























General

 Project	 Config	 New sch	 New Spice	 Save	 BW	 FW	 Schematic	 NetList	 Run	 View
Circuit-1.sch							Simulation-1.sp			
Circuit-2.sch							Simulation-2.sp			
Circuit-3.sch							Simulation-3.sp			

Simulation-3.sp		Suggestions/Results	
1	.ic v(out)=3	<p>tran: permite iniciar una simulación transitoria.</p> <p>Uso: tran <paso> <final> <inicial></p> <p>Ejemplo: tran 1u 1m 0</p>	
2	.control		
3	tran 1u 6m 0 uic		
4	write out/tran1.raw v(out) 0		
5	rusage everything		
6	.endc		
7			
...			

Circuit-2.net	
1	*===== Begin SPICE netlist of
2	main design =====
3	C1 out 0 1u
4	R1 0 out 1k
5	.end
6	
7	
...	

menu-2

 Project		 Config		 New sch		 New Spice		 Save		 BW		 FW		 Schematic		 NetList		 Run		 View	
Circuit-1.sch												Simulation-1.sp									
Circuit-2.sch												Simulation-2.sp									
Circuit-3.sch												Simulation-3.sp									

Simulation-3.sp		Suggestions/Results	
1	.ic v(out)=3	<p>tran: permite iniciar una simulación transitoria.</p> <p>Uso: tran <paso> <final> <inicial></p> <p>Ejemplo: tran 1u 1m 0</p>	
2	.control		
3	tran 1u 6m 0 uic		
4	write out/tran1.raw v(out) 0		
5	rusage everything		
6	.endc		
7			
...			

Circuit-2.net	
1	*===== Begin SPICE netlist of
2	main design =====
3	C1 out 0 1u
4	R1 0 out 1k
5	.end
6	
7	
...	

New sch

The screenshot shows the main interface of a circuit simulation tool. At the top is a toolbar with icons for Project, Config, New sch, New Spice, Save, BW, FW, Schematic, NetList, Run, and View. Below the toolbar is a file list on the left containing 'Circuit-1.sch', 'Circuit-2.sch', 'Circuit-3.sch', and 'new-schematic.sch'. The 'new-schematic.sch' file is selected. To the right of the file list is a panel with a tab labeled 'new-spice.sp' and a 'Suggestions/Results' section. The 'Suggestions/Results' section contains text explaining the 'tran' command: 'tran: permite iniciar una simulación transitoria.', 'Uso: tran <paso> <final> <inicial>', and 'Ejemplo: tran 1u 1m 0'. A 'New schematic' dialog box is open on the right, with the title 'New schematic' and a text input field containing 'new-schematic.sch'. The dialog has 'Cancel' and 'Ok' buttons. Arrows point from the 'New sch' button in the toolbar to the 'new-schematic.sch' file in the file list, from the 'new-spice.sp' tab to the 'Suggestions/Results' section, and from the 'Ok' button in the dialog box to the 'new-schematic.sch' file.

Project Config New sch New Spice Save BW FW Schematic NetList Run View

Circuit-1.sch
Circuit-2.sch
Circuit-3.sch
new-schematic.sch

new-spice.sp Suggestions/Results

1
2
3
4
5
6
7
...

tran: permite iniciar una simulación transitoria.
Uso:
tran <paso> <final>
<inicial>
Ejemplo:
tran 1u 1m 0

Circuit-2.net

1 *===== Begin SPICE netlist of
2 main design =====
3 C1 out 0 1u
4 R1 0 out 1k
5 .end
6
7
...

New schematic












Inserta el nombre del archivo con la extensión.sch

new-schematic.sch

Cancel Ok

Spices vacios por ser nuevo esquemático

menu-3

 Project	 Config	 New sch	 New Spice	 Save	 BW	 FW	 Schematic	 NetList	 Run	 View
Circuit-1.sch						Simulation-1.sp				
Circuit-2.sch						Simulation-2.sp				
Circuit-3.sch						Simulation-3.sp				

Simulation-3.sp		Suggestions/Results	
1	.ic v(out)=3	<p>tran: permite iniciar una simulación transitoria.</p> <p>Uso: tran <paso> <final> <inicial></p> <p>Ejemplo: tran 1u 1m 0</p>	
2	.control		
3	tran 1u 6m 0 uic		
4	write out/tran1.raw v(out) 0		
5	rusage everything		
6	.endc		
7			
...			

Circuit-2.net	
1	*===== Begin SPICE netlist of
2	main design =====
3	C1 out 0 1u
4	R1 0 out 1k
5	.end
6	
7	
...	

new-spice

The screenshot displays a SPICE simulation software interface. At the top is a toolbar with icons for Project, Config, New sch, New Spice, Save, BW, FW, Schematic, NetList, Run, and View. Below the toolbar is a file list on the left with entries: Circuit-1.sch, Circuit-2.sch, Circuit-3.sch, Simulation-1.sp, Simulation-2.sp, Simulation-3.sp, and new-spice.sp. The 'new-spice.sp' file is selected. To the right of the file list is a pane titled 'Suggestions/Results' containing text about the 'tran' command: 'tran: permite iniciar una simulación transitoria.', 'Uso: tran <paso> <final> <inicial>', and 'Ejemplo: tran lu 1m 0'. Below the file list is a text editor showing the contents of 'Circuit-2.net', which is a SPICE netlist starting with '*===== Begin SPICE netlist of' and ending with '.end'. A 'New spice' dialog box is open on the right, with the title 'New spice' and a prompt 'Inserta el nombre del archivo con la extensión.sp'. The text 'new-spice.sp' is entered in the input field. The dialog has 'Cancel' and 'Ok' buttons. Arrows indicate the flow from the file list to the dialog and from the 'Suggestions/Results' pane to the dialog.

Project Config New sch New Spice Save BW FW Schematic NetList Run View

Circuit-1.sch
Circuit-2.sch
Circuit-3.sch
Simulation-1.sp
Simulation-2.sp
Simulation-3.sp
new-spice.sp

new-spice.sp Suggestions/Results

tran: permite iniciar una simulación transitoria.

Uso:
tran <paso> <final>
<inicial>

Ejemplo:
tran lu 1m 0

Circuit-2.net

*===== Begin SPICE netlist of
main design =====
C1 out 0 1u
R1 0 out 1k
.end












New spice

Inserta el nombre del archivo con la extensión.sp

new-spice.sp

Cancel Ok

menu-4

 Project	 Config	 New sch	 New Spice	 Save	 BW	 FW	 Schematic	 NetList	 Run	 View
Circuit-1.sch							Simulation-1.sp			
Circuit-2.sch							Simulation-2.sp			
Circuit-3.sch							Simulation-3.sp			

Simulation-3.sp		Suggestions/Results	
1	.ic v(out)=3	<p>tran: permite iniciar una simulación transitoria.</p> <p>Uso: tran <paso> <final> <inicial></p> <p>Ejemplo: tran 1u 1m 0</p>	
2	.control		
3	tran 1u 6m 0 uic		
4	write out/tran1.raw v(out) 0		
5	rusage everything		
6	.endc		
7	...		

Circuit-2.net	
1	*===== Begin SPICE netlist of
2	main design =====
3	C1 out 0 1u
4	R1 0 out 1k
5	.end
6	
7	
...	

Schematic

Project

Config

New sch

New Spice

Save

BW

FW

Schematic

Netlist

Run

View

Circuit-1.sch

Circuit-2.sch

Circuit-3.sch

Simulation-1.sp

Simulation-2.sp

Simulation-3.sp

Simulation-3.sp

Suggestions/Results

1

2

3

4

5

6

7

...

.ic v(out)=3

.control

tran lu 6m 0 uic

write out/tran1.raw v(out) 0

rusage everything

.endc

tran: permite iniciar una simulación transitoria.

Uso:

tran <paso> <final>

<inicial>

Ejemplo:

tran lu 1m 0

Circuit-2.net

1

2

3

4

5

6

7

*===== Begin SPICE netlist of

main design =====

C1 out 0 lu

R1 0 out 1k

.end

lepton-eda Circuit-2.sch

NetList

The screenshot shows a software interface for circuit simulation. At the top is a menu bar with icons for Project, Config, New sch, New Spice, Save, BW, FW, Schematic, NetList, Run, and View. Below the menu bar are two panels: 'Circuit-1.sch' and 'Simulation-1.sp'. The 'Circuit-1.sch' panel contains three sub-panels: 'Circuit-1.sch', 'Circuit-2.sch', and 'Circuit-3.sch'. The 'Simulation-1.sp' panel contains three sub-panels: 'Simulation-1.sp', 'Simulation-2.sp', and 'Simulation-3.sp'. The 'Simulation-3.sp' panel is selected and shows the following text:

```
1 .ic v(out)=3
2 .control
3 tran 1u 6m 0 uic
4 write out/tran1.raw v(out) 0
5 rusage everything
6 .endc
7
...
```

Below the 'Simulation-3.sp' panel is a panel labeled 'Circuit-2.net' showing the following text:

```
1 *===== Begin SPICE netlist of
2 main design =====
3 C1 out 0 1u
4 R1 0 out 1k
5 .end
6
7
...
```

On the right side of the interface is a panel labeled 'Suggestions/Results' containing the text: 'Resultado de la generación del netlist'. Two arrows point from the 'NetList' menu item to this panel.

Run

The screenshot shows a software interface for running SPICE simulations. At the top is a toolbar with icons for Project, Config, New sch, New Spice, Save, BW, FW, Schematic, NetList, Run, and View. Below the toolbar are two lists: 'Circuit-1.sch', 'Circuit-2.sch', and 'Circuit-3.sch' on the left; and 'Simulation-1.sp', 'Simulation-2.sp', and 'Simulation-3.sp' on the right. The 'Run' button is highlighted with a red flame icon. A curved arrow points from the 'Run' button to the 'Suggestions/Results' panel on the right. The 'Simulation-3.sp' panel is active, showing a list of lines (1-7) and a SPICE netlist. The 'Suggestions/Results' panel contains the text 'Se imprime el resultado de la simulación' with an arrow pointing to it from the 'Run' button. Below the simulation panel is the 'Circuit-2.net' panel, showing a list of lines (1-7) and a SPICE netlist.

Project Config New sch New Spice Save BW FW Schematic NetList Run View

Circuit-1.sch
Circuit-2.sch
Circuit-3.sch

Simulation-1.sp
Simulation-2.sp
Simulation-3.sp

Simulation-3.sp

```
1 .ic v(out)=3
2 .control
3 tran 1u 6m 0 uic
4 write out/tran1.raw v(out) 0
5 rusage everything
6 .endc
7
...
```

Suggestions/Results

Se imprime el resultado de la simulación

Circuit-2.net

```
1 *===== Begin SPICE netlist of
2 main design =====
3 C1 out 0 1u
4 R1 0 out 1k
5 .end
6
7
...
```

scope

