



ΕΛΛΗΝΙΚΗ ΔΗΜΟΚΡΑΤΙΑ  
Εθνικό και Καποδιστριακό  
Πανεπιστήμιο Αθηνών

HELLENIC REPUBLIC  
National and Kapodistrian  
University of Athens







# ΕΡΓΑΣΤΗΡΙΟ ΗΛΕΚΤΡΟΝΙΚΗΣ

*Μάθημα 1<sup>ο</sup> – Εισαγωγή,*



# Πληροφορίες – Επικοινωνία

Σελίδα μαθήματος: <http://users.uoa.gr/~matakias/>

		ΕΛΛΗΝΙΚΗ ΔΗΜΟΚΡΑΤΙΑ Εθνικό και Καποδιστριακό Πανεπιστήμιο Αθηνών	HELLENIC REPUBLIC National and Kapodistrian University of Athens		
Home Αρχική	Curriculum Vitae Βιογραφικό	Publications - Research Interests Δημοσιεύσεις - Ενδιαφέροντα		Projects	Μαθήματα
		<b>Μαθήματα</b>			
<a href="#">Sotirios Matakias</a>		• Teaching Assistant, (Φυσική εαρινού εξαμήνου 2008, 2009, 2010, 2011, 2012) <a href="#">Φυσική Β-εξαμήνου. (ασκήσεις)</a>			
Email: matakias at di.uoa.gr		• Διδασκαλία μαθήματος: <b>Σχεδίαση Τηλεπικοινωνιακών VLSI Κυκλωμάτων</b> Χειμερινό Εξάμηνο 2012-2013 Ωρες φοιτητών: Δευτέρα 15:00-16:00, Γραφείο: 110 Το μάθημα διδάσκεται στο ΠΜΣ του Τμήματος Πληροφορικής και Τηλεπικοινωνιών με τίτλο: "Σχεδίαση Τηλεπικοινωνιακών VLSI Κυκλωμάτων" και στο διατμηματικό ΠΜΣ Ηλεκτρονικής και Ραδιοηλεκτρολογίας με τίτλο: "Σχεδίαση Τηλεπικοινωνιακών Κυκλωμάτων και Συστημάτων".			
<b>Μάθημα</b>		<b>Αντικείμενο</b>		<b>Διαφάνειες</b>	
1ο		<a href="#">Εισαγωγή στη Σχεδίαση RF Κυκλωμάτων</a>			

[matakias@di.uoa.gr](mailto:matakias@di.uoa.gr)



# Εξοπλισμός Εργαστηρίου

- Πολύμετρο
- Τροφοδοτικό
- Παλμογράφος
- Γεννήτρια
- Breadboard



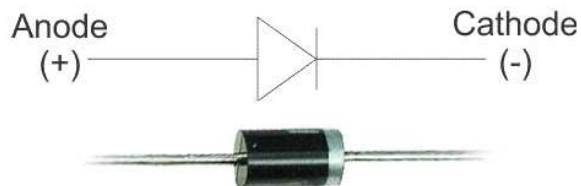
# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων



**Δίοδος Γερμανίου  
επαφής μετάλλου– ημιαγωγού**

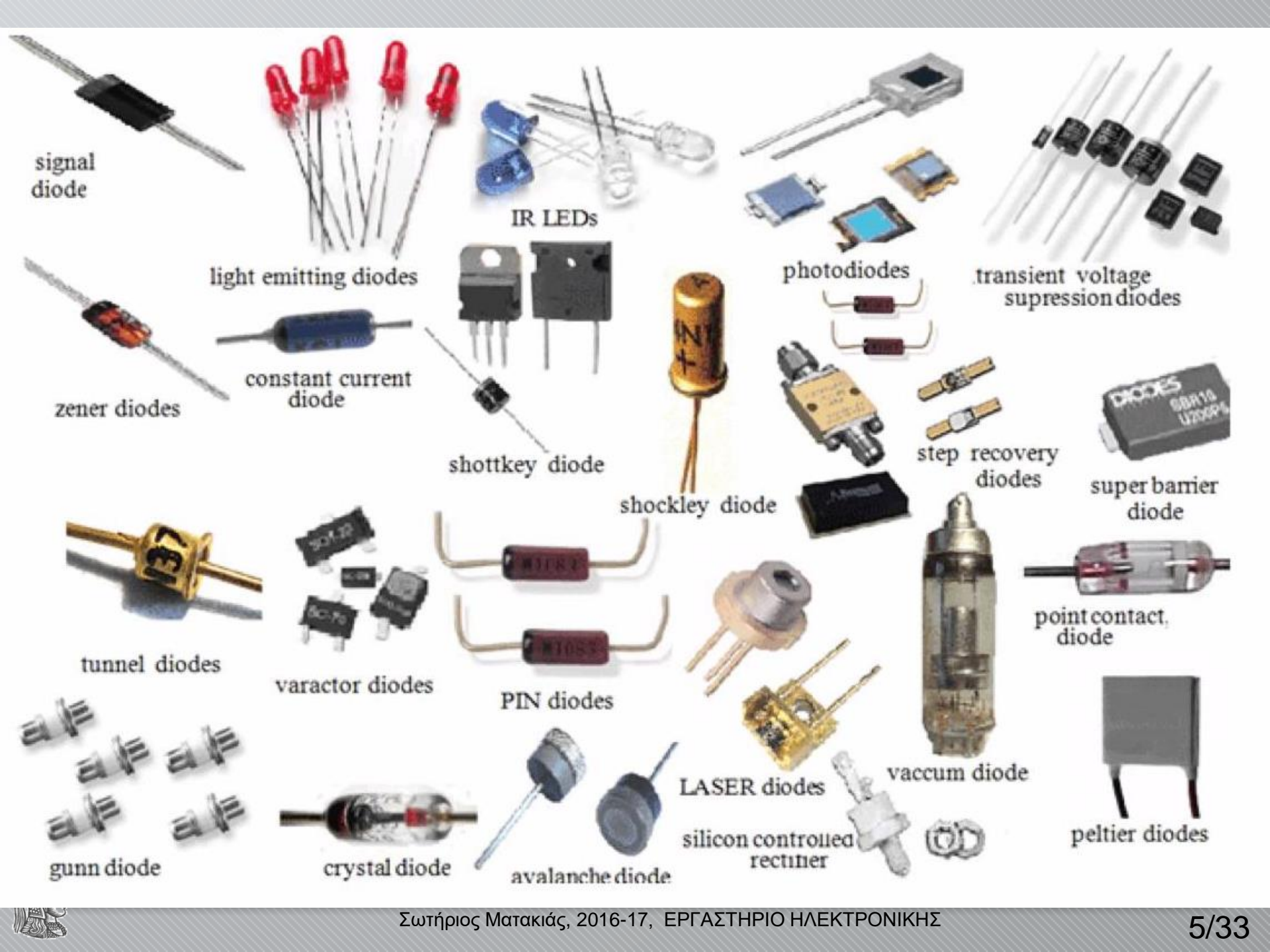


**Δίοδος Ανόρθωσης  
Πυριτίου (pn)**



**Συμβολισμός  
ακροδέκτες**





signal diode

light emitting diodes

IR LEDs

photodiodes

transient voltage suppression diodes

zener diodes

constant current diode

shottkey diode

shockley diode

step recovery diodes

super barrier diode

tunnel diodes

varactor diodes

PIN diodes

point contact diode

gunn diode

crystal diode

avalanche diode

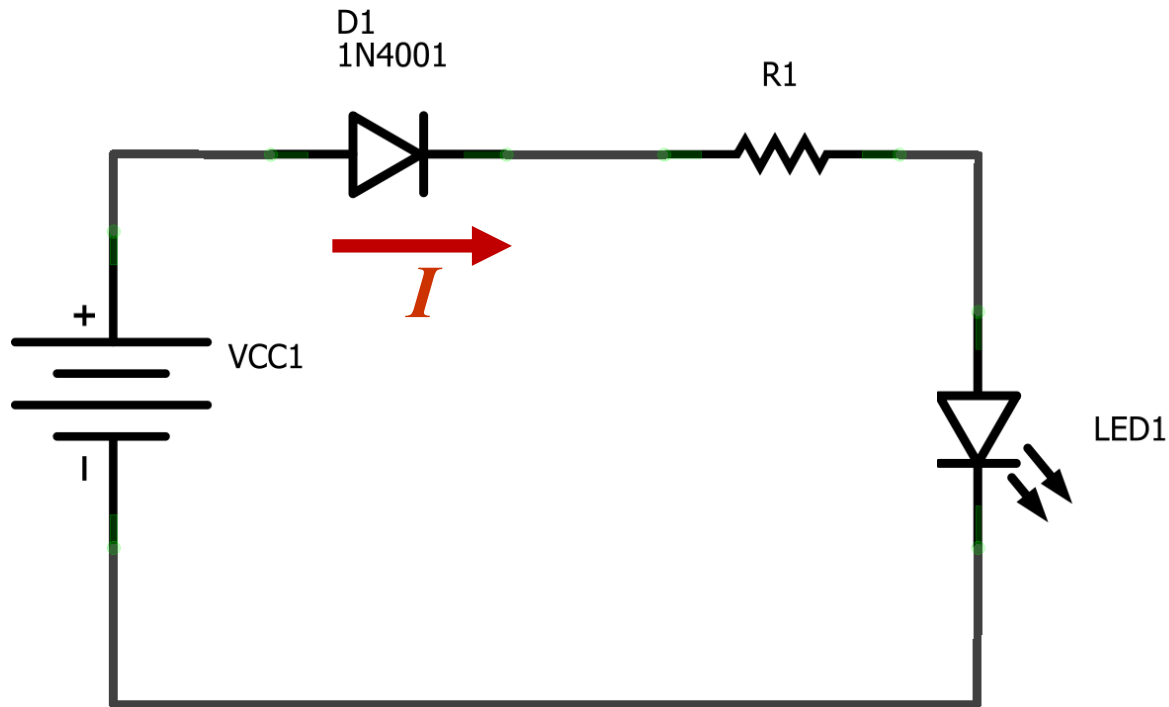
LASER diodes

vaccum diode

peltier diodes

silicon controlled rectifier

# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων

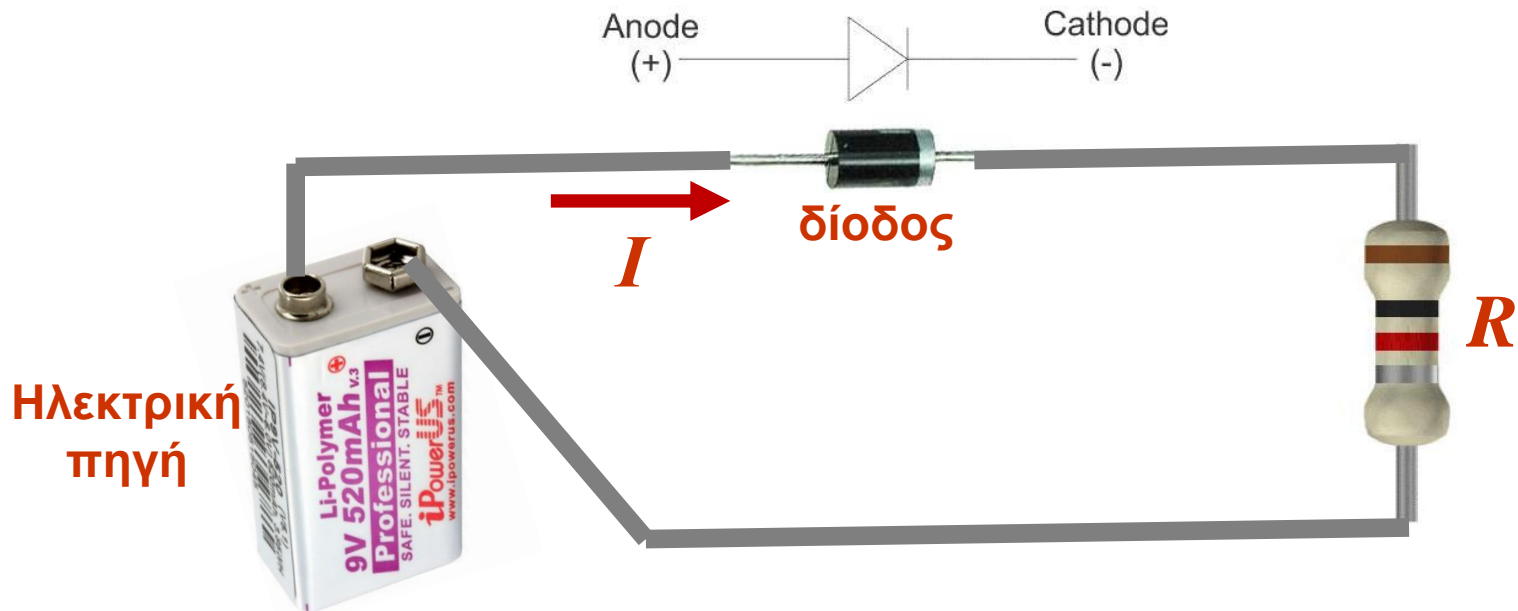


**$I$  = Ρεύμα διόδου**

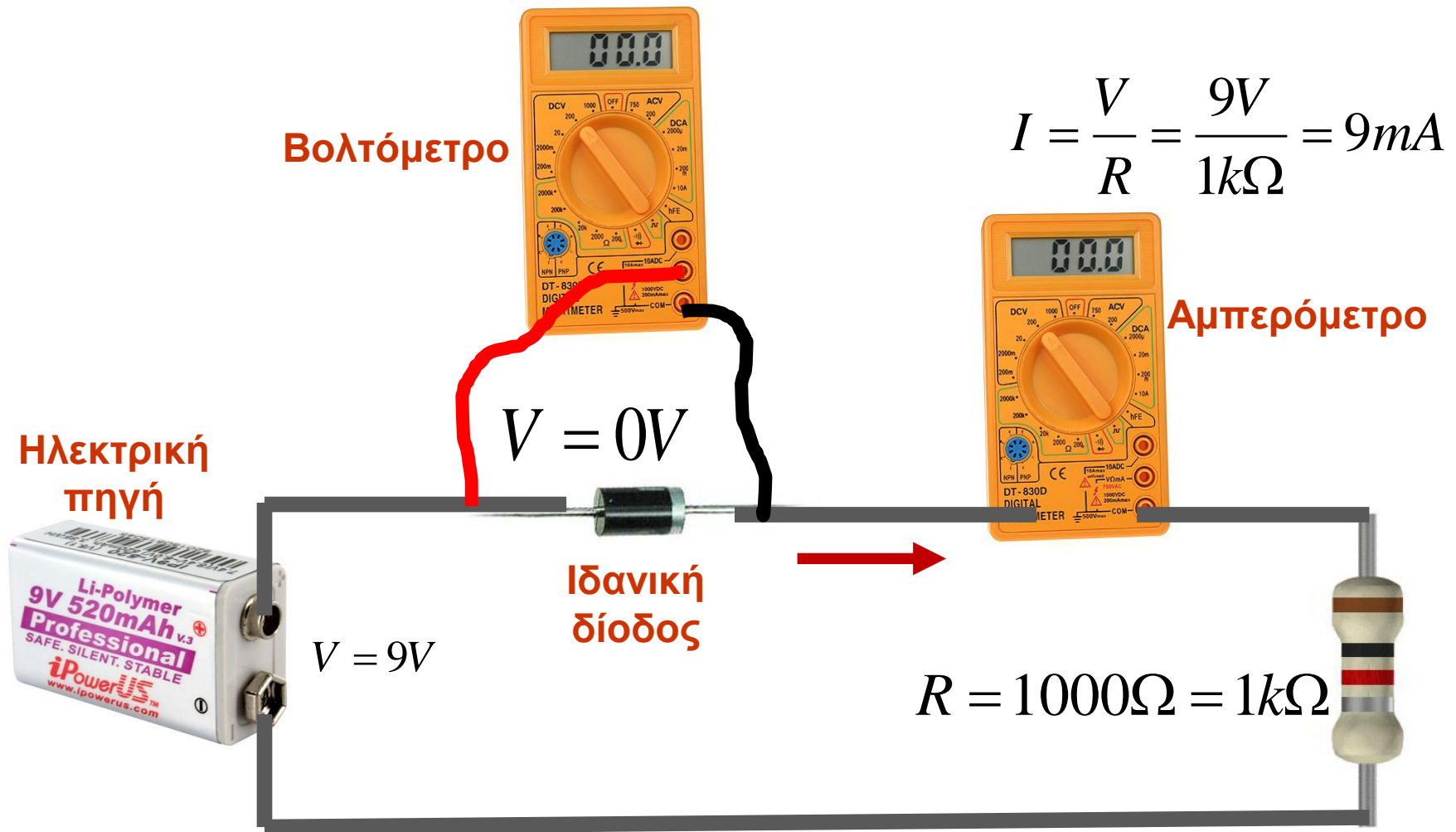


# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων

Σύνδεση πηγής και ορθή πόλωση διόδου → το κύκλωμα διαρρέεται από ρεύμα  $I$



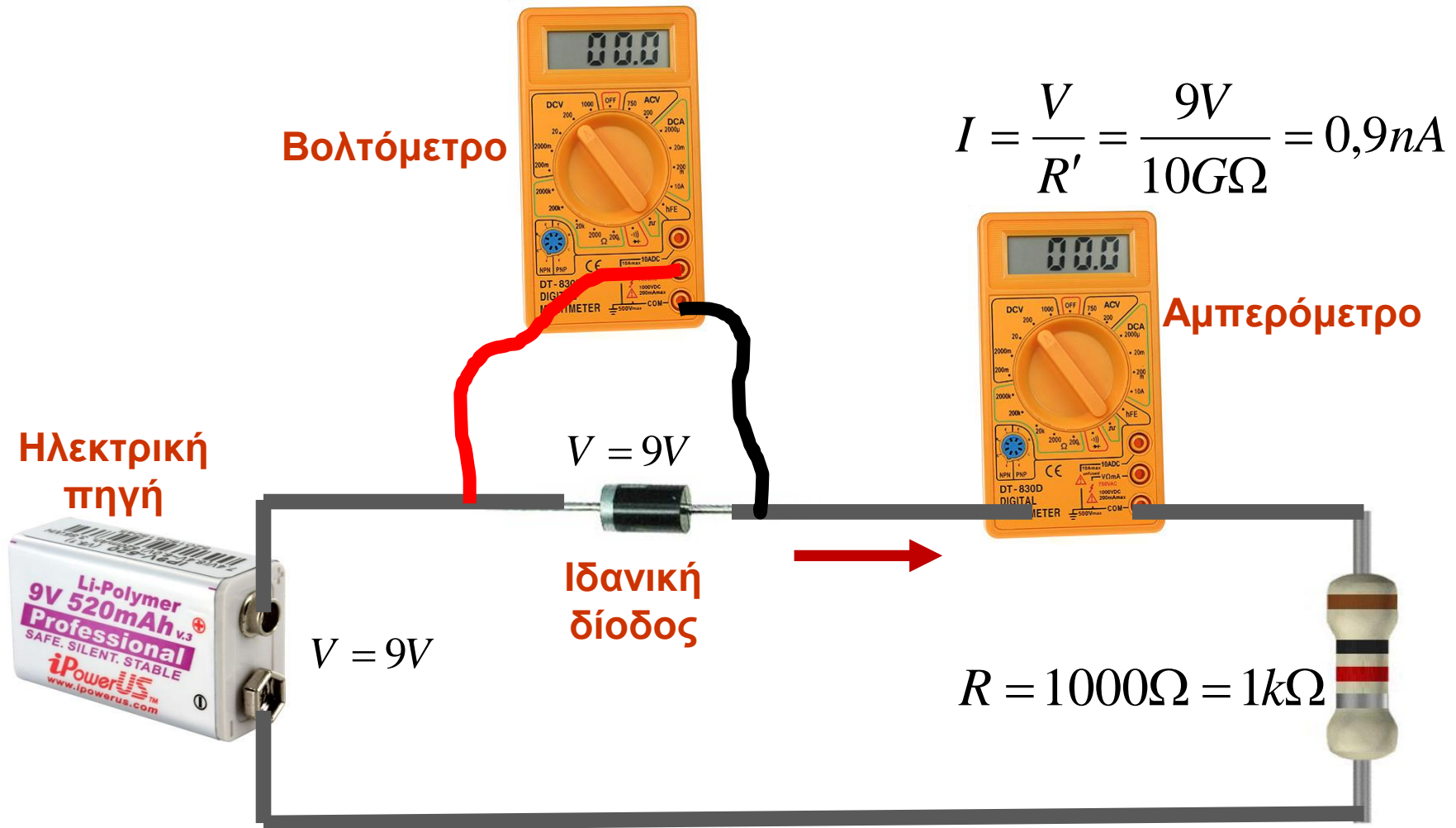
# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων



ορθή πόλωση διόδου → το κύκλωμα διαρρέεται από ρεύμα  $I$



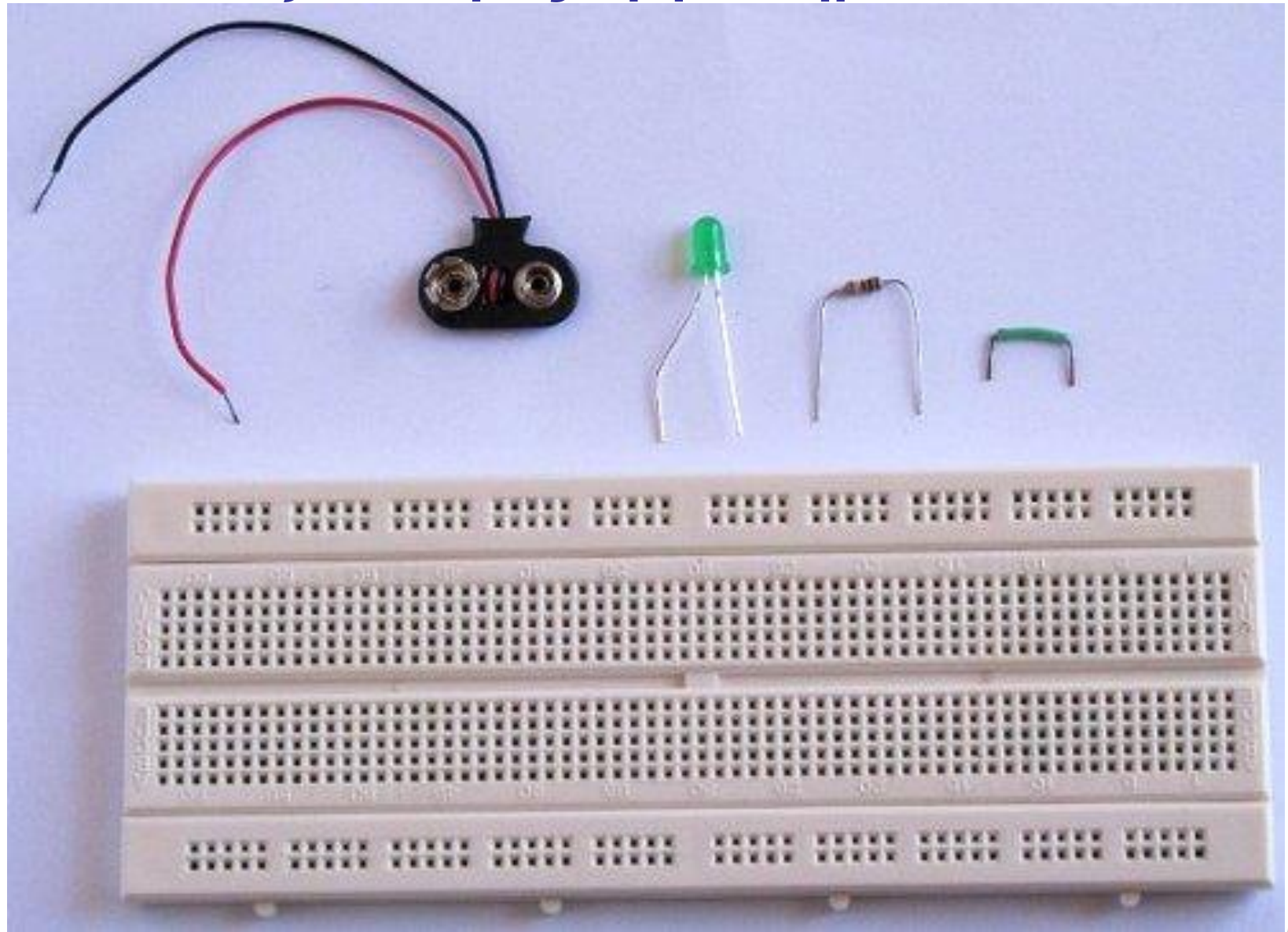
# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων



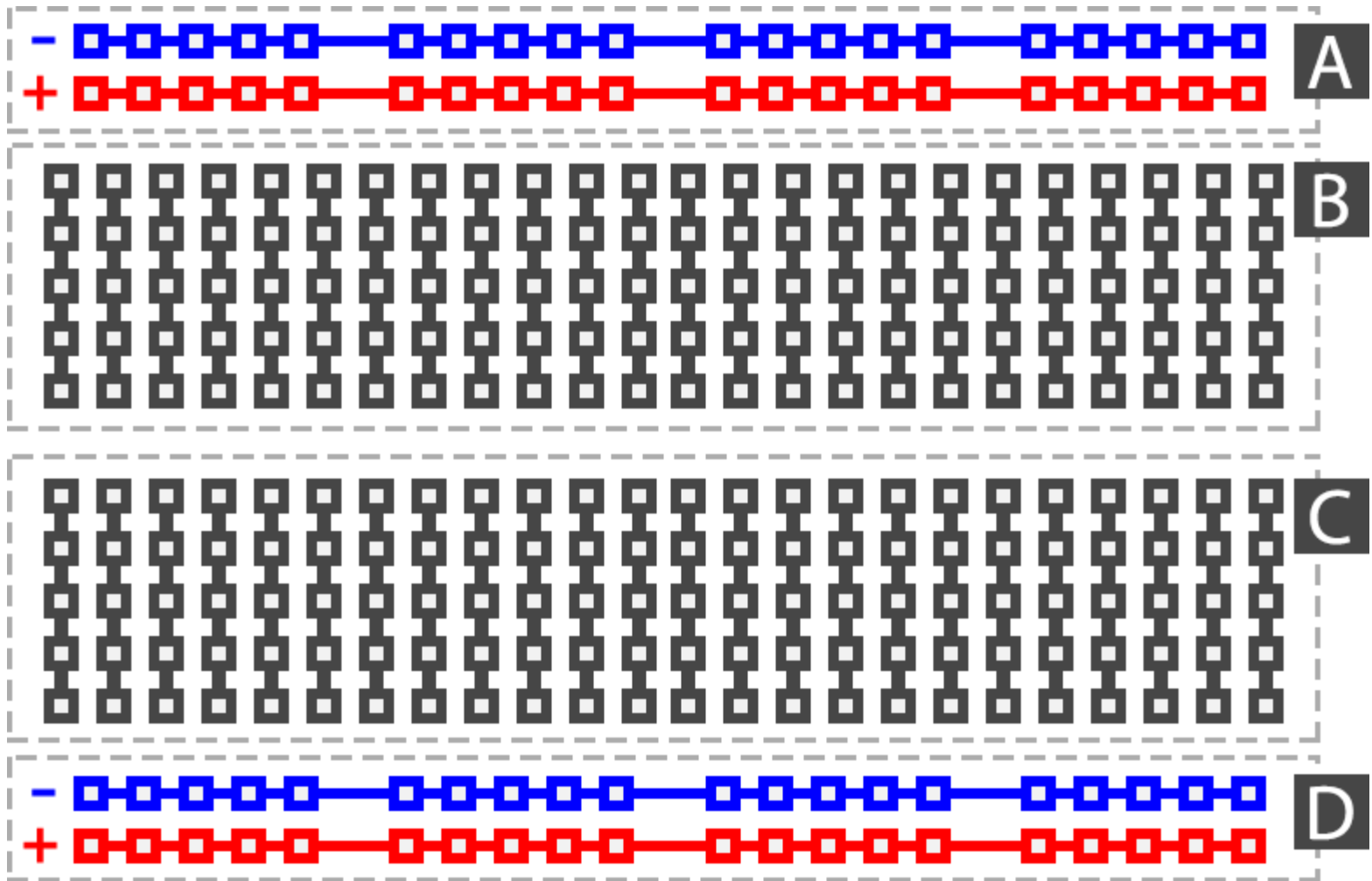
**Ανάστροφη πόλωση → το κύκλωμα δεν διαρρέεται από ρεύμα**



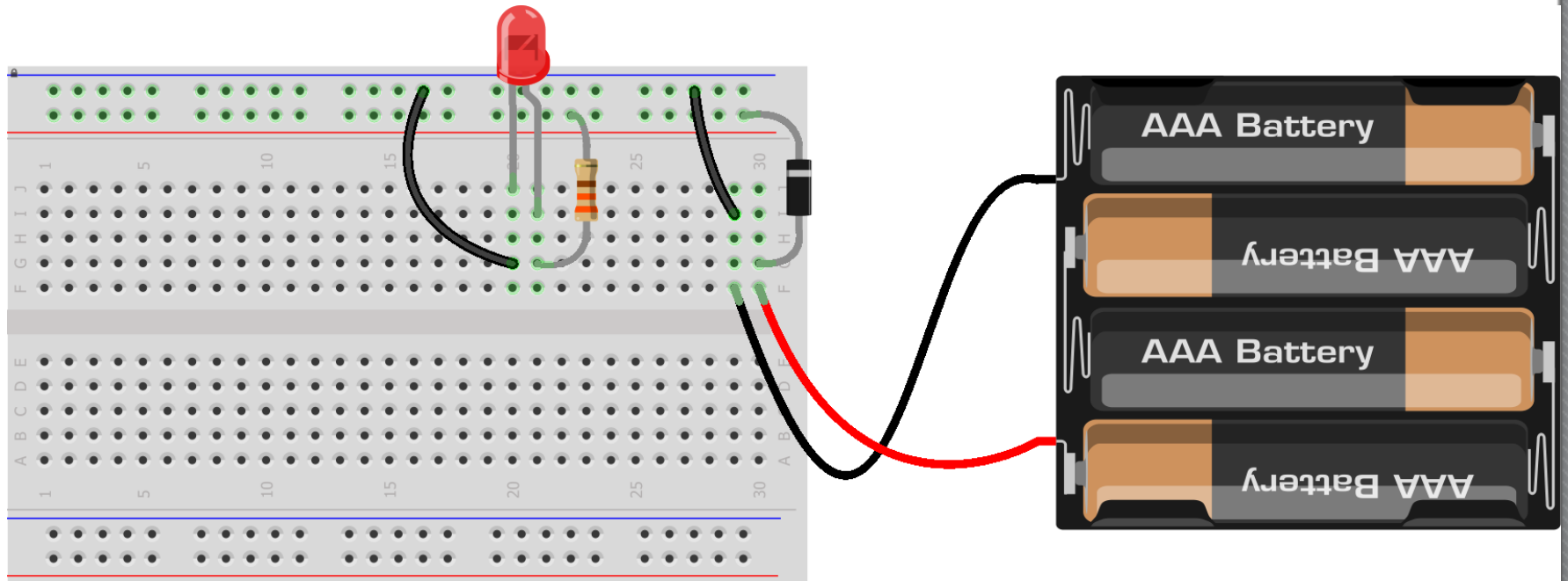
# Εξοπλισμός Εργαστηρίου



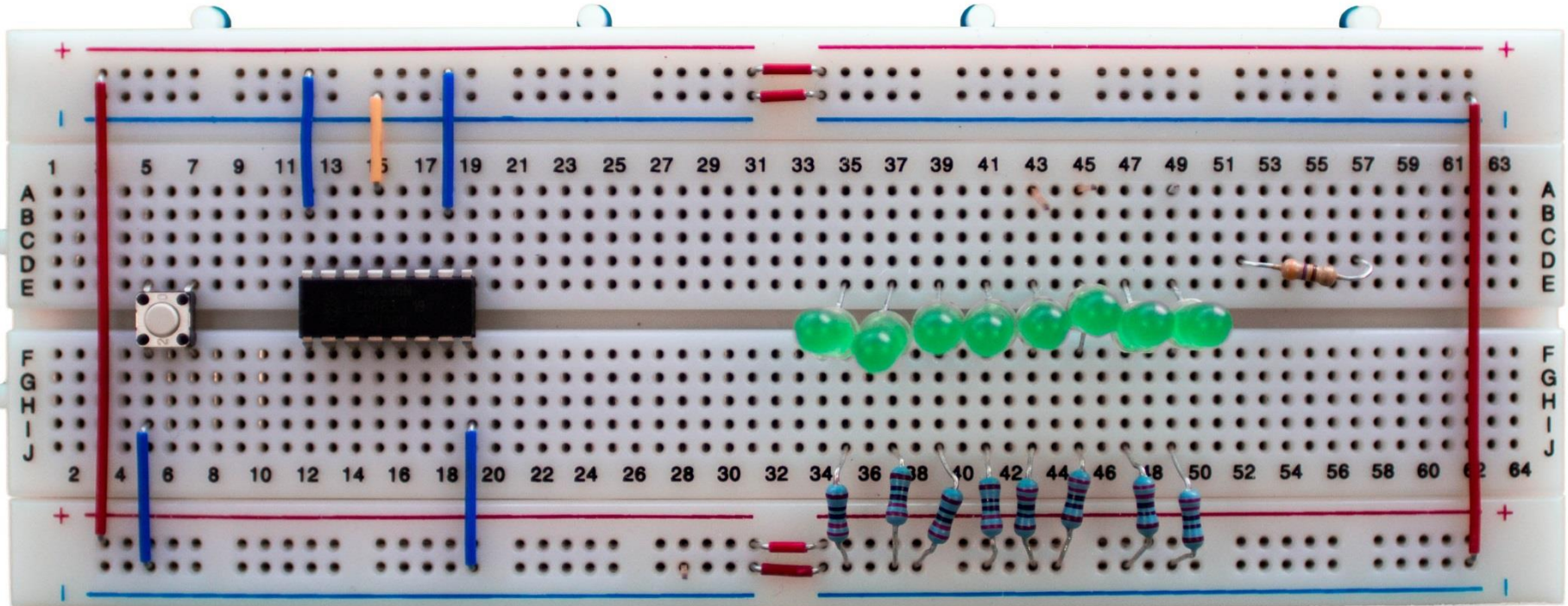
# Εξοπλισμός Εργαστηρίου



# 1<sup>η</sup> ενότητα – Κυκλώματα Διόδων με breadboard



# Εξοπλισμός Εργαστηρίου



# Προσομοίωση ηλεκτρονικών κυκλωμάτων

- **spice**
- **Tina (Texas Instruments)**
- **Multisim (National Instruments, NI\_Circuit\_Design\_Suite)**
- **ElectronicWorkbench**
- **Ltspice XVII**
- **ORCAD (Cadence) 9.1 (or 9.2)**



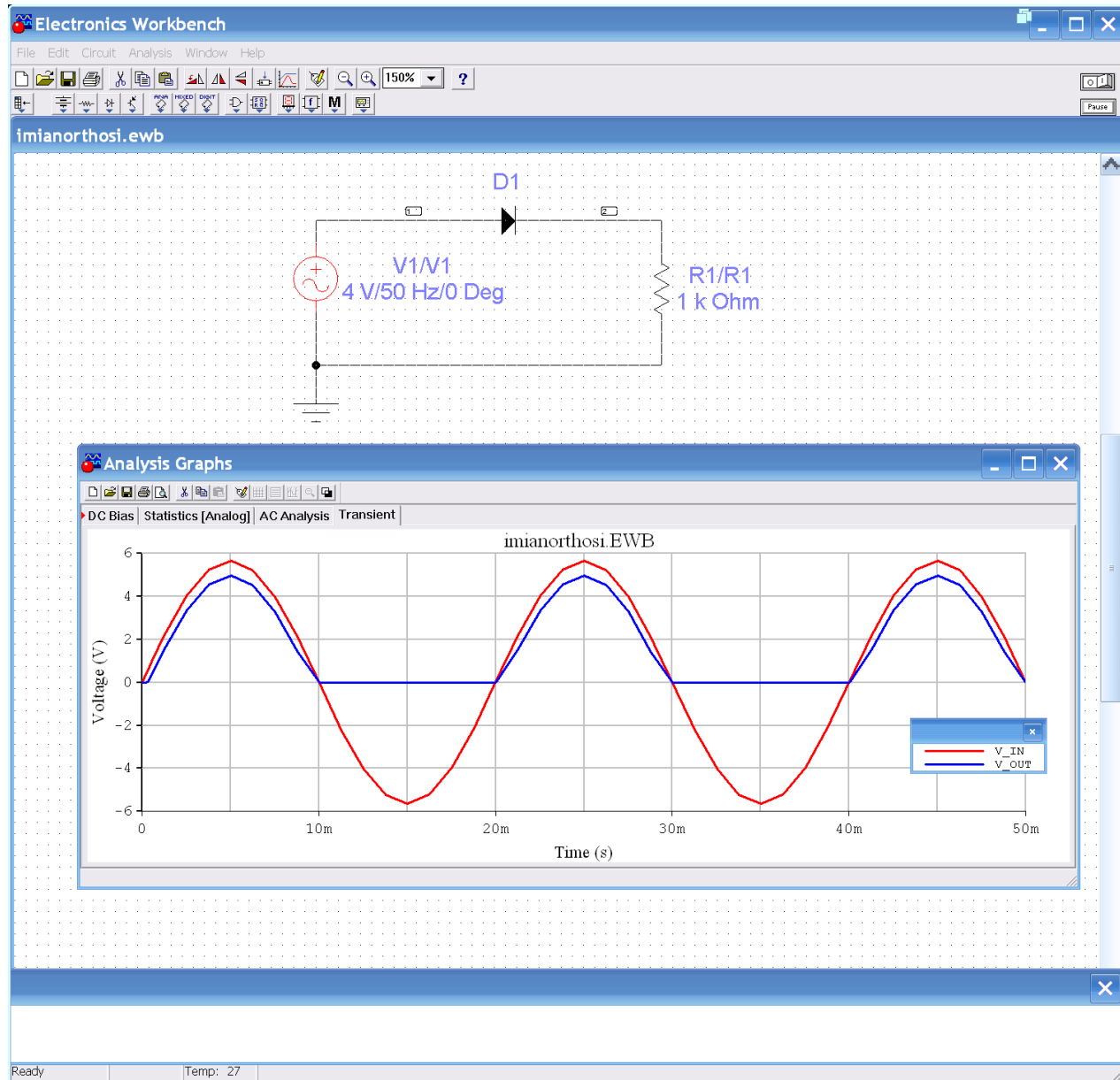
# ELECTRONICS WORKBENCH 5.12

The screenshot displays the Electronics Workbench 5.12 software interface. The title bar reads "Electronics Workbench". The menu bar includes "File", "Edit", "Circuit", "Analysis", "Window", and "Help". The "Analysis" menu is open, showing options: "Activate" (Ctrl+G), "Pause" (F9), "Stop" (Ctrl+T), "Analysis Options..." (Ctrl+Y), "DC Operating Point", "AC Frequency...", "Transient..." (highlighted), "Fourier...", and "Monte Carlo...". A "Display Graphs" checkbox is checked.

The circuit diagram is on a grid background. It consists of an AC voltage source labeled "V1/V1" with parameters "4 V/50 Hz/0 Deg". The source is connected in series with a diode labeled "D1" and a resistor labeled "R1/R1" with a value of "1 k Ohm". The circuit is grounded at the bottom.



# ELECTRONICS WORKBENCH 5.12





# ORCAD 9

The screenshot displays the OrCAD Capture software interface. The main window is titled "OrCAD Capture" and shows a schematic diagram of a circuit. The schematic includes a 5Vac AC voltage source labeled "V1" and a diode component. A "Place Part" dialog box is open, showing a list of parts including "SOURCE" which is highlighted. The dialog box also shows libraries, a graphic section with "Normal" and "Convert" options, and a packaging section with "Parts per Pkg: 1".

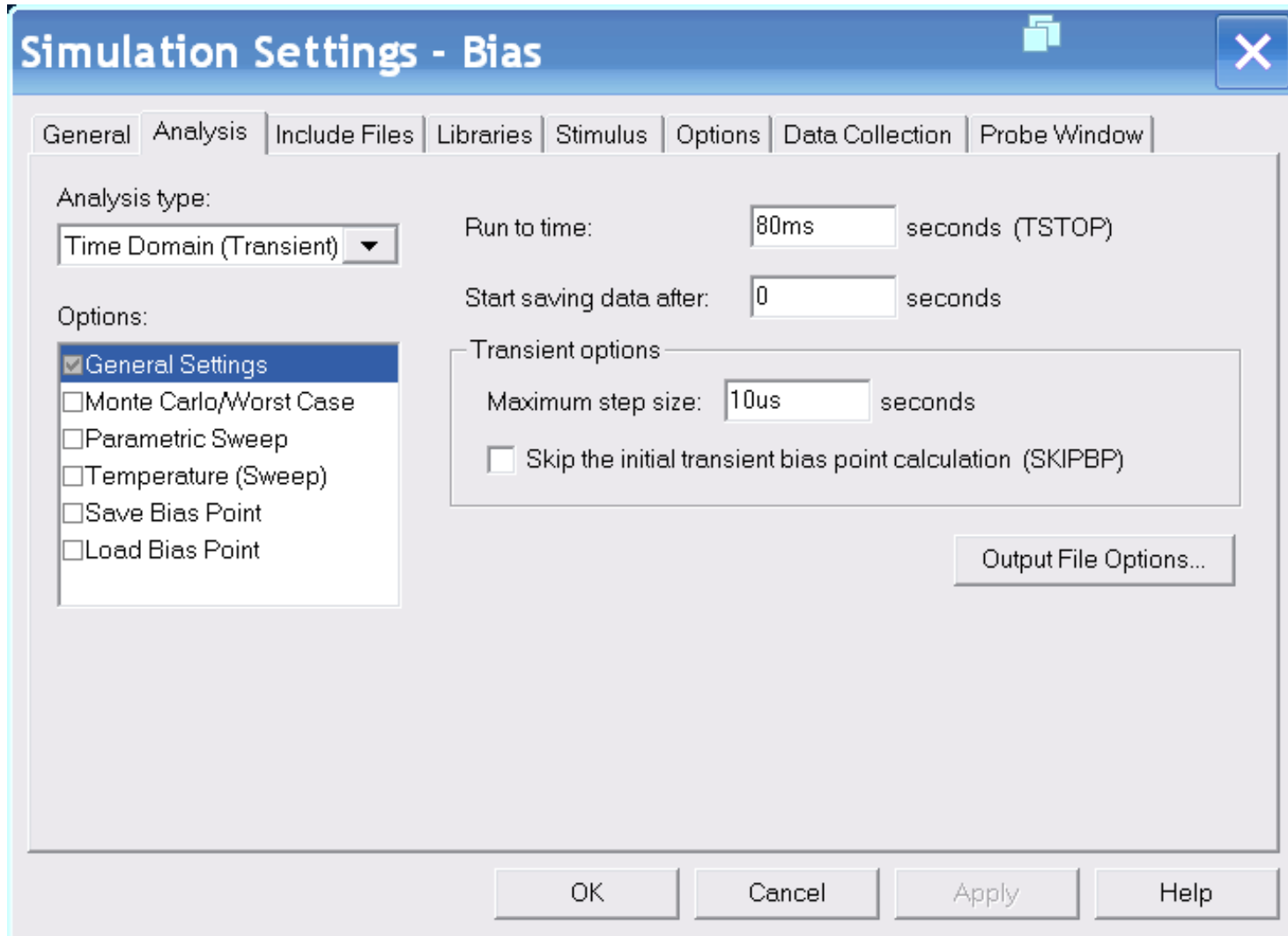
The interface includes a menu bar (File, Edit, View, Place, Macro, PSpice, Accessories, Options, Window, Help) and a toolbar. The left pane shows a hierarchy of design resources, including "Schematic1" and "PAGE1". The bottom pane shows a "Session Log" window.

# ORCAD 9

The screenshot displays the OrCAD 9 software interface. The main window shows a schematic diagram of a diode circuit. The circuit consists of an AC voltage source  $V1$  connected in series with a diode  $D1$  (1N4500) and a resistor  $R1$  (1k). The voltage source parameters are  $V_{OFF} = 0$ ,  $V_{AMPL} = 4$ , and  $FREQ = 50$ . The circuit is connected to ground  $0$ . The schematic is labeled with  $VIN$  at the input and  $VOUT$  at the output of the diode. The software interface includes a menu bar (File, Edit, View, Place, Macro, PSpice, Accessories, Options, Window, Help) and a toolbar. The left sidebar shows the file hierarchy for the project, including Design Resources, Library, Outputs, and PSpice Resources. The bottom status bar displays the following text:

```
PSpice netlist generation complete
Creating PSpice Netlist
Writing PSpice Flat Netlist C:\DOCUMENTS
PSpice netlist generation complete
Creating PSpice Netlist
Writing PSpice Flat Netlist C:\DOCUMENTS
PSpice netlist generation complete
```





**Simulation Settings - Bias**

General | Analysis | Include Files | Libraries | Stimulus | Options | Data Collection | Probe Window

Filename:  
 Browse...

Library files

Add as Global  
 Add to Design  
 Edit  
 Change

Library Path  
 Browse...

OK Cancel Apply Help

**Open**

Look in: PSpice

1_shot.lib	74hct.lib	analog.lib	cd4000.lib	dig_r
74ac.lib	74l.lib	anl_misc.lib	cel.lib	dig_p
74act.lib	74ls.lib	anlg_dev.lib	comlinr.lib	dig_p
74als.lib	74s.lib	apex.lib	darlingtn.lib	digfil
74as.lib	7400.lib	bipolar.lib	dc_prim.lib	diode
74f.lib	adv_lin.lib	breakout.lib	dig_ecl.lib	ebipc
74h.lib	amp.lib	buffer.lib	dig_gal.lib	edioc
74hc.lib	ana_swit.lib	burr_brn.lib	dig_io.lib	elant

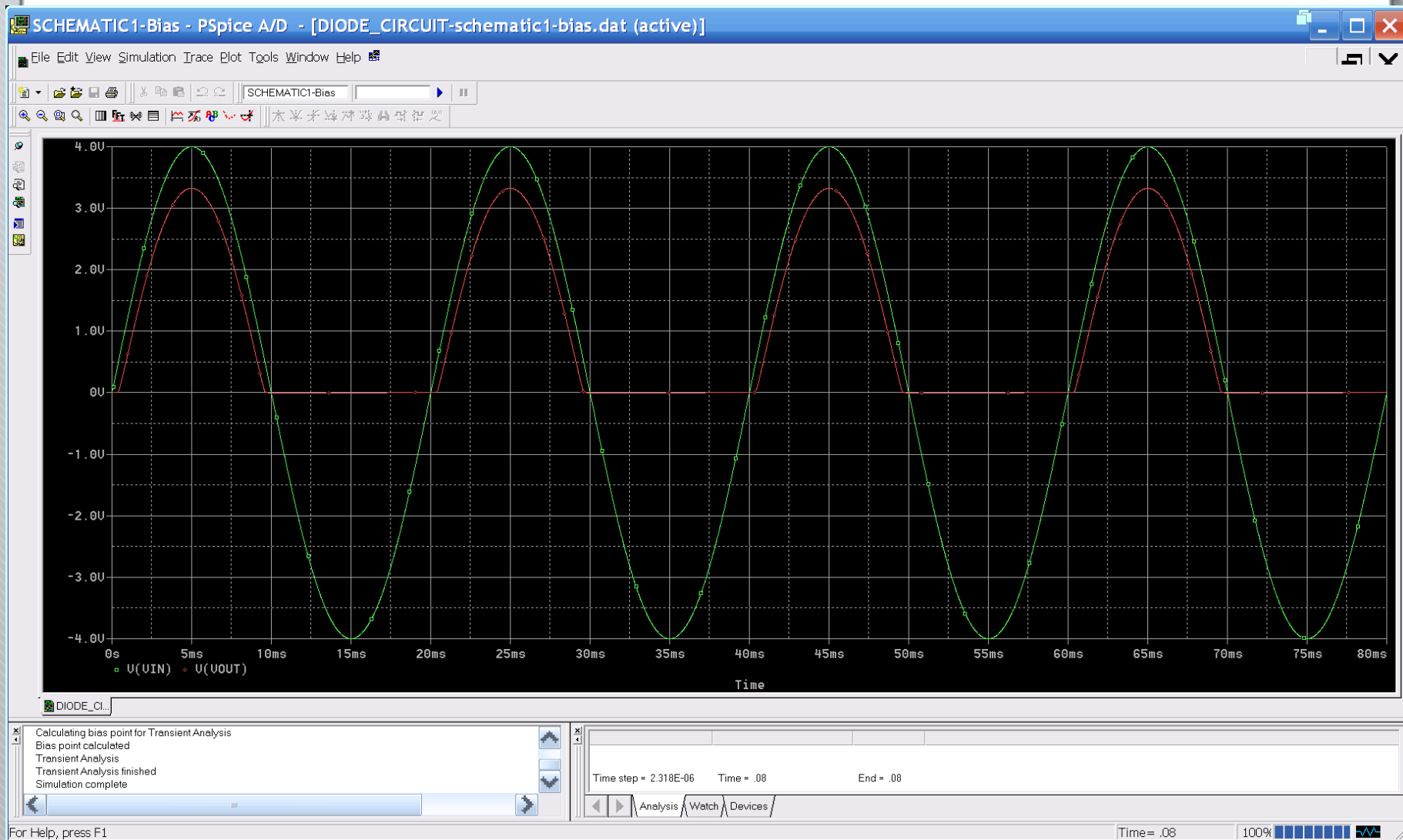
File name:

Files of type: Library Files (\*.lib)

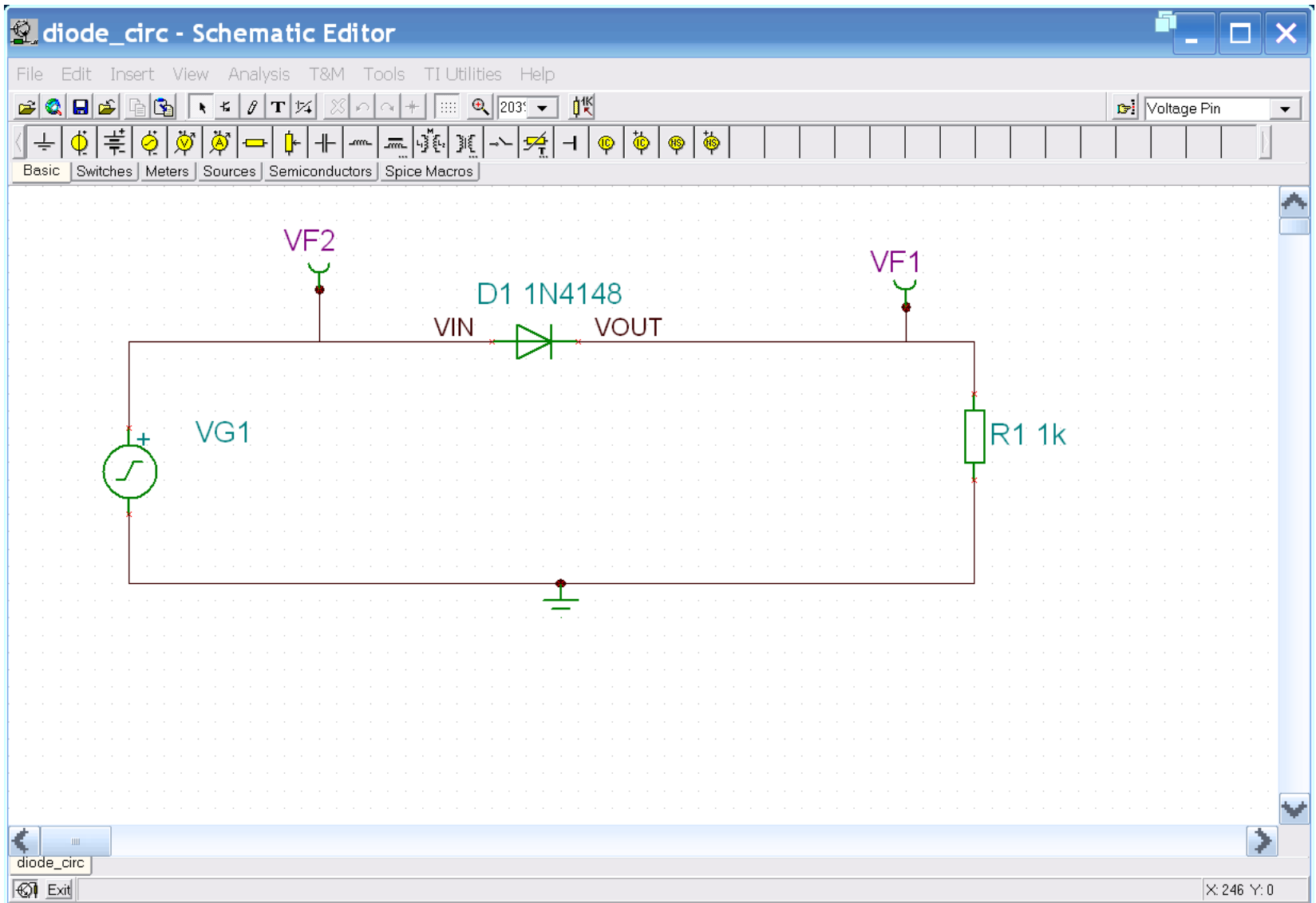
Open Cancel




# ORCAD 9



# TINA (TI)





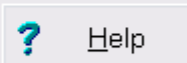
**Transient Analysis** 

Start display  [s]

End display  [s]

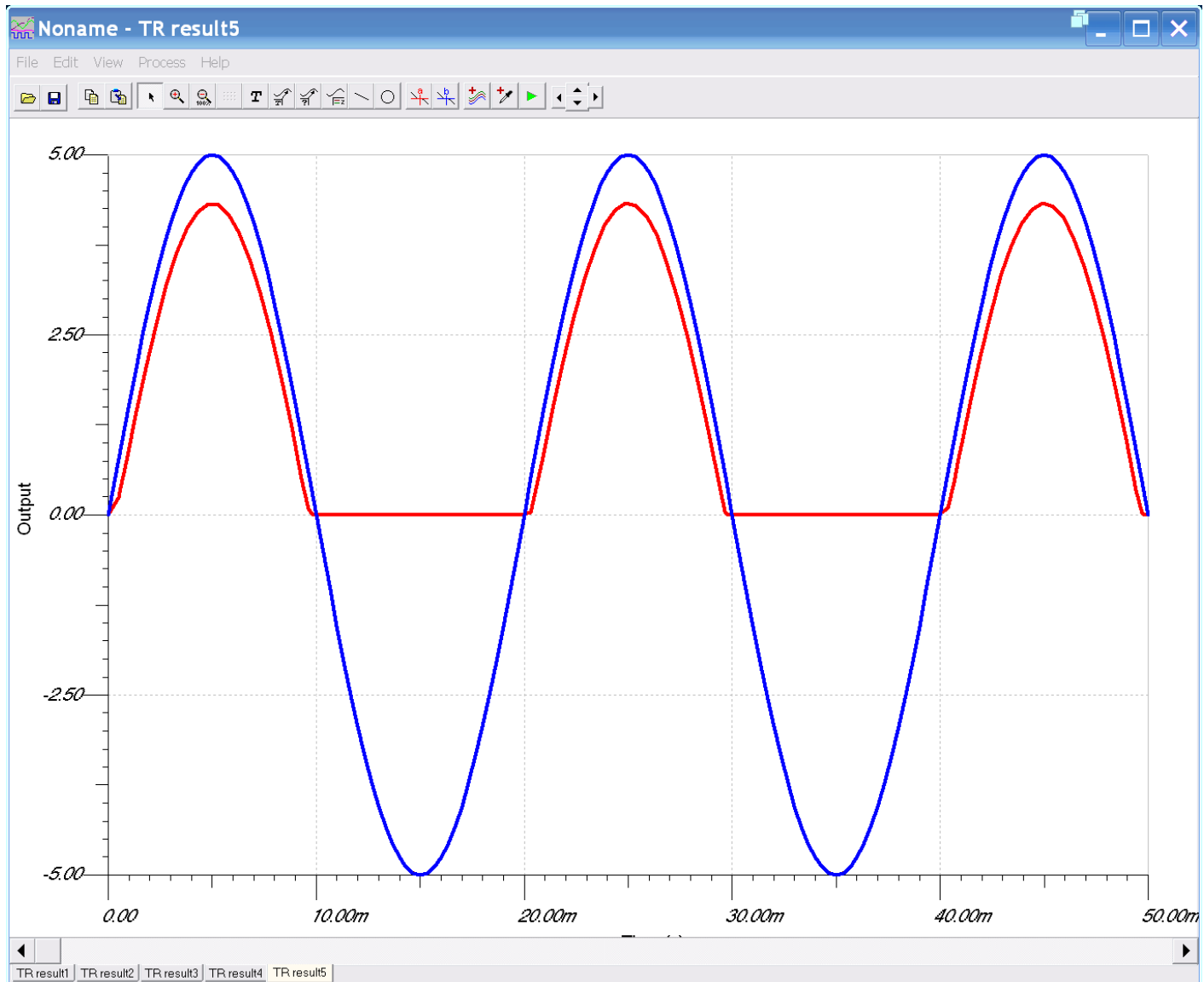
Calculate operating point  
 Use initial conditions  
 Zero initial values

Draw excitation

 OK  
 Cancel  
 Help



# TINA (TI)





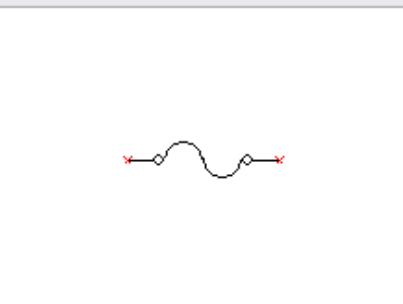
# MULTISIM (NI)

## Select a Component

Database: Master Database

Component: 0.5\_AMP

Symbol (ANSI Y32.2)



Function: 0.5 Amp Fuse

Model manufacturer/ID: IIT / FUSE

Package manufacturer/type: Generic / FUSE1

Hyperlink:

Group: All <All groups>

- All <All groups>
- Sources
- Basic
- Diodes
- Transistors
- Analog**
- TTL
- CMOS
- MCU
- Advanced\_Peripherals
- Misc Digital
- Mixed
- Indicators
- Power
- Misc
- RF
- Electro\_Mechanical
- Ladder\_Diagrams
- Connectors
- NI\_Components

Component list:

- 0.5\_AMP
- 02BZ2.2
- 02DZ4.7
- 05AZ2.2
- 1 mF [T491E108M004AT]
- 1 mF [T491X108M004AT]
- 1 mF [T494E108M004AT]
- 1 mF [T494X108M004AT]
- 1 mF [T495E108M004ATE035]
- 1 mF [T495E108M006ATE050]
- 1 mF [T495X108M004ATE070]
- 1 mF [T510E108M004ATE010]
- 1 mF [T510E108M004ATE018]
- 1 mF [T510X108M004ATE018]
- 1 mF [T510X108M004ATE023]
- 1 mF [T513E108M004AH6110]
- 1 mF [T513X108M004AH6110]
- 1 mF [T513X108M004AH6120]
- 1 mF [T520D108M2R5ATE015]
- 1 mF [T520D108M2R5ATE030]
- 1 mF [T520X108M003ATE015]

Components: 34606    Searching:    Filter: off



# MULTISIM (NI)

The screenshot displays the Multisim software interface with a circuit simulation setup. The main workspace shows a circuit with the following components:

- V1:** An AC voltage source with parameters: 4Vrms, 50Hz, and 0° phase.
- D1:** A diode component labeled 1N4148.
- R1:** A resistor component labeled 1kΩ.

The circuit is connected to a ground symbol. The interface includes a menu bar (File, Edit, View, Place, MCU, Simulate, Transfer, Tools, Reports, Options, Window, Help), a toolbar, a Design Toolbox on the left, and a Spreadsheet View at the bottom. The status bar at the bottom indicates the date and time: "Multisim - Κυριακή, 19 Φεβρουαρίου 2017, 8:59:57 μμ".



# MULTISIM (NI)

**Design 1 - Multisim - [Design 1 \*]**

File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help

Full screen F11  
Parent sheet

Design Toolbox  
Zoom in Ctrl+Num +  
Zoom out Ctrl+Num -  
Zoom area F10  
Zoom sheet F7  
Zoom to magnification... Ctrl+F11  
Zoom selection F12

Grid  
Border  
Print page bounds  
Ruler bars  
Status bar  
Design Toolbox  
Spreadsheet View  
SPICE Netlist Viewer  
LabVIEW Co-simulation Terminals  
Circuit Parameters  
Description Box Ctrl+D  
Toolbars

Show comment/probe  
Grapher

V1: 4Vrms, 50Hz, 0°  
D1: 1N4148  
R1: 1kΩ  
XSC1: Scope showing a sine wave  
PR1: V: 2.84 V, V(p-p): 5.00 V, V(rms): 2.43 V, V(dc): 1.50 V, V(freq): 50.0 Hz  
PR2: V: -5.65 V, V(p-p): 11.3 V, V(rms): 4.00 V, V(dc): -3.00 mV, V(freq): 50.0 Hz

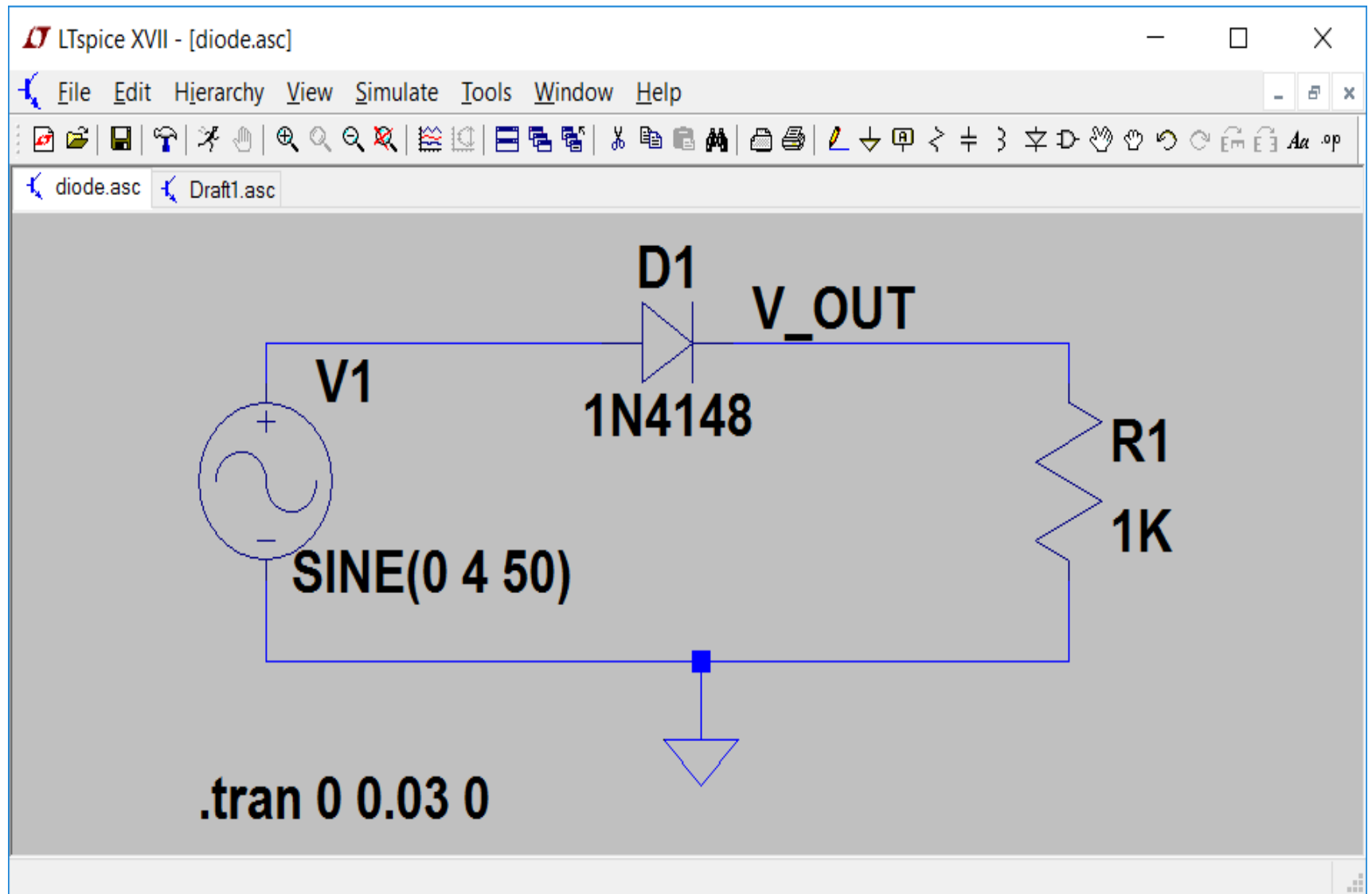
Tran: 0.235 s

Show the Grapher.

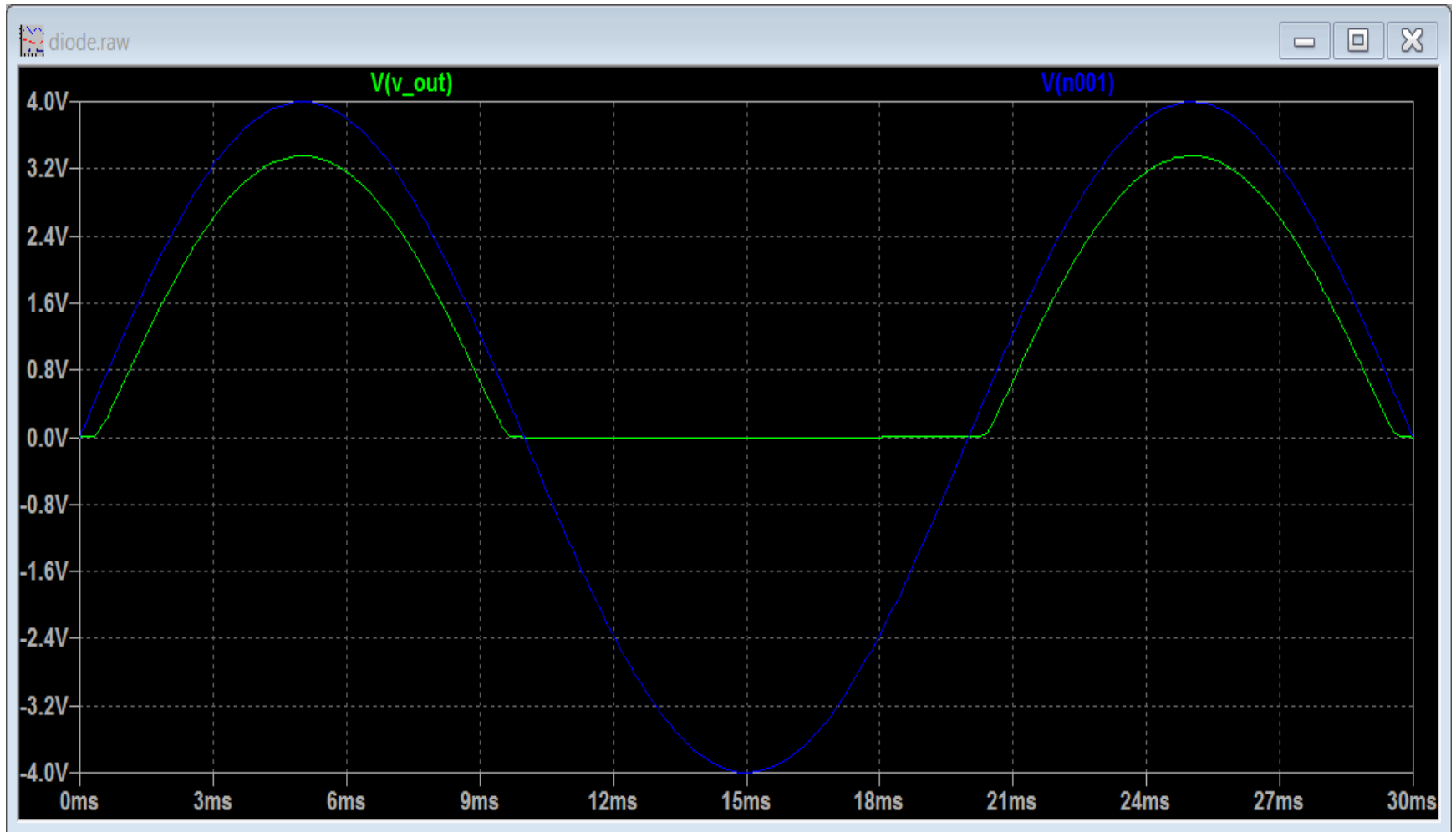
# MULTISIM (NI)



# LTspice



# LTspice



**Τέλος 1<sup>ου</sup> Ρ.Ρ.**

