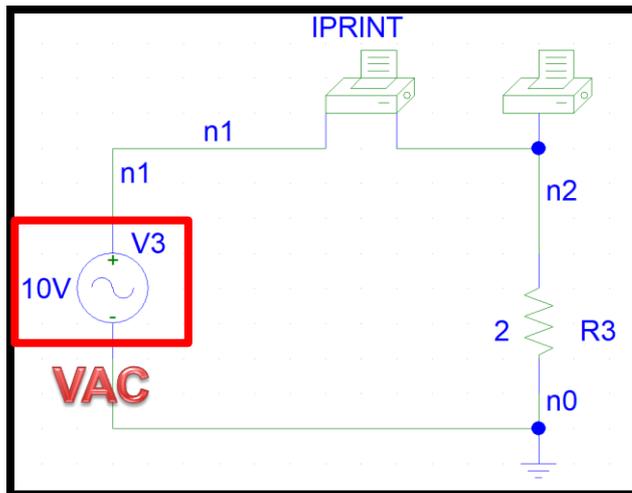
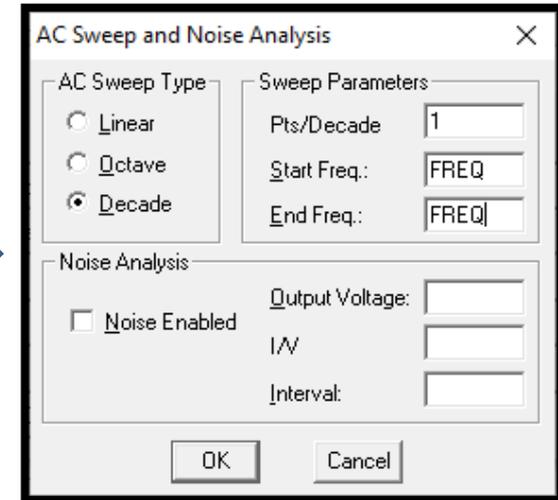
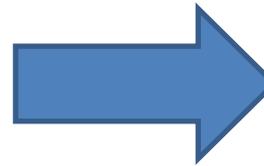
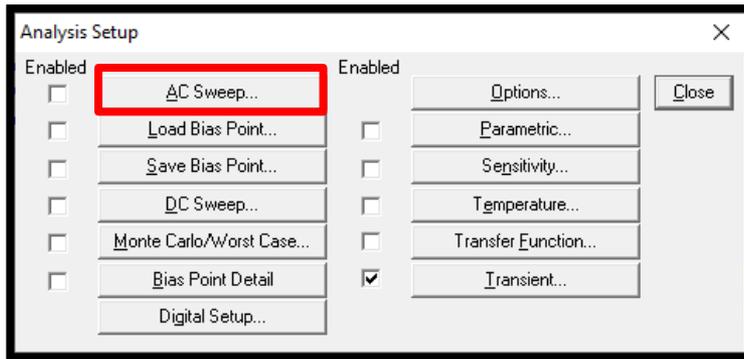


??? Exercício 02 ???



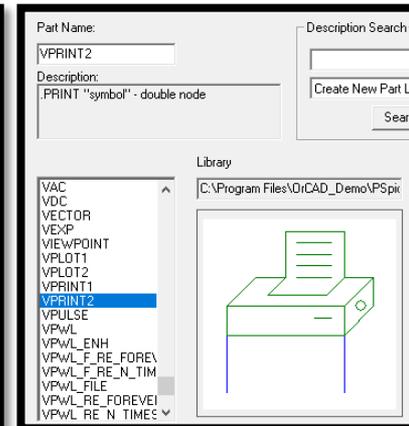
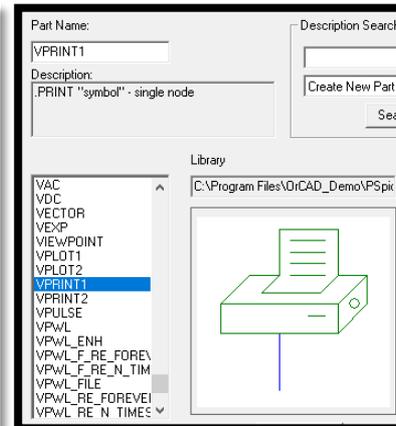
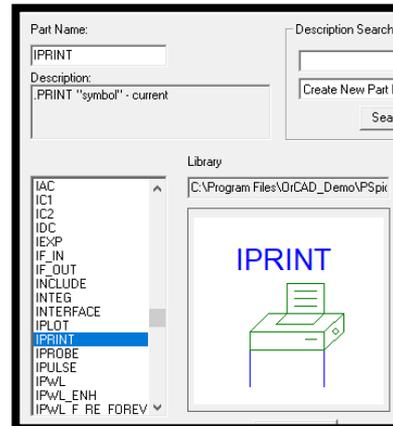
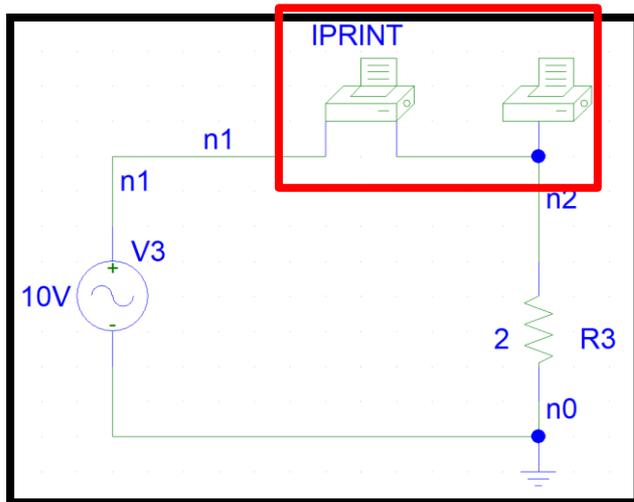
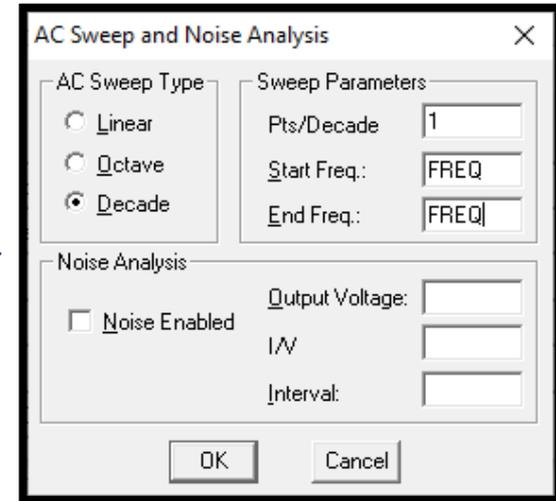
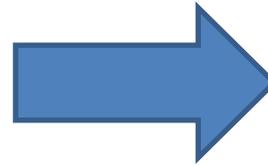
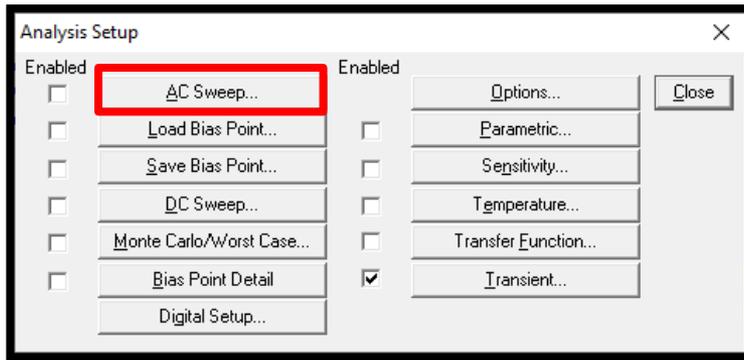
(Parte II) Tipos de Análises

- ✓ Análises Fasorial de Circuitos com Fontes AC
- ✓ Análises onde a obtenção dos fasores é importante.
- ✓ Chamada pelo PSPICE de “AC Sweep...”



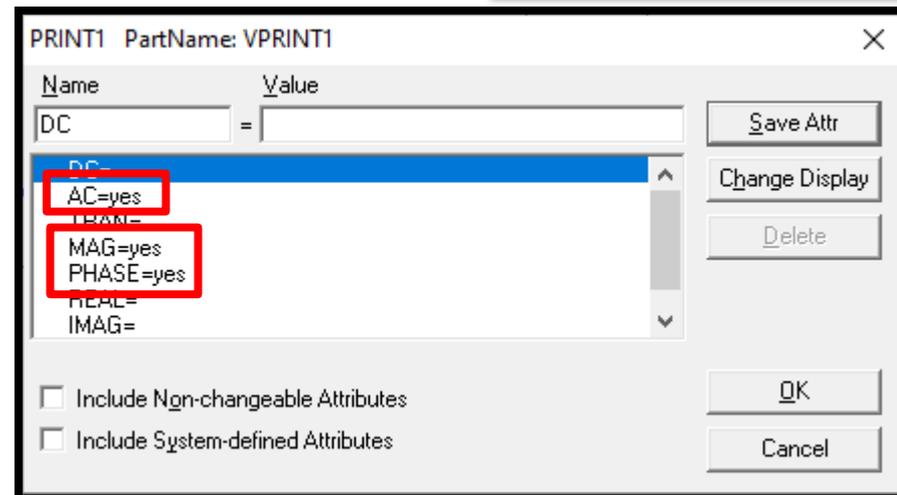
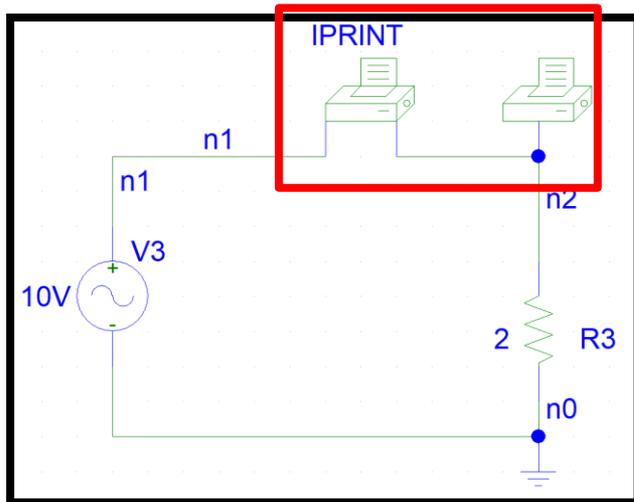
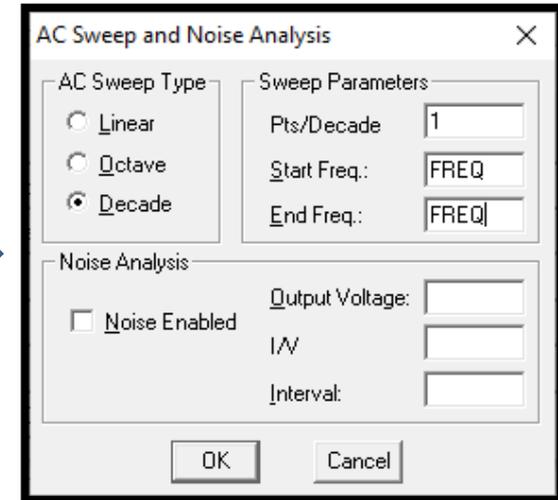
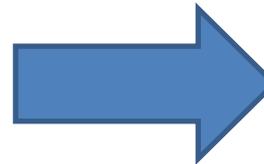
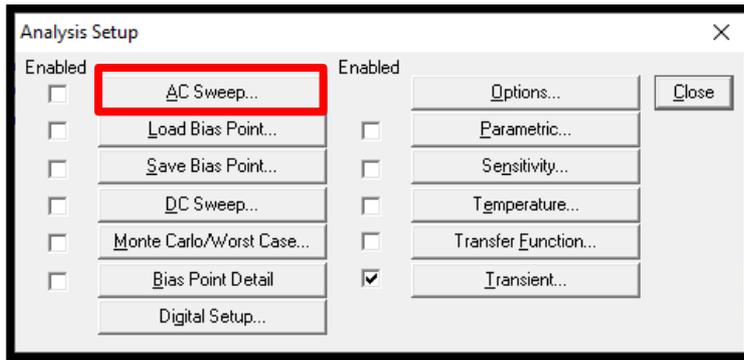
(Parte II) Tipos de Análises

- ✓ Análises Fasorial de Circuitos com Fontes AC
- ✓ Análises onde a obtenção dos fasores é importante.
- ✓ Chamada pelo PSPICE de “AC Sweep...”



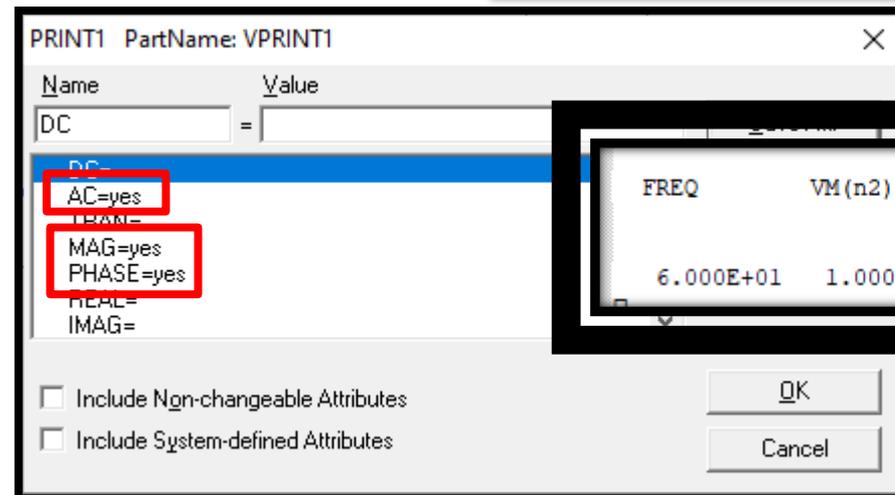
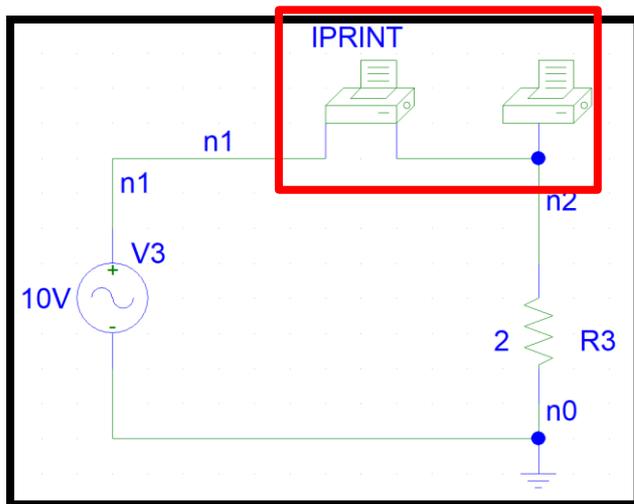
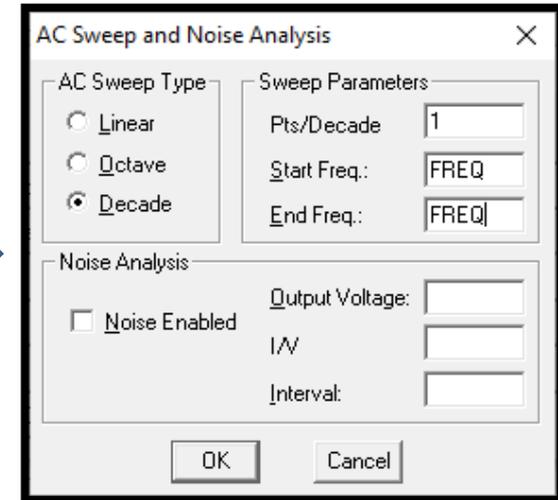
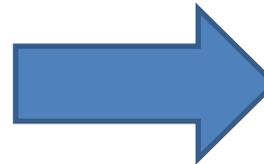
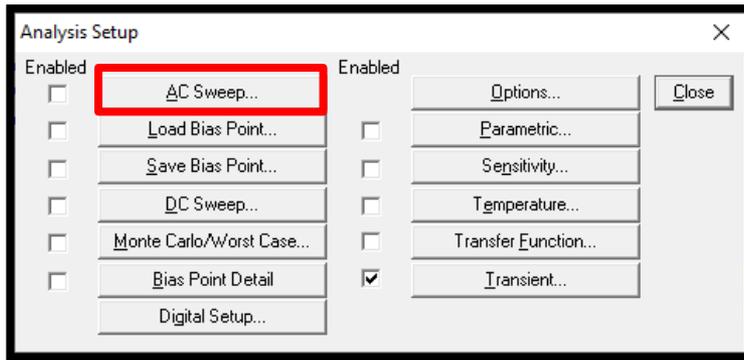
(Parte II) Tipos de Análises

- ✓ Análises Fasorial de Circuitos com Fontes AC
- ✓ Análises onde a obtenção dos fasores é importante.
- ✓ Chamada pelo PSPICE de “AC Sweep...”



(Parte II) Tipos de Análises

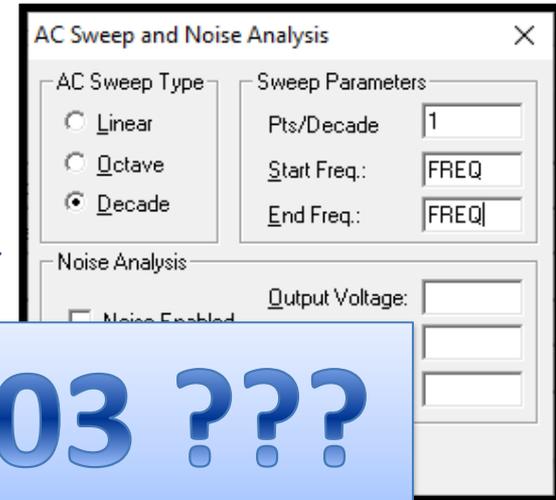
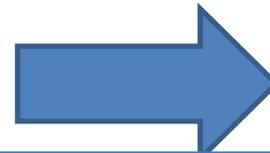
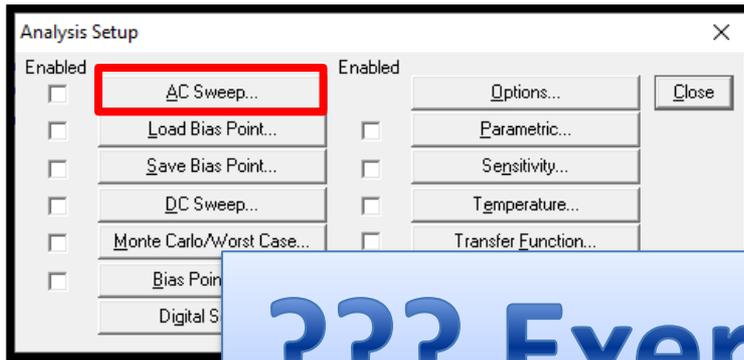
- ✓ Análises Fasorial de Circuitos com Fontes AC
- ✓ Análises onde a obtenção dos fasores é importante.
- ✓ Chamada pelo PSPICE de “AC Sweep...”



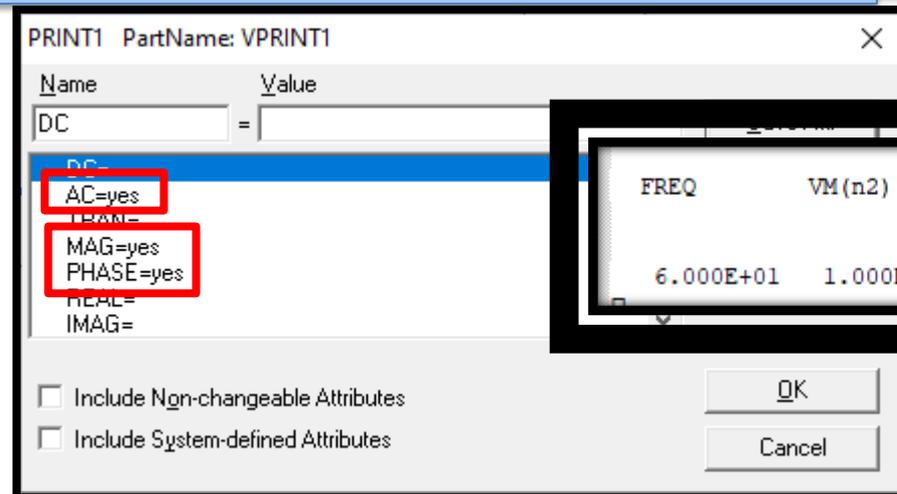
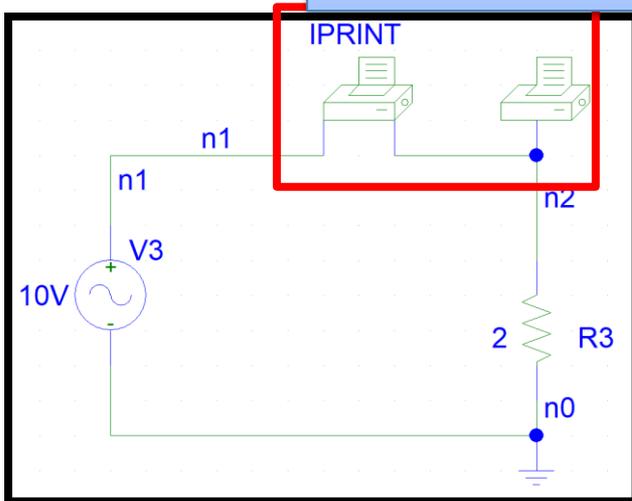
FREQ	VM(n2)	VP(n2)
6.000E+01	1.000E+01	3.500E+01

(Parte II) Tipos de Análises

- ✓ Análises Fasorial de Circuitos com Fontes AC
- ✓ Análises onde a obtenção dos fasores é importante.
- ✓ Chamada pelo PSPICE de “AC Sweep...”



??? Exercício 03 ???



FREQ	VM(n2)	VP(n2)
6.000E+01	1.000E+01	3.500E+01

(Parte II) Etapas da Simulação

Desenhar o
circuito a ser
simulado

Parametrizar
todos os
componentes
do circuito
**(inserir ponto
de
referência)**

Inserir
componentes
para saída de
resultados
(medidores)

Definir o tipo
de simulação
e
configurações
da mesma

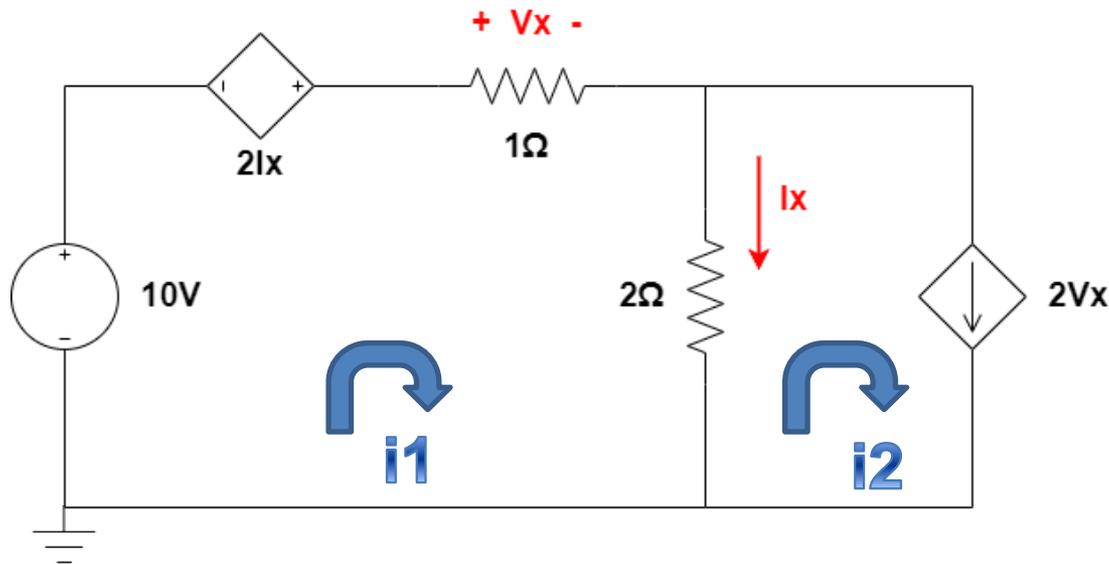
Executar a
simulação

Analisar e
documentar
os resultados

(Parte III) Exemplos

Exemplo 1:

Para o circuito abaixo, determine a tensão V_x e a corrente I_x .



$$10 + 2Ix - Vx - 2Ix = 0 \rightarrow Vx = 10V$$

$$i2 = 2Vx = 20A$$

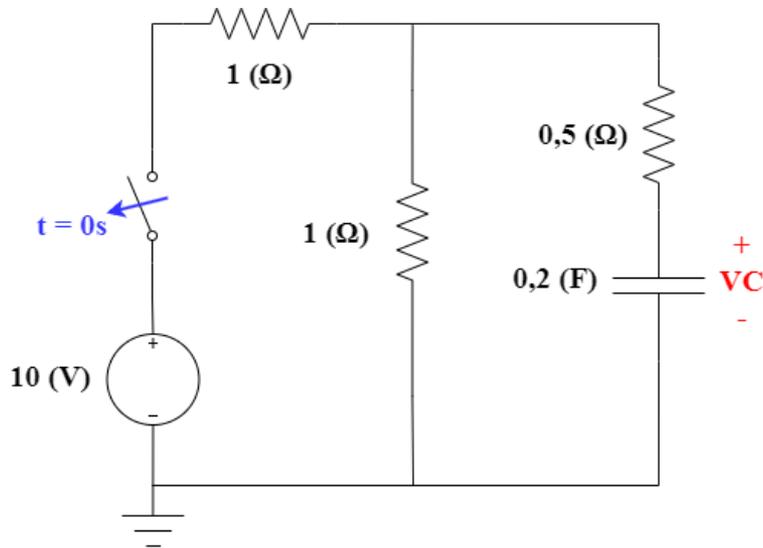
$$i1 * 1 = Vx = 10A$$

$$Ix = i1 - i2 \rightarrow Ix = -10A$$

(Parte III) Exemplos

Exemplo 2:

Para o circuito abaixo calcule e $VC(t)$ para $t > 0$, considerando as condições iniciais do circuito.



Para $t = 0^-s$:

$$VC(0) = 5V$$

Para $t > 0s$:

$$\text{Constante } RC = 0,2 * 1,5 = 0,3\ s$$

$$\text{Logo: } VC(t) = VC(0)e^{-t/RC}$$

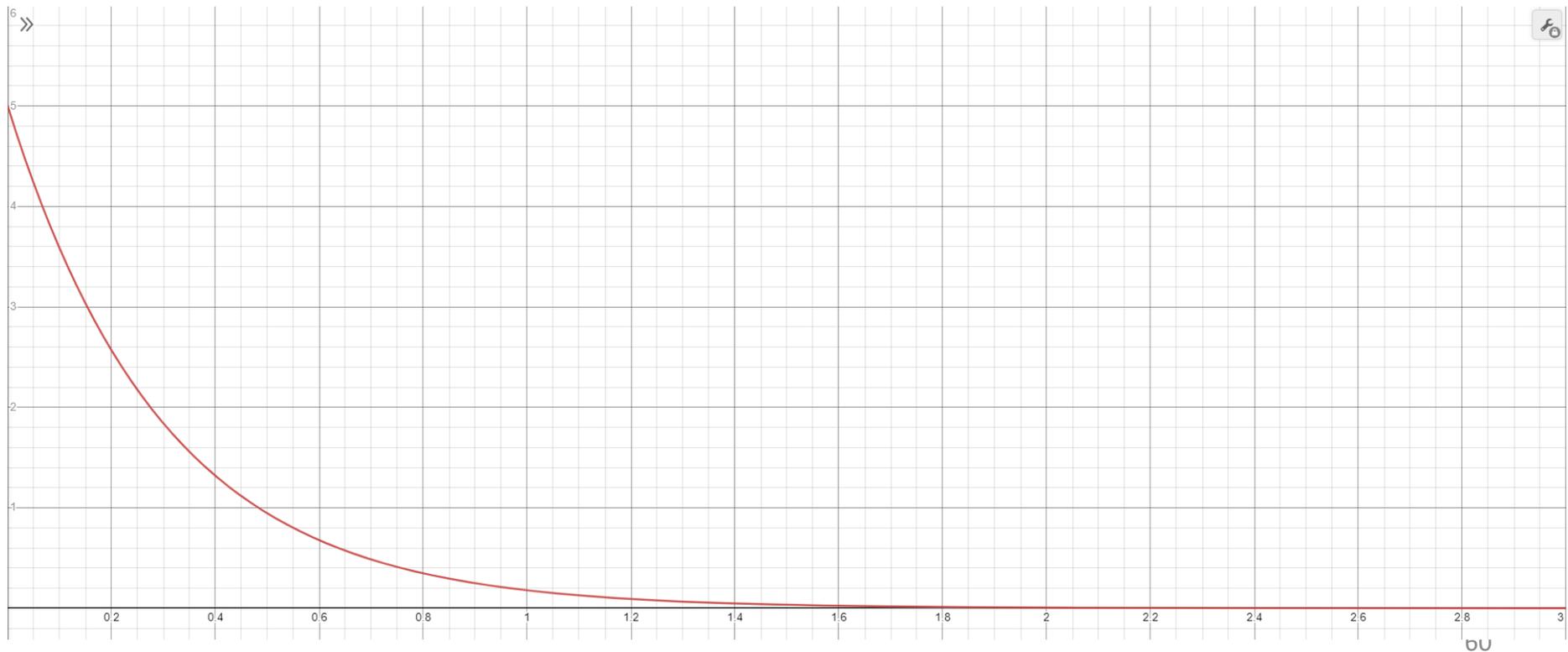
$$\text{Logo: } VC(t) = 5e^{-t/0,3}$$

(Parte III) Exemplos

Exemplo 2:

Para o circuito abaixo calcule $VC(t)$ para $t > 0$, considerando as condições iniciais do circuito.

Logo: $VC(t) = 5e^{-t/0,3}$



(Parte III) Material de Apoio

JOHNSON, David E., *et. al. Fundamentos de Análise de Circuitos Elétricos*, 4a ed.
Editora LTC. Rio de Janeiro, 2000.

PSPICE User's Guide:

<https://www.electronics-lab.com/wp-content/uploads/2015/08/PSPICE.pdf>

PSPICE Schematics User's Guide:

<http://orion.ipt.pt/~jorge/Electronica1/PSPICE/Manuais/scug.pdf>

Vídeo tutoriais:

<https://www.youtube.com/watch?v=eBGepKS18L0&list=PL0IfaY4kueVrZLGUuqzBTQHD1he87kwyu>

Introdução à Análise de Circuitos Elétricos Utilizando “PSPICE”

Atendimento:

Sexta-feira: 08:10 às 09:00h

Datas importantes:

Exercício 1 – 14/04 até 28/04

Moisés J. B. B. Davi
moisesdavi@usp.br