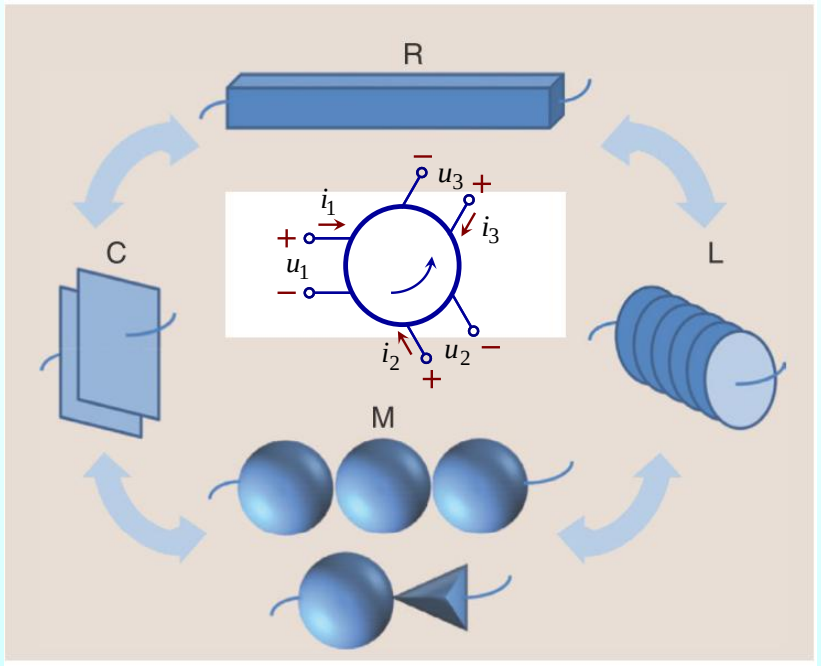
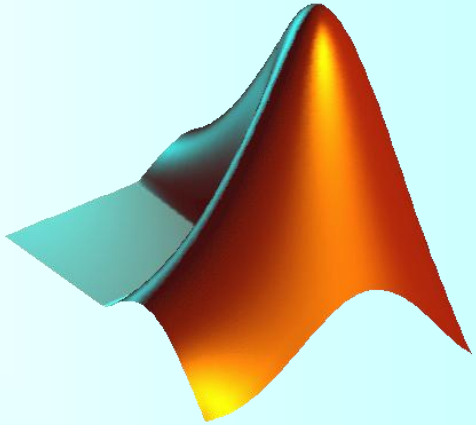
др Милка Потребић, ред. проф., др Никола Баста, доцент, *Практикум из рачунарске анализе кола*, Универзитет у Београду – Електротехнички факултет, 2021.

Практикум из рачунарске анализе кола





Милка Потребић

Никола Баста

Рачунарски (софтверски) алати

- *Mathematica*, WolframAlpha, MuPAD, **Maxima**, **SymPy**, **SymPy Live**, **SymPy Gamma**
- MATLAB, **Scilab**, **Octave**, **FreeMat**, **Julia**
- LTspice, **QucsStudio**, **ngspice**, **Xyce**, **XCircuit**
- **Python**, MathCAD, MAPLE, GeoGebra, Sage
- Symbolab, SpeQ Mathematics, meta-calculator, Desmos, …
- Android apk: NCalc+, CalcES, SymJa CAS, Scientific Calculator Pro, CYZSoft Scientific Calculator Plus

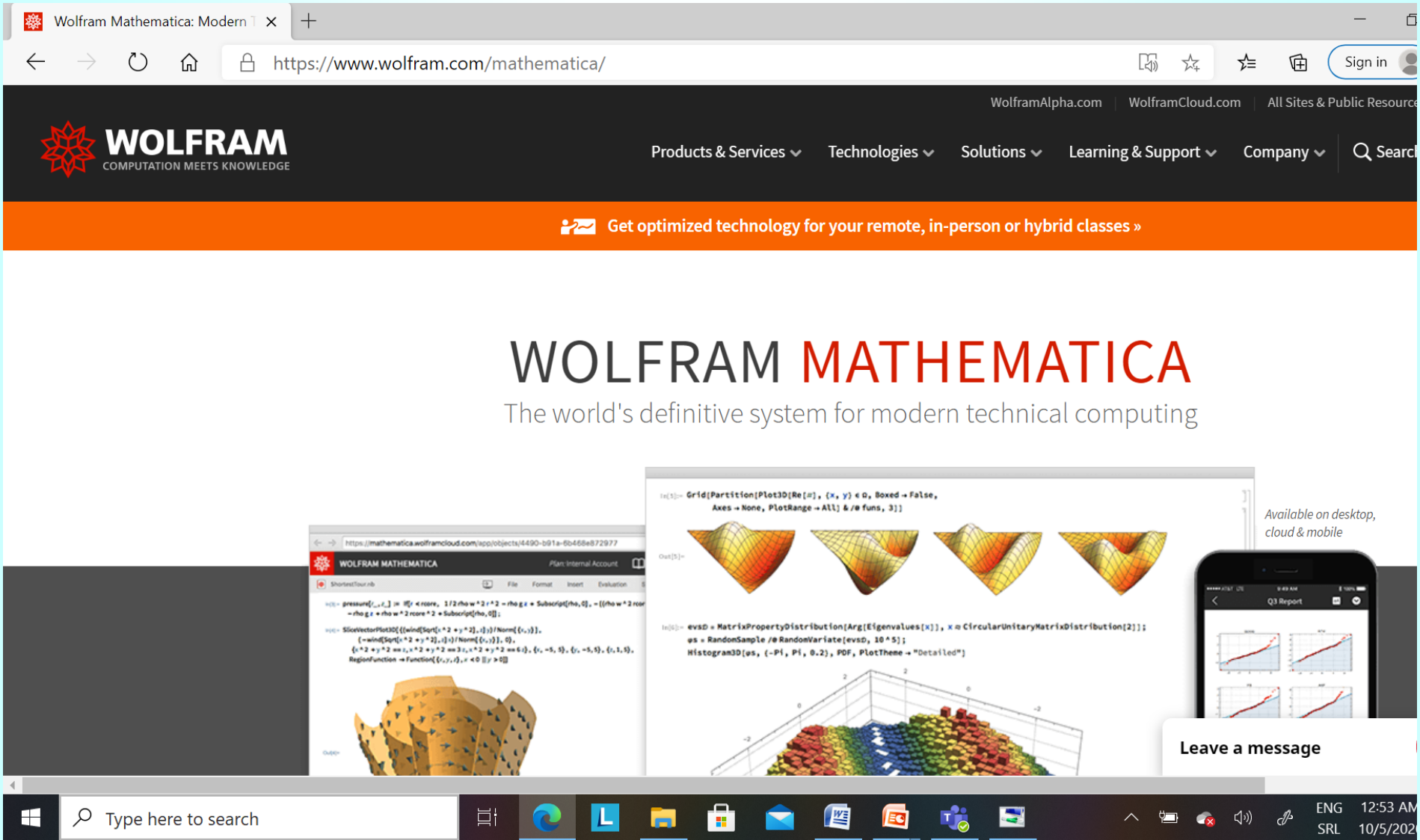
Free/Libre Open Source Software (FLOSS)

Моделовање електричних кола

- PSpice, ngSpice, Xcos, SciLab, TINA-TI, Micro-Cap, Plexim, 5Spice…
- **Online Circuit Simulator**: DoCircuits, Circuit Lab, PartSim, EasyEda, Multisim, PartQuest (SystemVision Cloud)...
- **Android circuit simulator:**
  Electric Circuit Studio,
  Circuit Safari SPICE Simulator...

Mathematica

https://www.wolfram.com/mathematica/



Mathematica free on every Raspberry Pi!

https://www.wolfram.com/raspberry-pi/

www.wolframalpha.com

Give your brain a quick workout with Wolfram Problem Generator.

WolframAlpha computational... knowledge engine

y' +1000 * y = 10000, y(0)=-10

Examples Random

Input:

$\{y'(x) + 1000\, y(x) = 10\,000,\ y(0) = -10\}$

ODE classification:

first-order linear ordinary differential equation

Alternate forms:

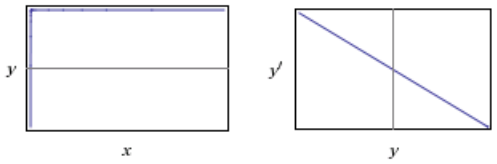
$\{y'(x) = 10\,000 - 1000\, y(x),\ y(0) = -10\}$

$\{y'(x) + 1000\,(y(x) - 10) = 0,\ y(0) = -10\}$

Differential equation solution: Approximate form | Step-by-step solution

$y(x) = 10 - 20\, e^{-1000\, x}$

Plots of the solution:



New to Wolfram|Alpha?

Take the Tour >>

New! Wolfram Problem Generator

Need a hint?

MATLAB

http://www.mathworks.com/

MathWorks - Makers of MATLAB

https://www.mathworks.com

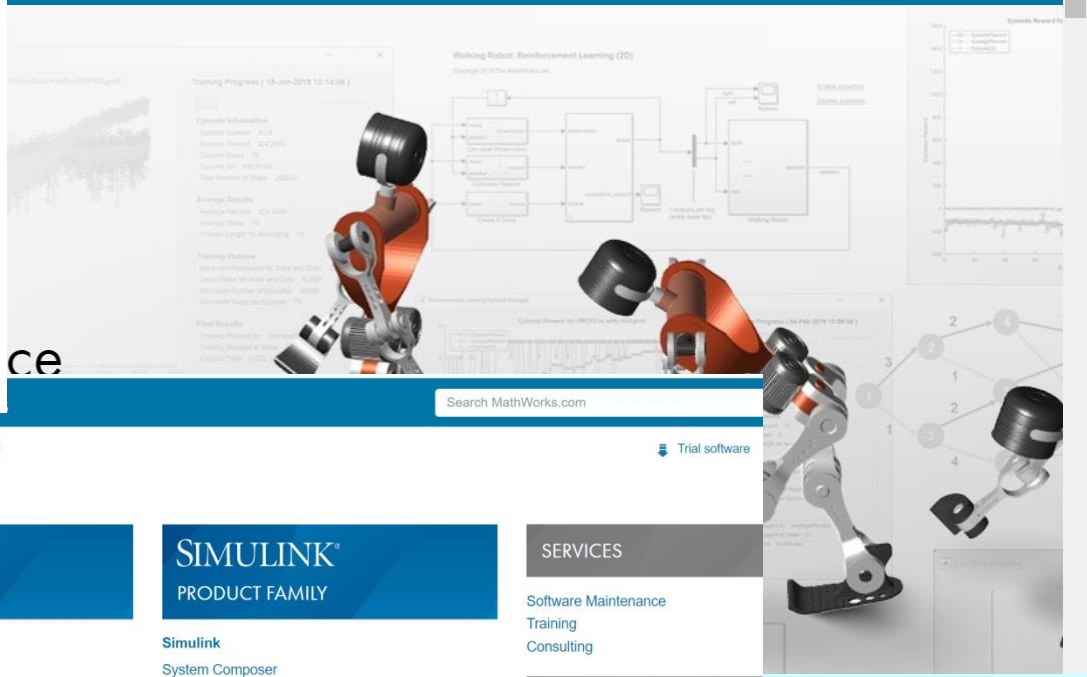
Sign in

MathWorks® Products Solutions Academia Support Community Events Get MATLAB

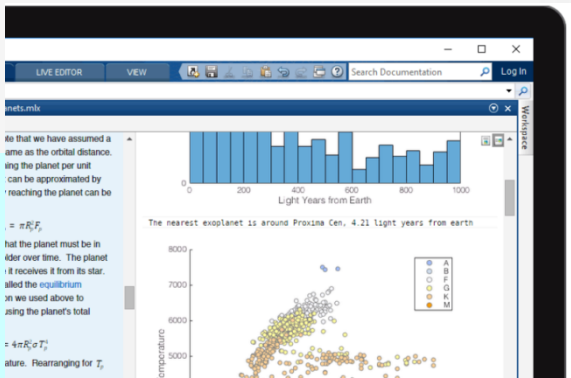
Search MathWorks.com

Millions of Engineers and Scientists Trust MATLAB

MATLAB® combines a desktop environment tuned for iterative analysis and design processes with a programming language that expresses matrix and array mathematics directly. It includes the Live Editor for creating scripts that combine code, output, and formatted text in an executable notebook.



Design AI models and AI-driven systems



Search MathWorks.com

Products by Category Alphabetical Trial software

MATLAB® PRODUCT FAMILY

MATLAB

Parallel Computing
- Parallel Computing Toolbox
- MATLAB Parallel Server

Math and Optimization
- Curve Fitting Toolbox
- Optimization Toolbox
- Global Optimization Toolbox
- Symbolic Math Toolbox
- Mapping Toolbox

SIMULINK® PRODUCT FAMILY

Simulink
- System Composer

Event-Based Modeling
- Stateflow
- SimEvents

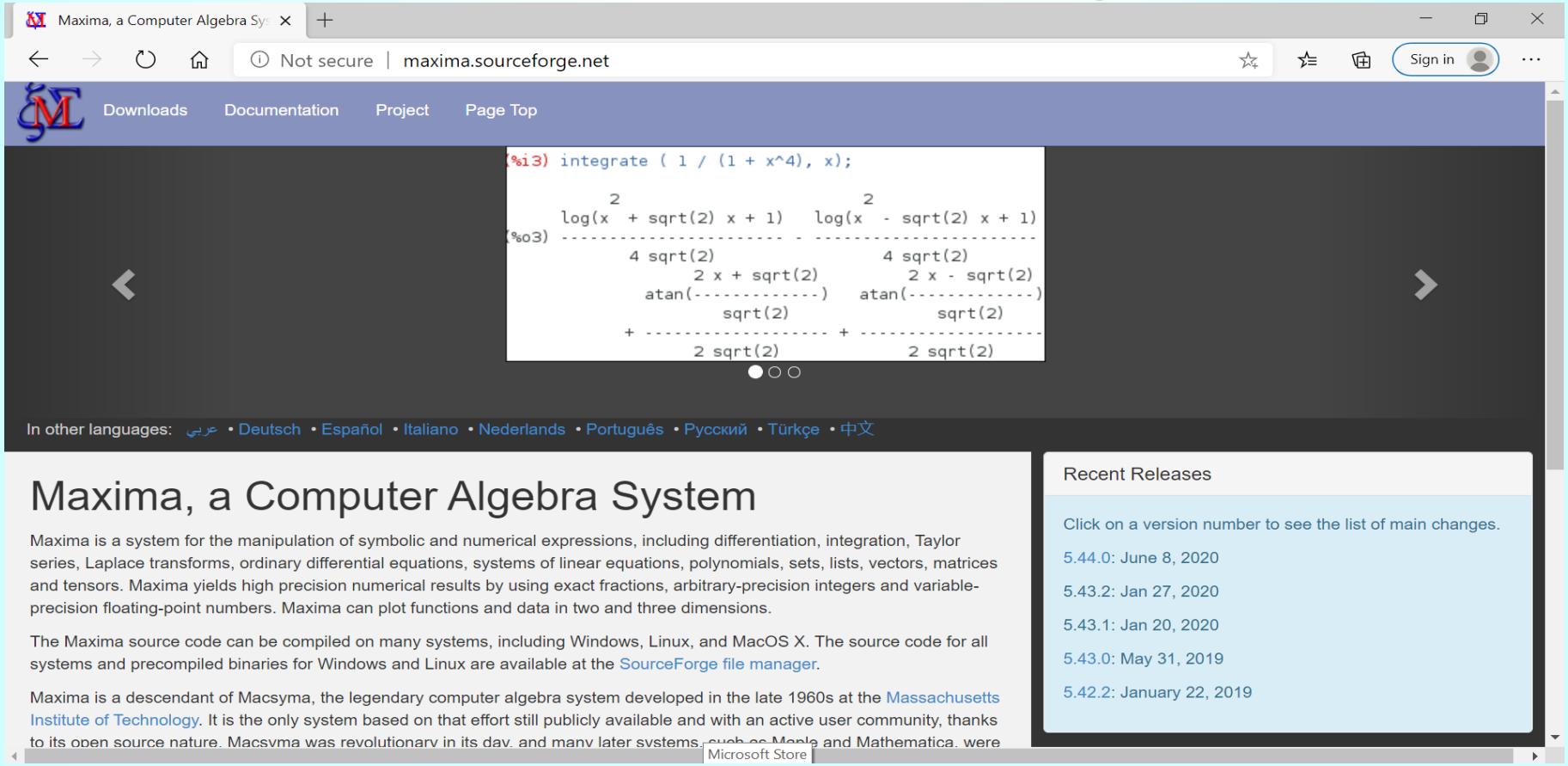
Physical Modeling
- Simscape
- Simscape Driveline
- Simscape Electrical
- Simscape Fluids

SERVICES
- Software Maintenance
- Training
- Consulting

LICENSE TYPES
- Industry Use
- Student Use
- University Use
- Startup Use
- Primary and Secondary School Use
- Home Use

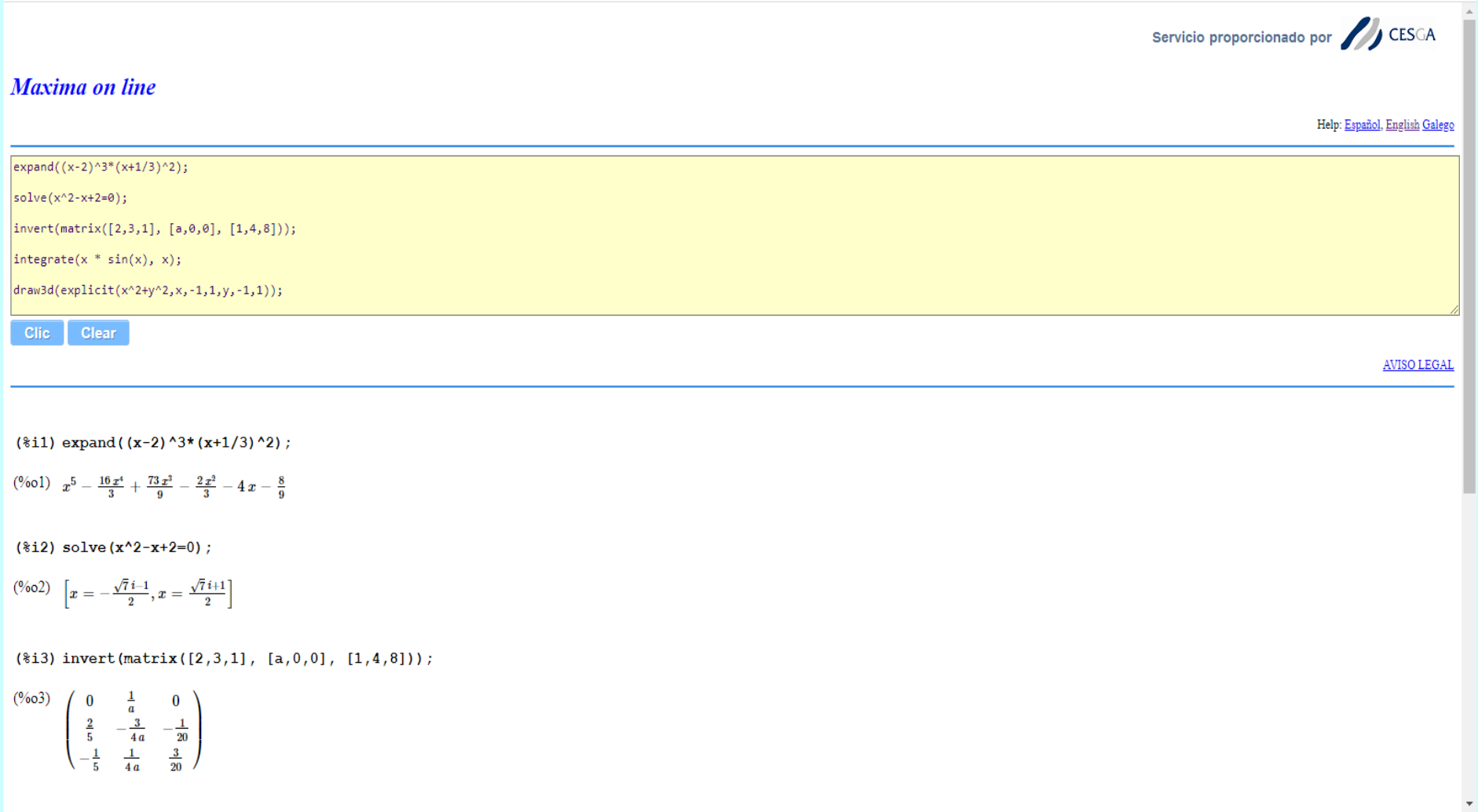
Maxima

http://maxima.sourceforge.net/



Maxima on line

http://maxima.cesga.es/



Maxima on Android

https://play.google.com/store/apps/details?id=jp.yhonda&hl=en_US&gl=US

https://play.google.com/store/apps/details?id=jp.yhonda&hl=en_US&gl=US

Google Play

Search

Apps

Categories

Home

Top charts

New releases

My apps

Shop

Games

Kids

Editors' Choice

Account

Payment methods

Play Points New

My subscriptions

Redeem

Buy gift card

My wishlist

My Play activity

Parent Guide

Maxima on Android

Yasuaki Honda Education

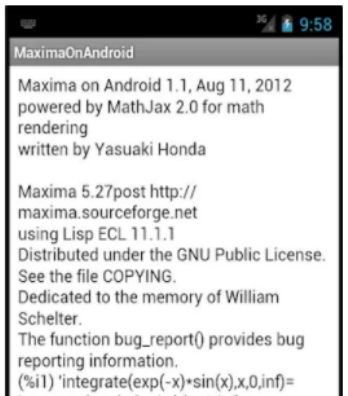
2,100

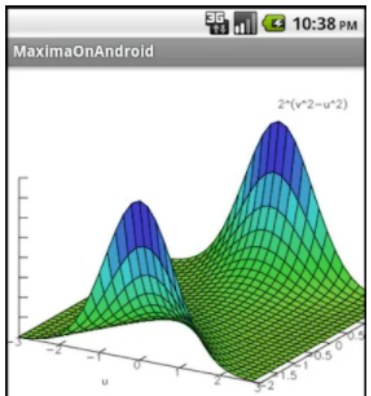
Everyone

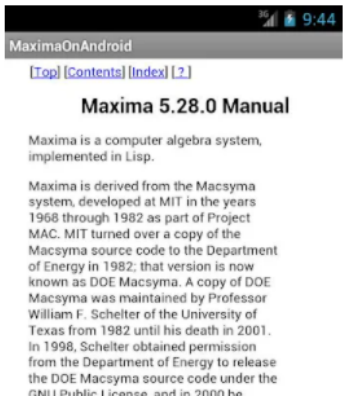
You don't have any devices

Add to Wishlist

Install

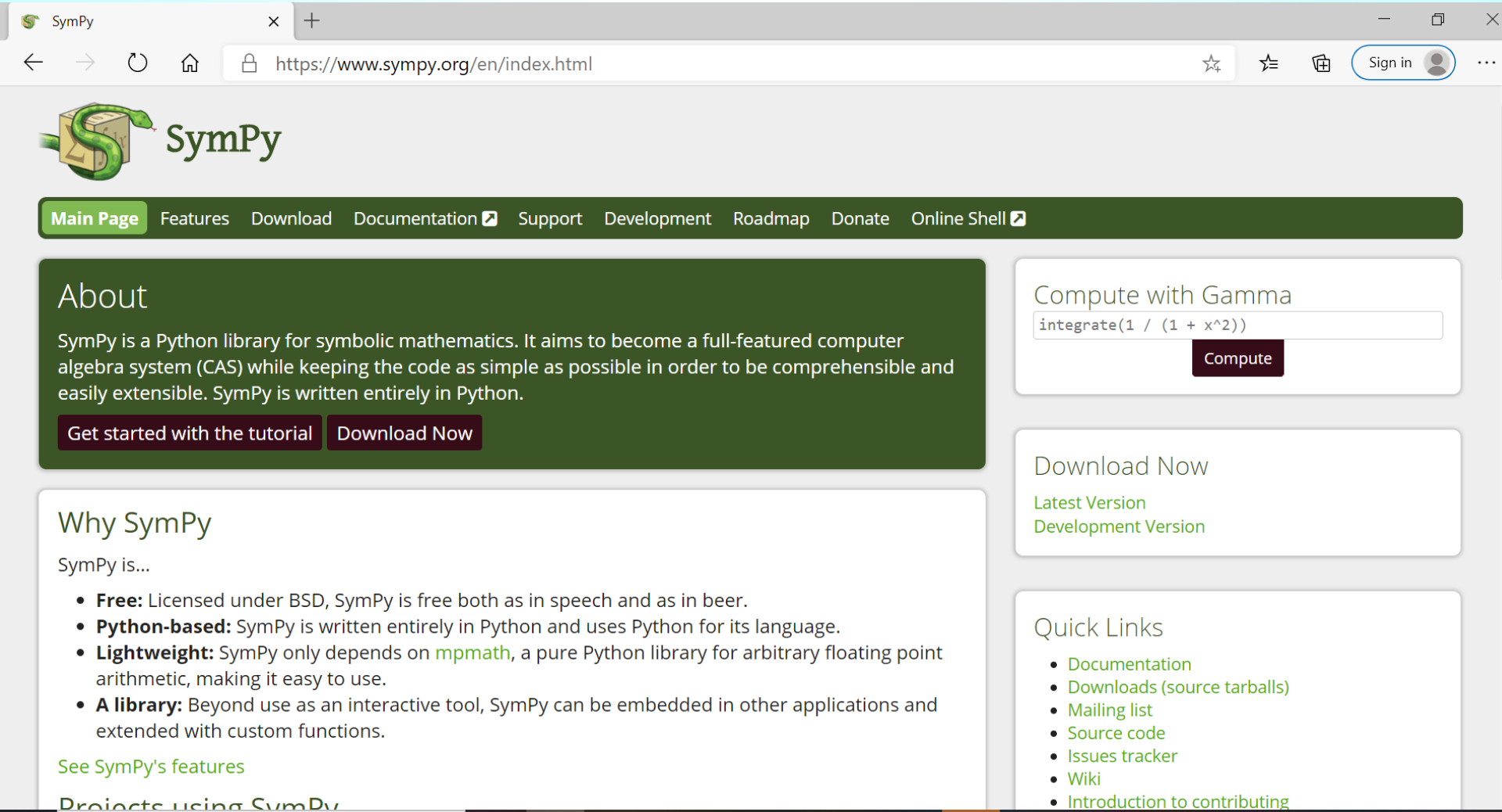






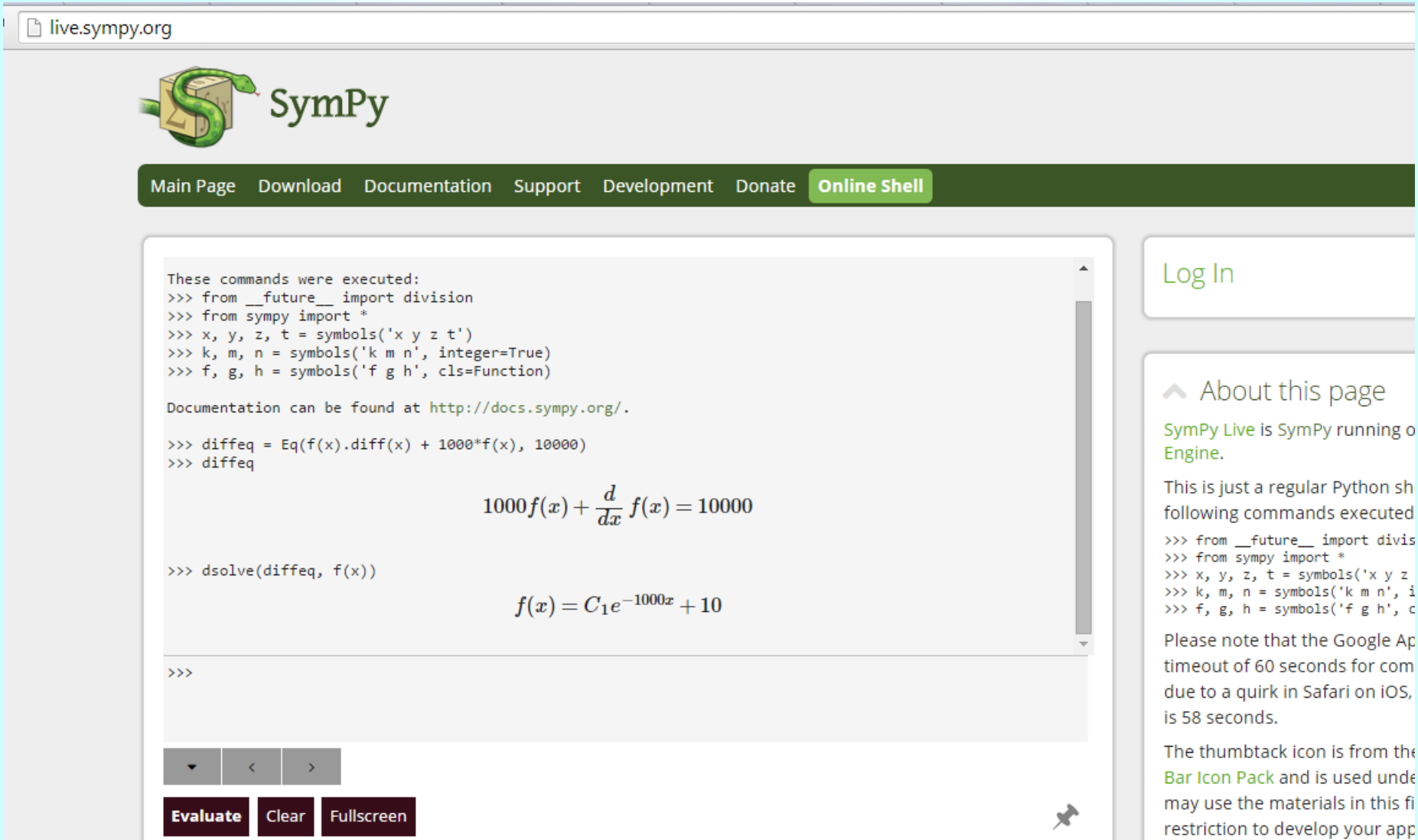
SymPy

www.sympy.org/



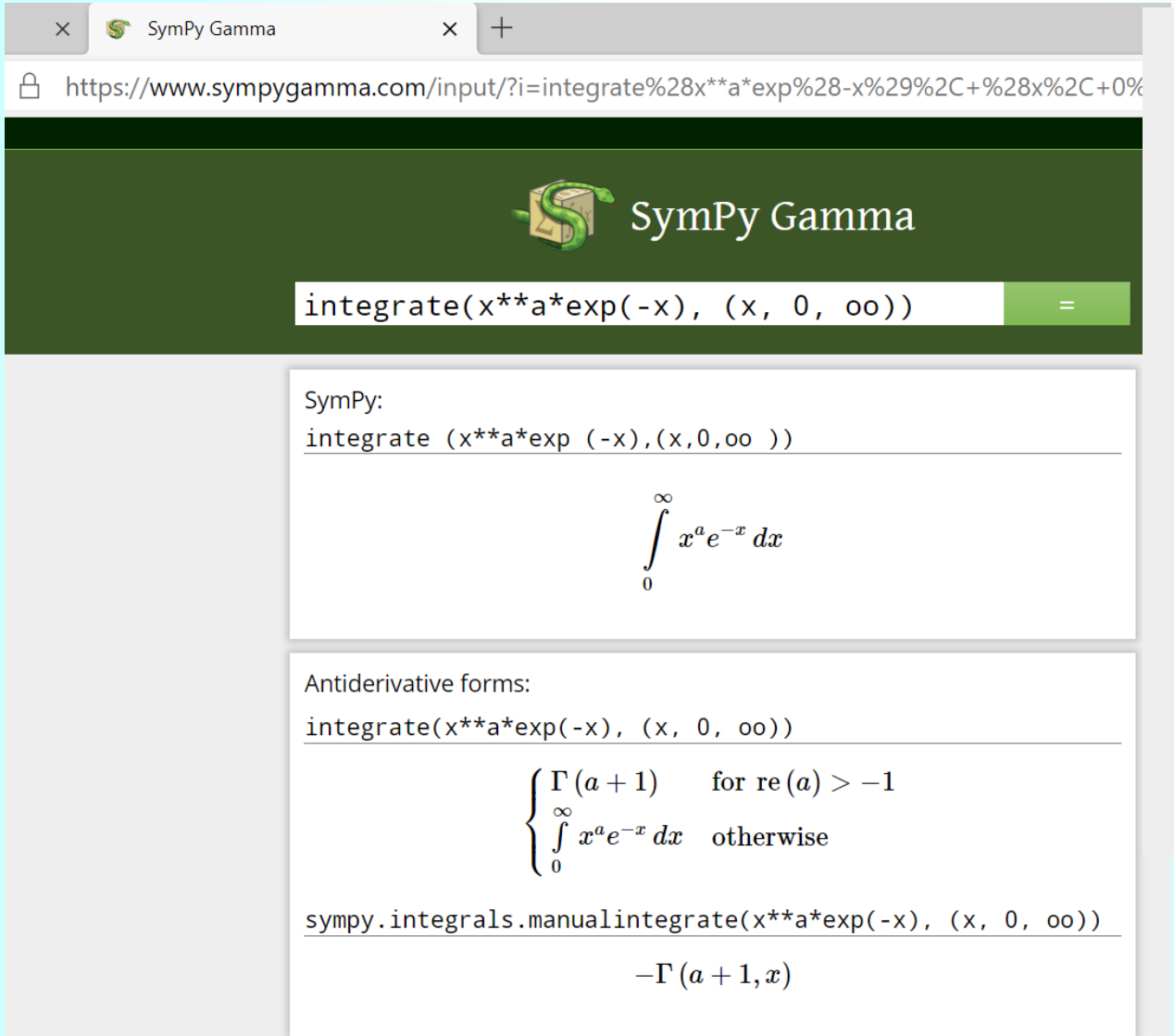
SymPy live

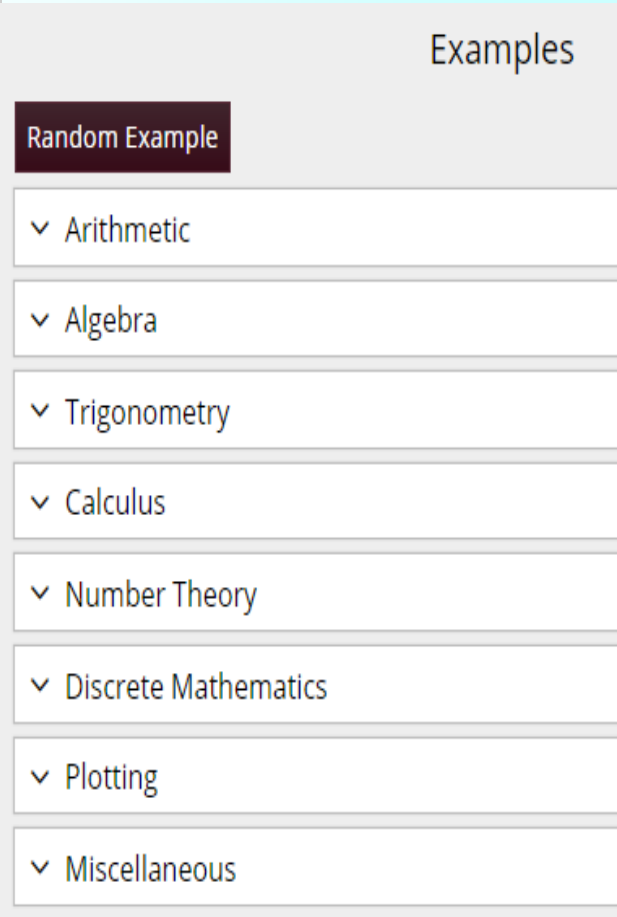
live.sympy.org



SymPy Gamma

https://www.sympygamma.com/





```
[1]: from sympy import *
     x = symbols('x')
     import sympy;sympy.__version__
[1]: '1.4'
[2]: Integral(exp(-x**2), (x, -oo, oo))
[2]: ∫_{-∞}^{∞} e^{-x^2} dx
[3]: _.doit()
[3]: √π
```

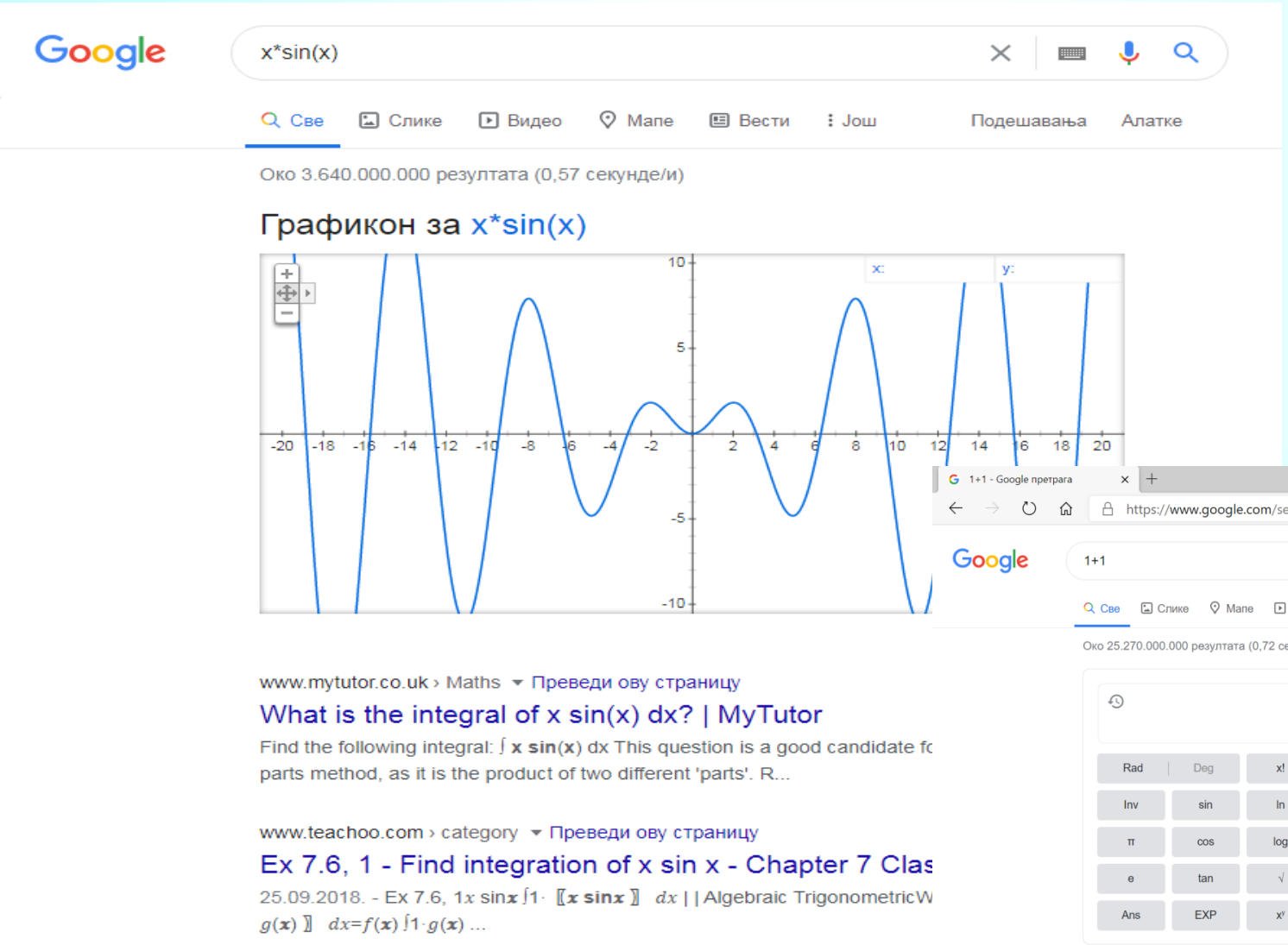
https://jupyter.org

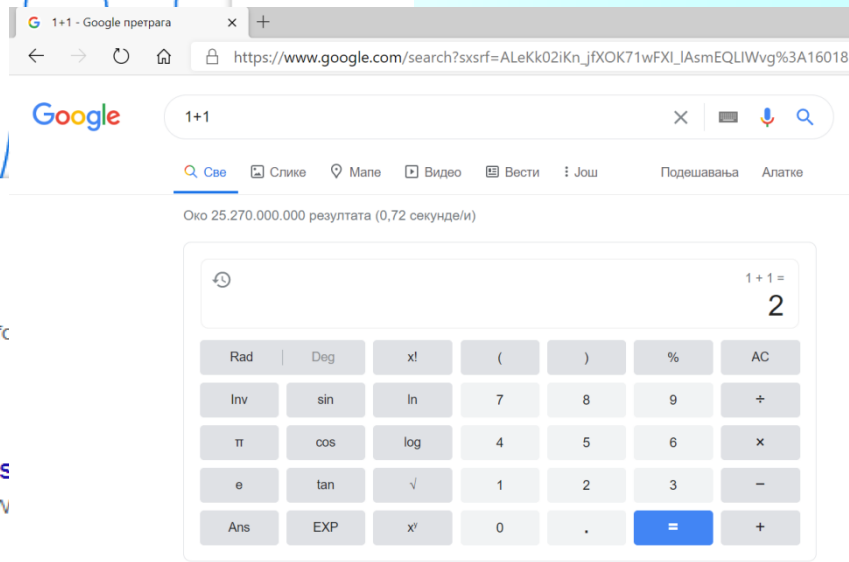
jupyter Install About Us Community Events Do

Sympy and the Jupyter Notebook for engineering calculations

Project Jupyter exists to develop open-source software, open-standards, and services
interactive computing across dozens of programming languages.

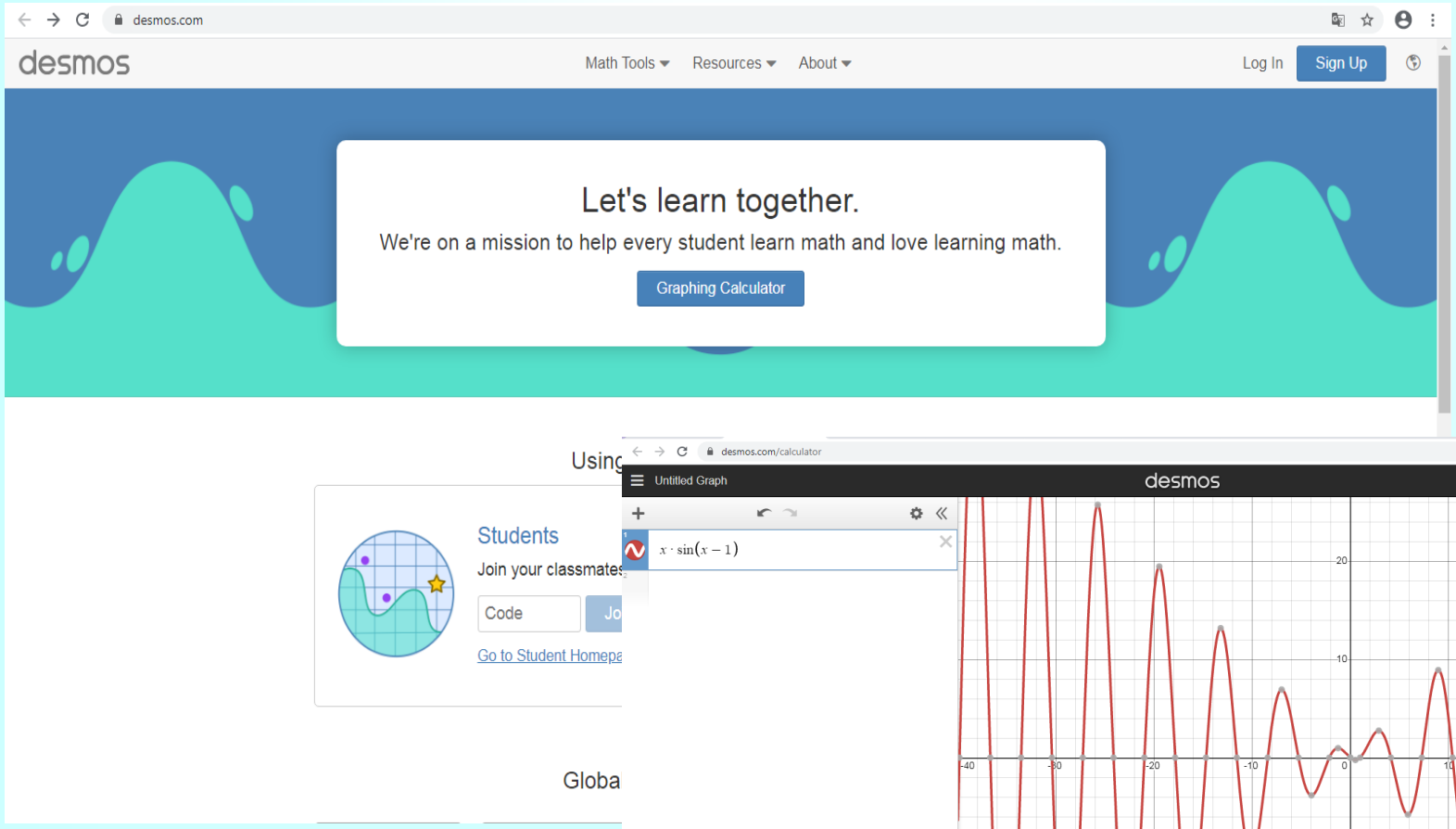
Google kalkulator

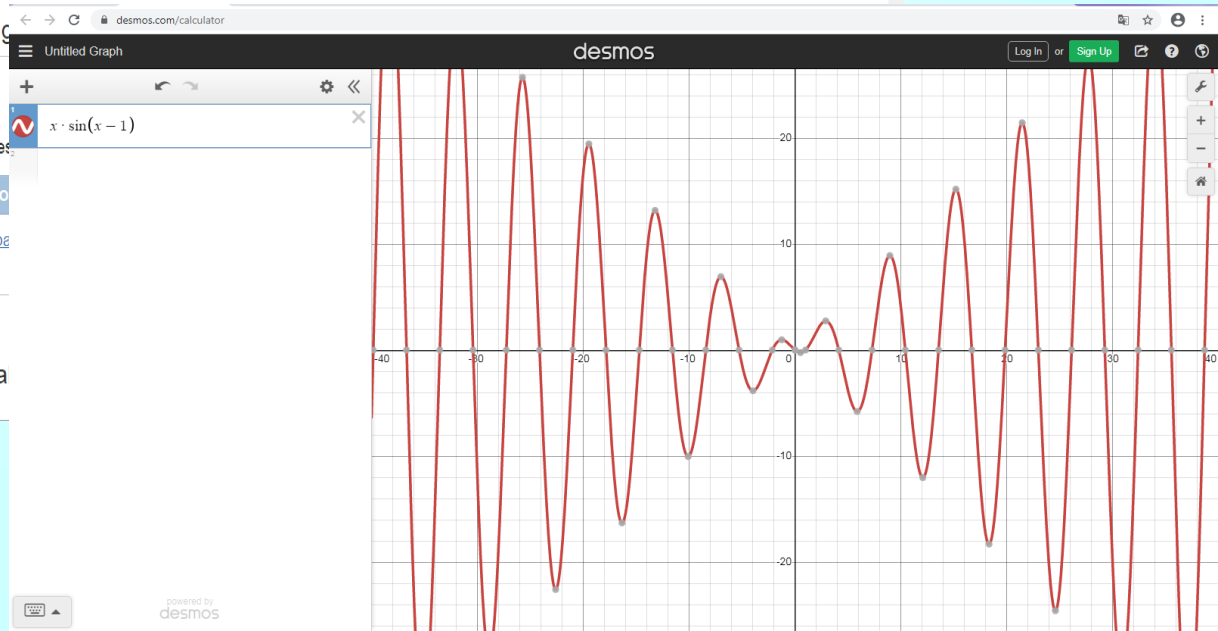




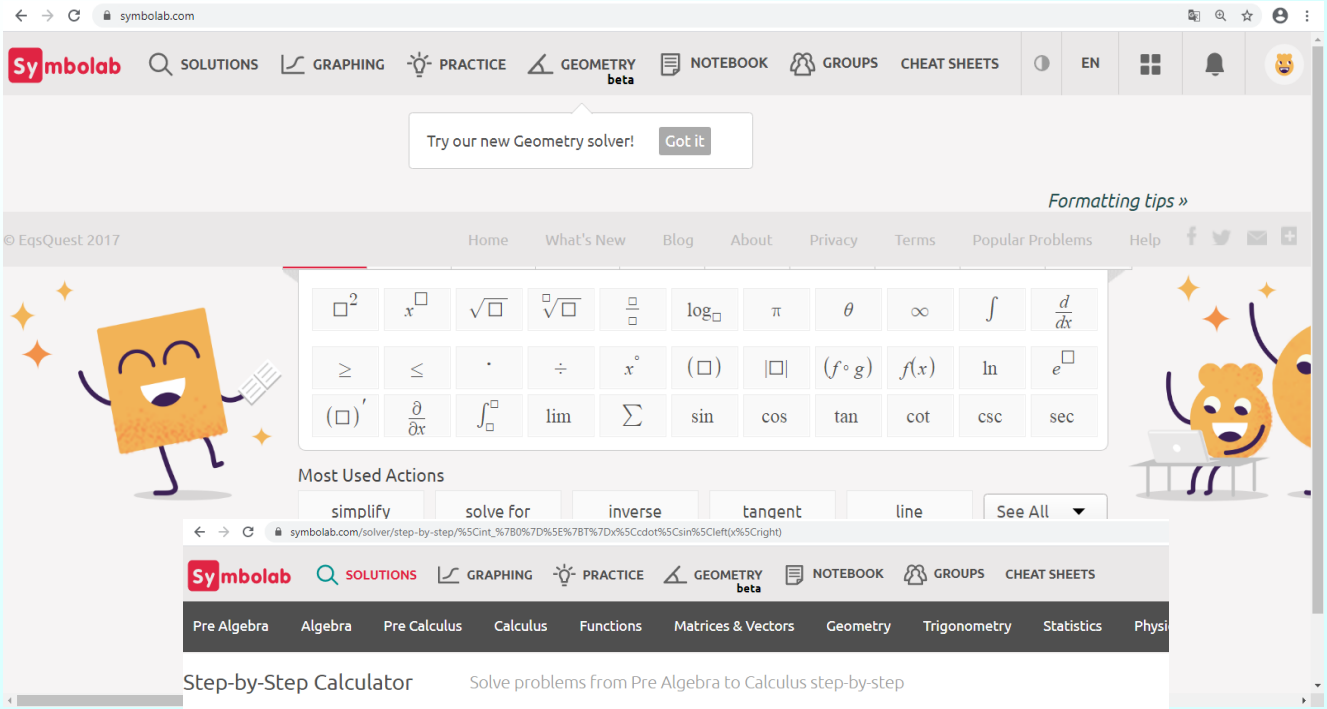
Graphing Calculator – Desmos

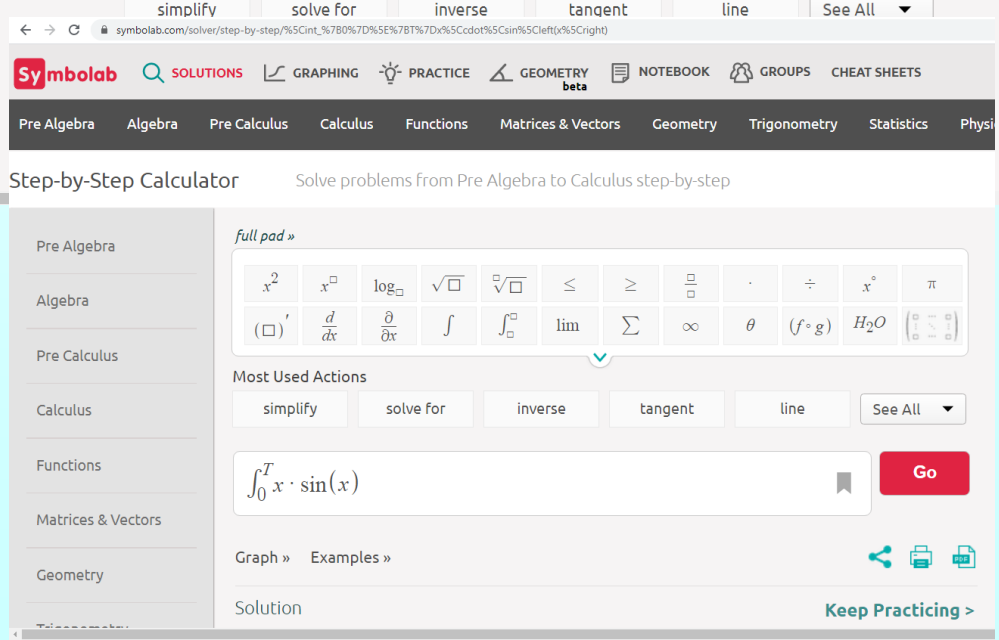
https://www.desmos.com/calculator

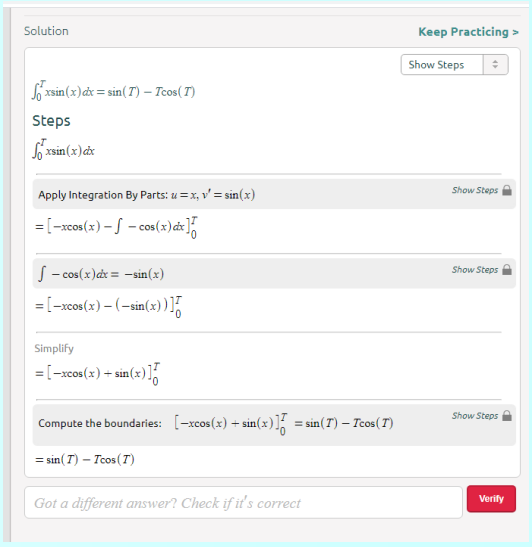


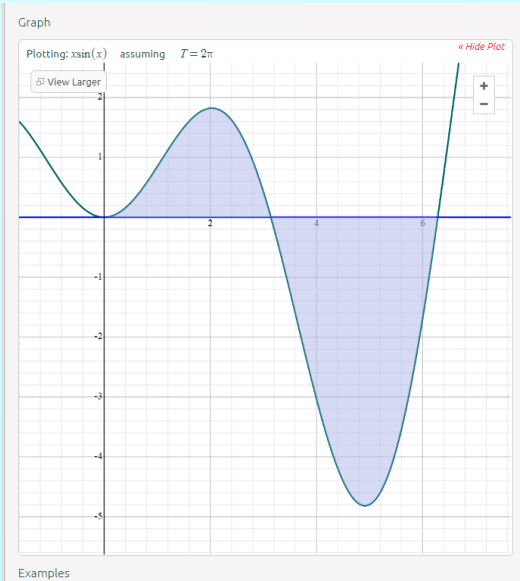


Symbolab Math Solver - Step by Step calculator https://www.symbolab.com/



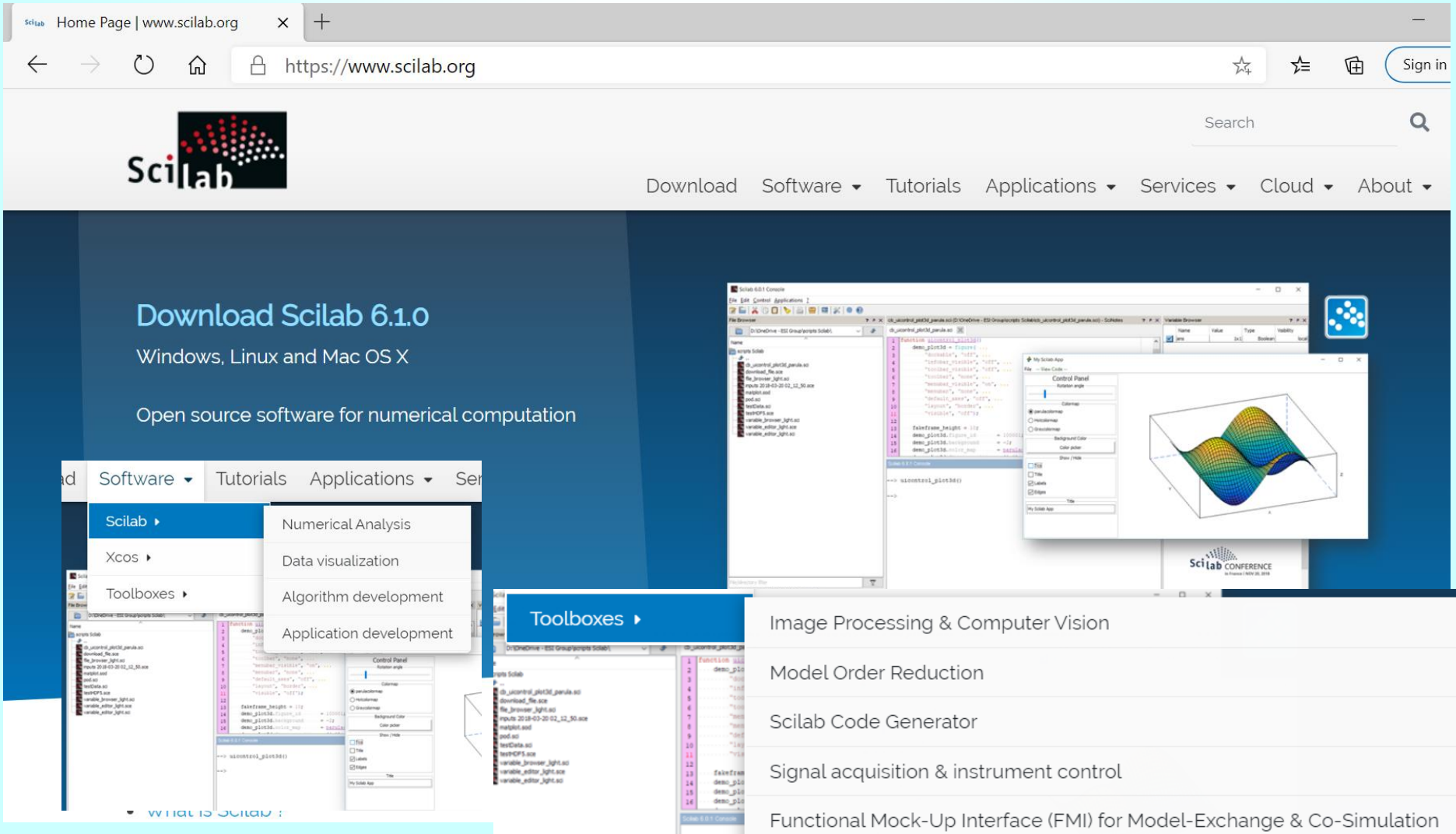






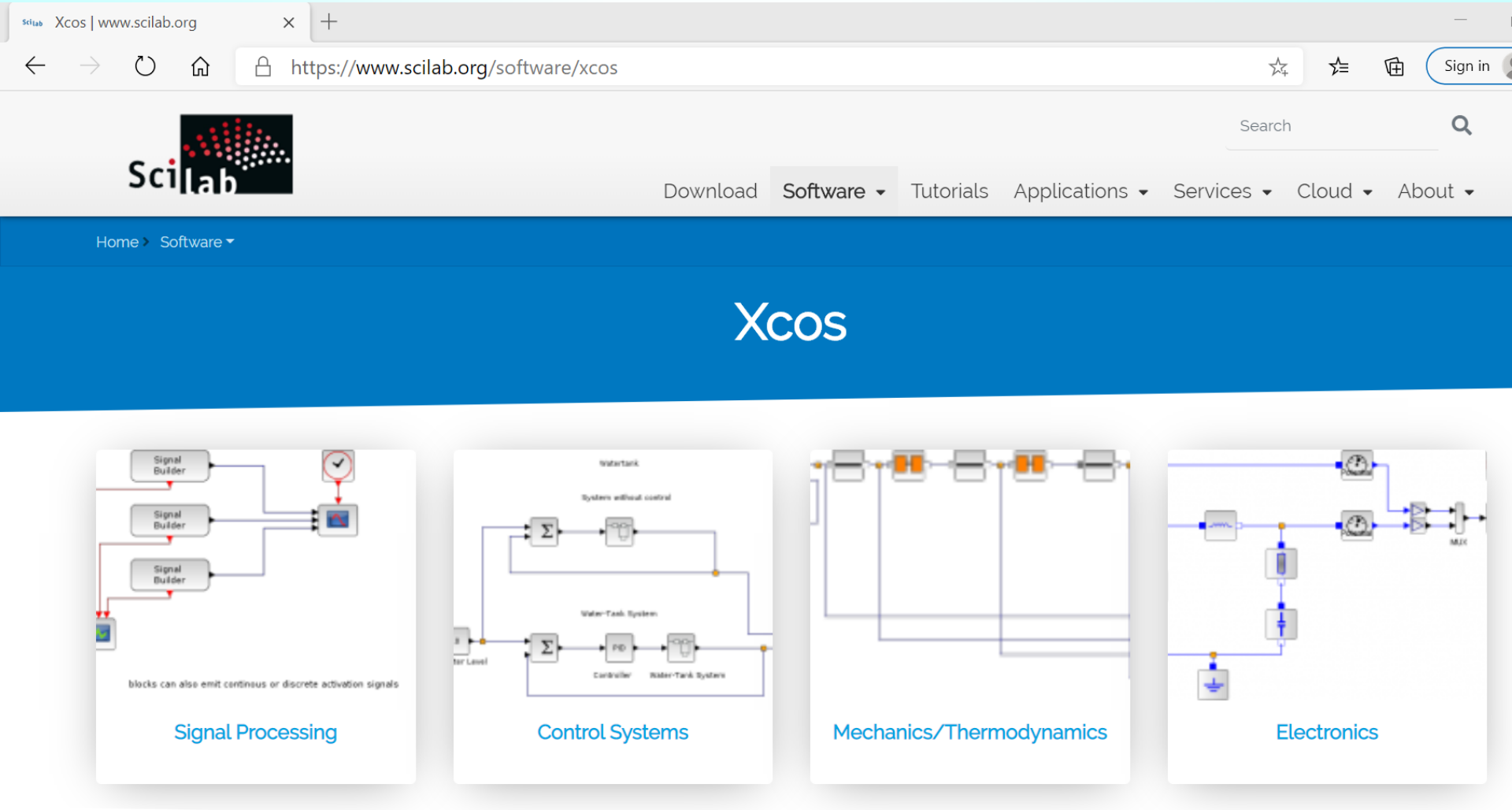
SciLab

https://www.scilab.org/



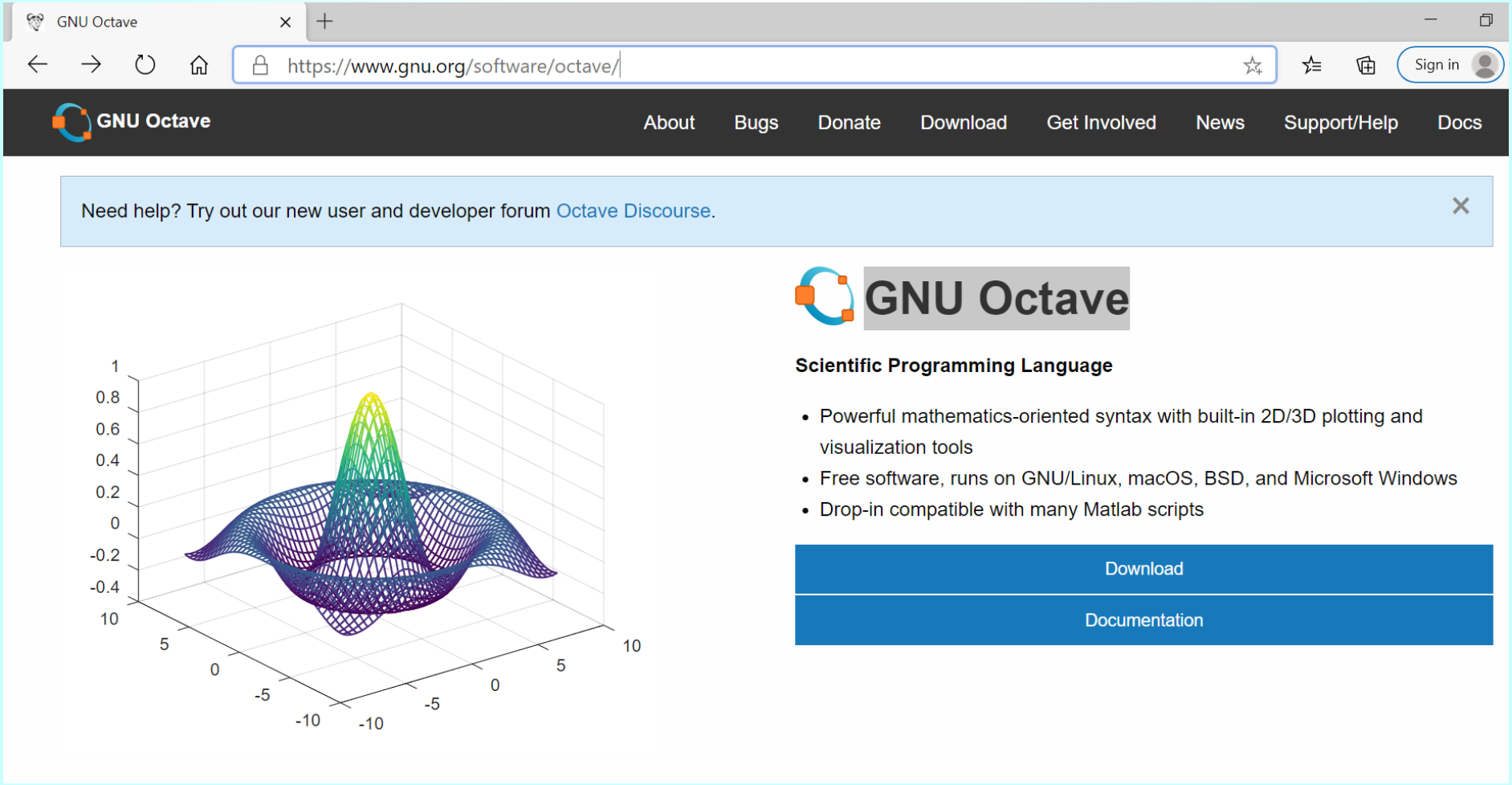
Xcos

https://www.scilab.org/software/xcos



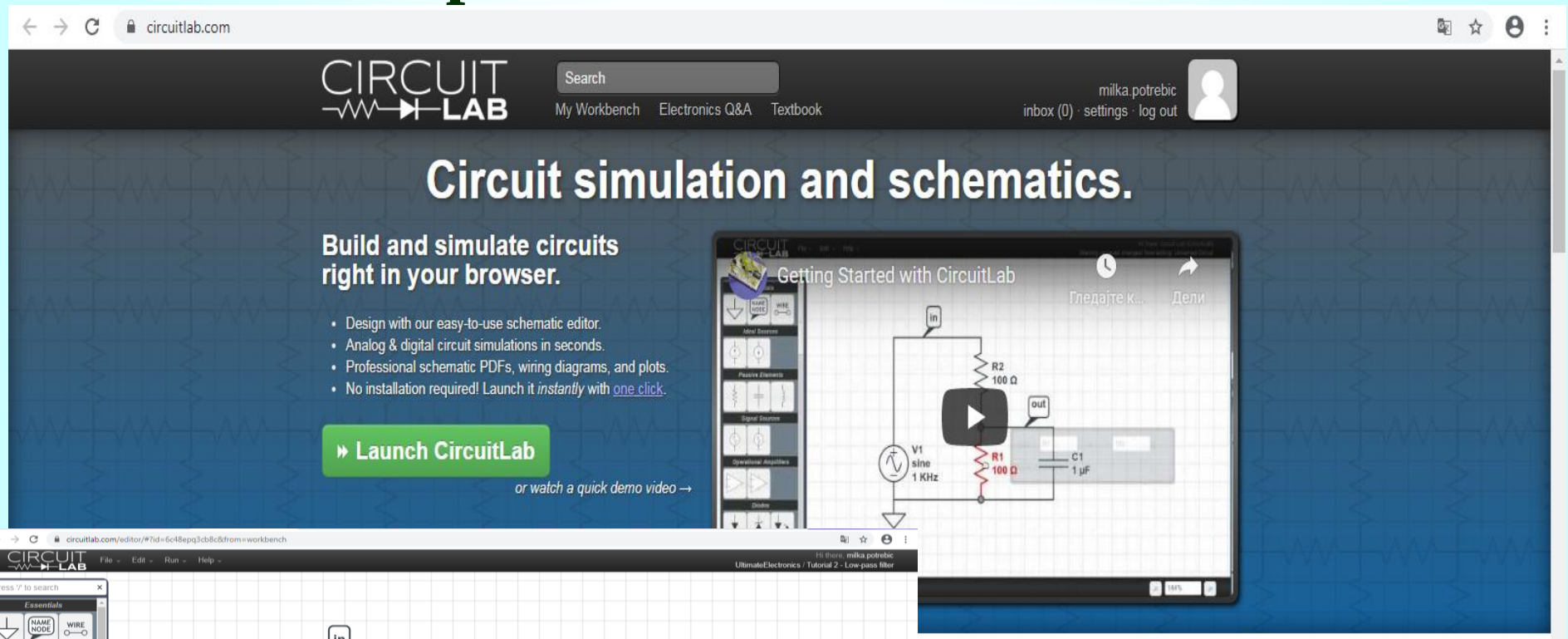
GNU Octave

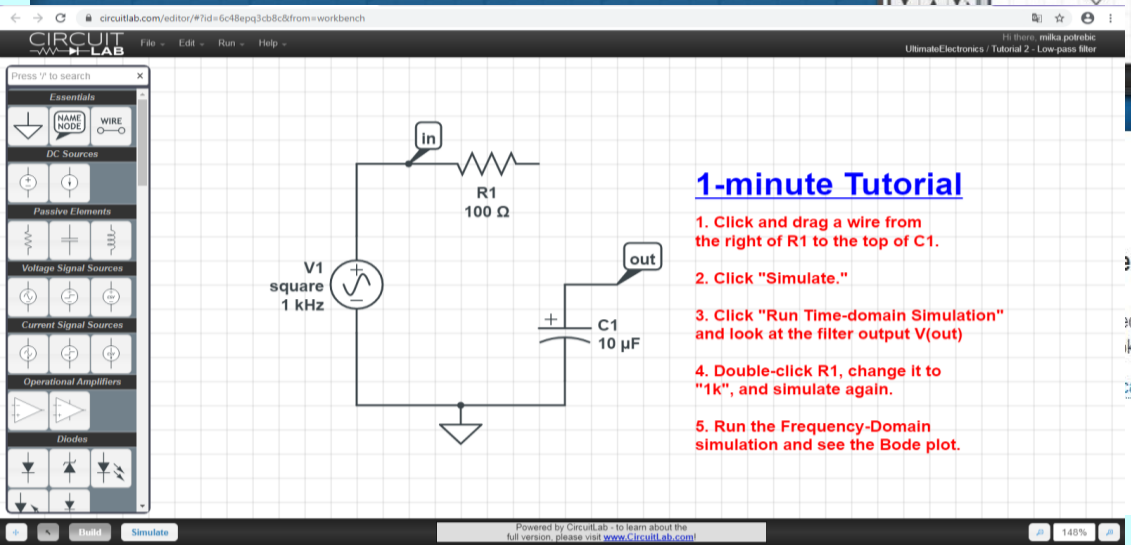
https://www.gnu.org/software/octave/

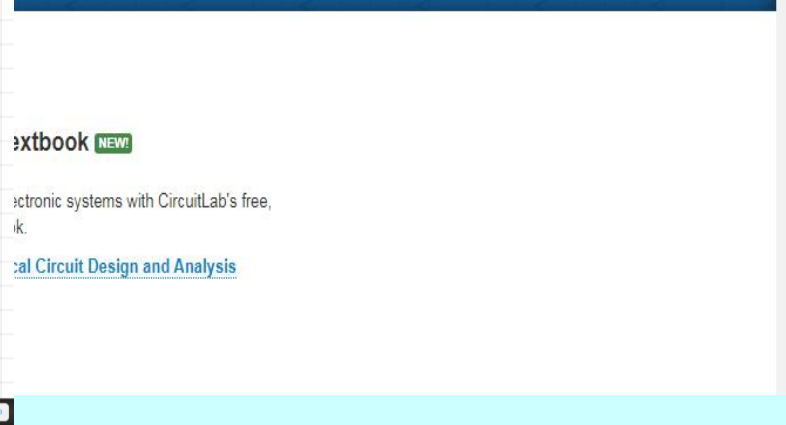


CircuitLab

https://www.circuitlab.com/







DoCircuits Circuit Simulator

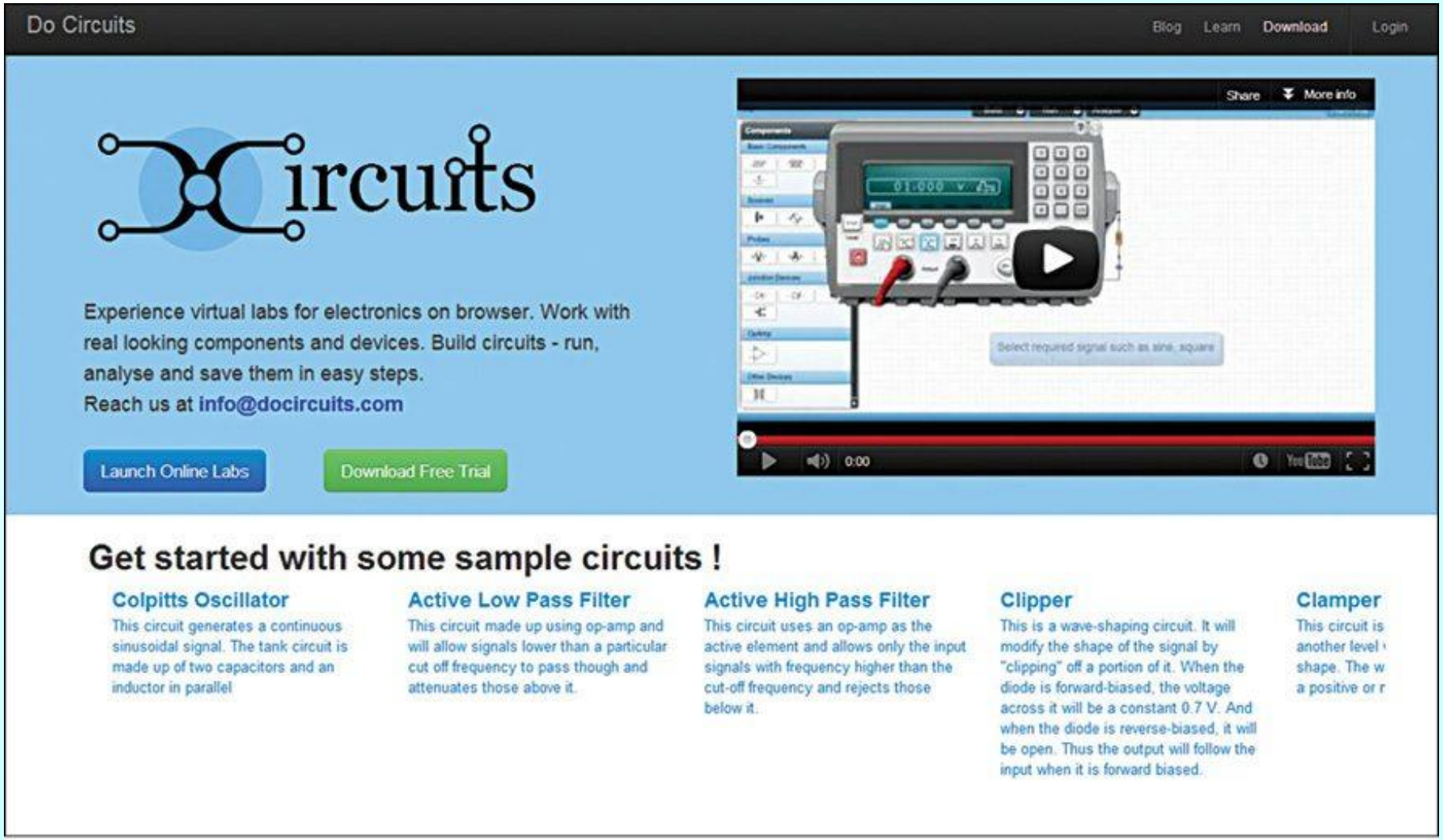


Fig. 1: DoCircuits - Home

ngSpice

http://ngspice.sourceforge.net/

Not secure | ngspice.sourceforge.net

NGSPICE

MIXED MODE - MIXED LEVEL CIRCUIT SIMULATOR

BASED ON BERKELEY'S SPICE3F5

NGSPICE SUMMARY

Home | News | Screenshots | Download | Documentation | Tutorials | Extras/Options | Applications | Development | Simulation Environments | Recipes

Ngspice Home

- Home
- What is ngspice ?
- Features, Extras & Options
- F.A.Q.
- Tutorials

Sourceforge Developer Pages

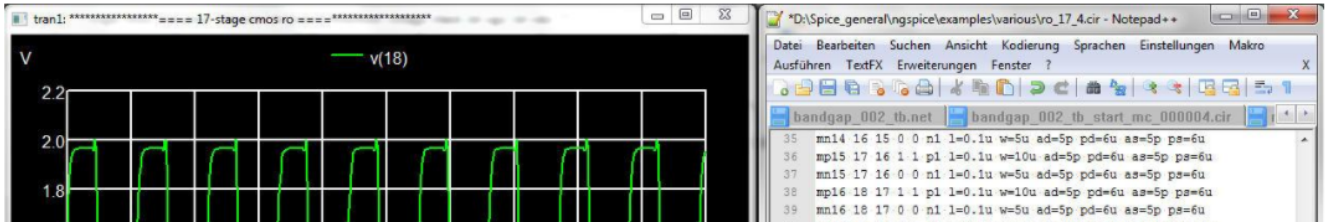
ngspice - open source spice simulator

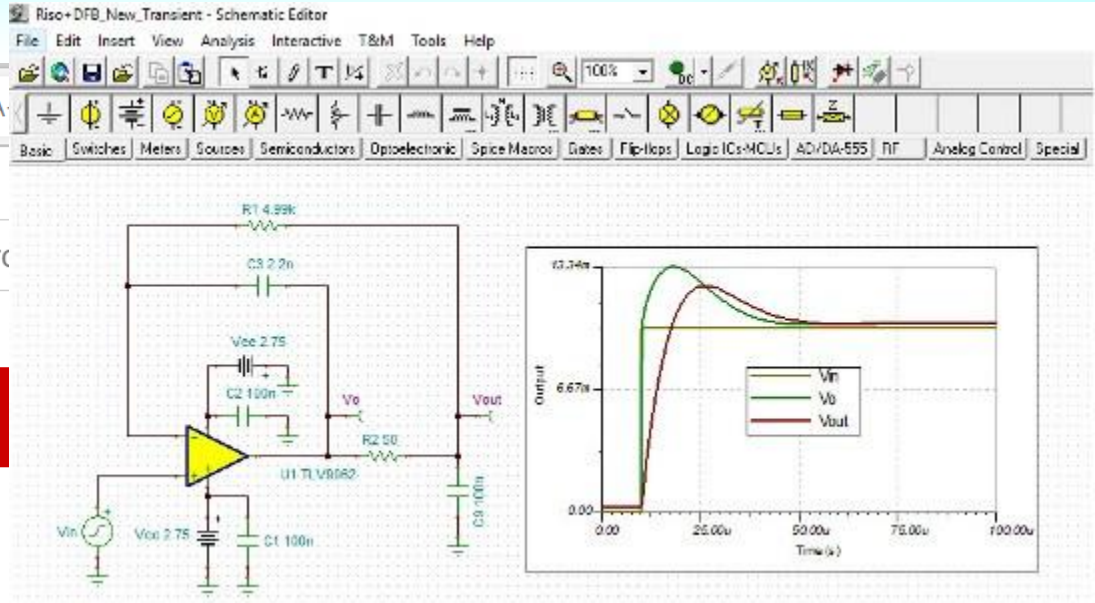
ngspice is the open source spice simulator for electric and electronic circuits.

Such a circuit may comprise of JFETs, bipolar and MOS transistors, passive elements like R, L, or C, diodes, transmission lines and other devices, all interconnected in a netlist. Digital circuits are simulated as well, event driven and fast, from single gates to complex circuits. And you may enter the combination of both analog and digital as a mixed-signal circuit.

ngspice offers a wealth of device models for active, passive, analog, and digital elements. Model parameters are provided by our collections, by the semiconductor device manufacturers, or from semiconductor foundries. The user adds her circuits as a netlist, and the output is one or more graphs of currents, voltages and other electrical quantities or is saved in a data file.

ngspice does not provide schematic entry. Its input is command line or file based. There are however third party interfaces available.





https://www.ti.com/tool/TINA-

TEXAS INSTRUMENTS

Search

Products Applications Design resources

Home / Design resources

TINA-TI

SPICE-based analog simulation program

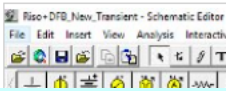
Overview | Downloads | Technical documentation | Support & training

TINA-TI

https://www.ti.com/tool/TINA-TI

Overview

TINA-TI provides all the conventional DC, transient and frequency domain analysis of SPICE and much more. TINA has extensive post-processing capability that allows you to format results the way you want them. Virtual instruments allow you to



Electric Circuit Studio

https://play.google.com/store/apps/details?id=com.ecstudiosystems.electriccircuitstudio&hl=en_US&gl=US

Google Play

Search

Apps

Categories

Home

Top charts

New releases

My apps

Shop

Games

Kids

Editors' Choice

Account

Payment methods

Play Points New

My subscriptions

Redeem

Buy gift card

My wishlist

My Play activity

Parent Guide

Electric Circuit Studio

ECStudio Systems Education

2,709

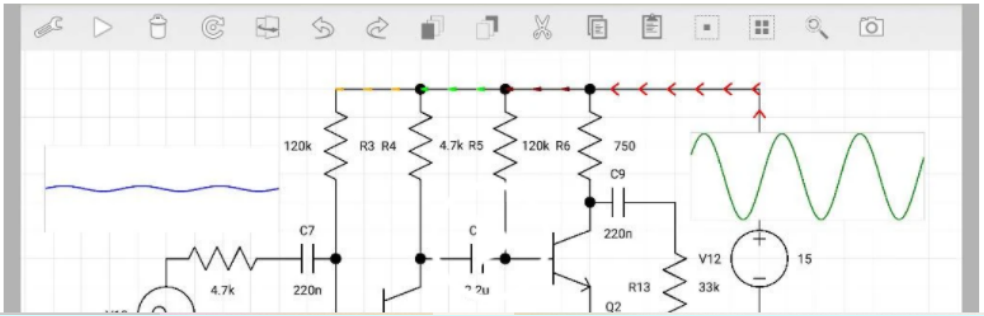
Everyone

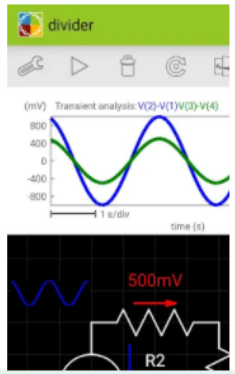
Contains Ads

You don't have any devices

Add to Wishlist

Install





CircuitSafari SPICE Simulator

https://play.google.com/store/apps/details?id=com.logipipe.circuitsafari&hl=en_US&gl=US

Google Play

Search

Apps

Categories

Home

Top charts

New releases

My apps

Shop

Games

Kids

Editors' Choice

Account

Payment methods

Play Points New

My subscriptions

Redeem

Buy gift card

My wishlist

My Play activity

Parent Guide

CircuitSafari SPICE Simulator (Early Access)

Logipipe, LLC Productivity

Everyone

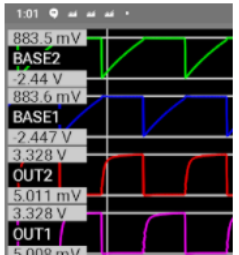
Offers in-app purchases

This app is in development. It may be unstable.

You don't have any devices

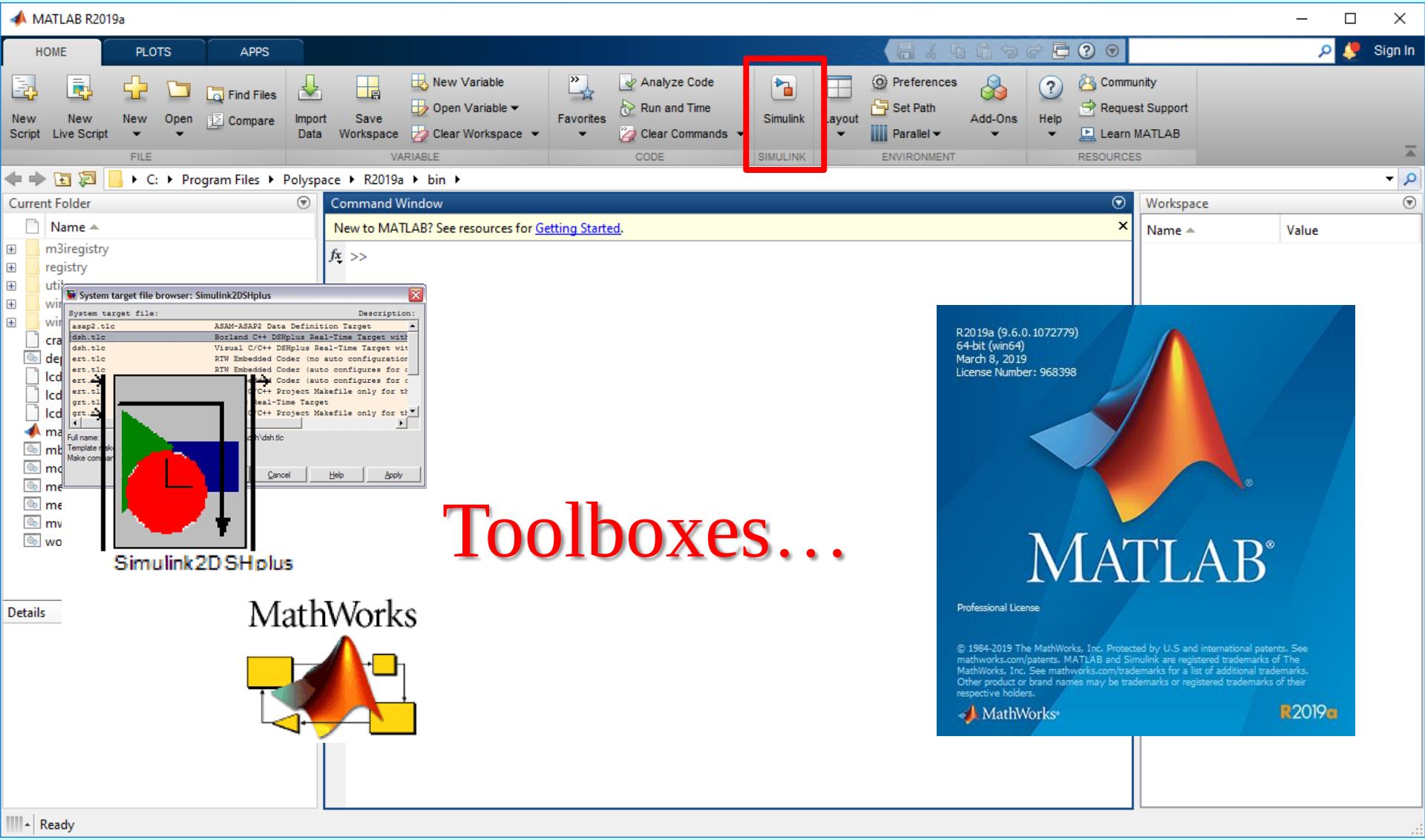
Add to Wishlist

Install



MATLAB

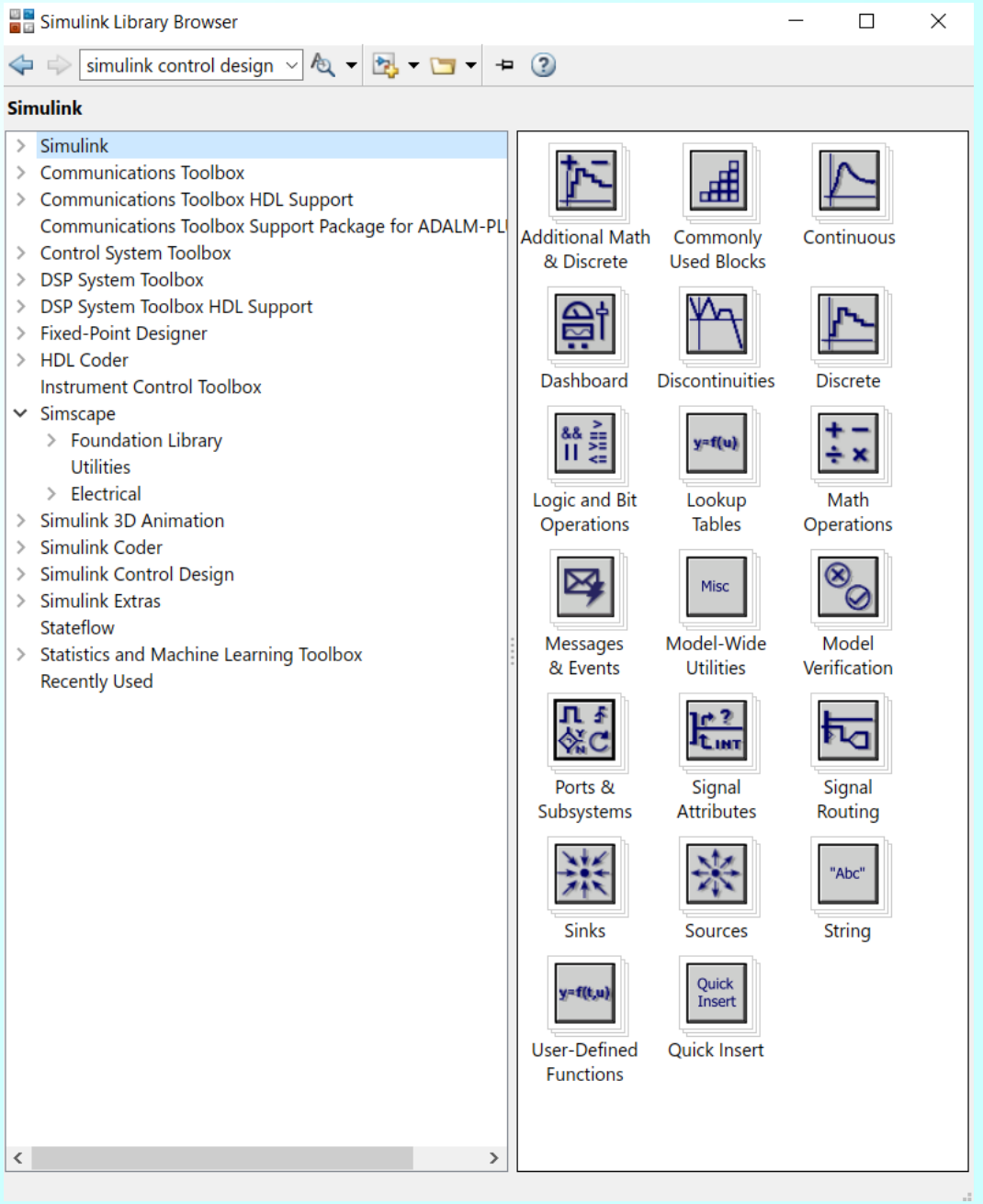
http://www.mathworks.com/



Toolboxes…

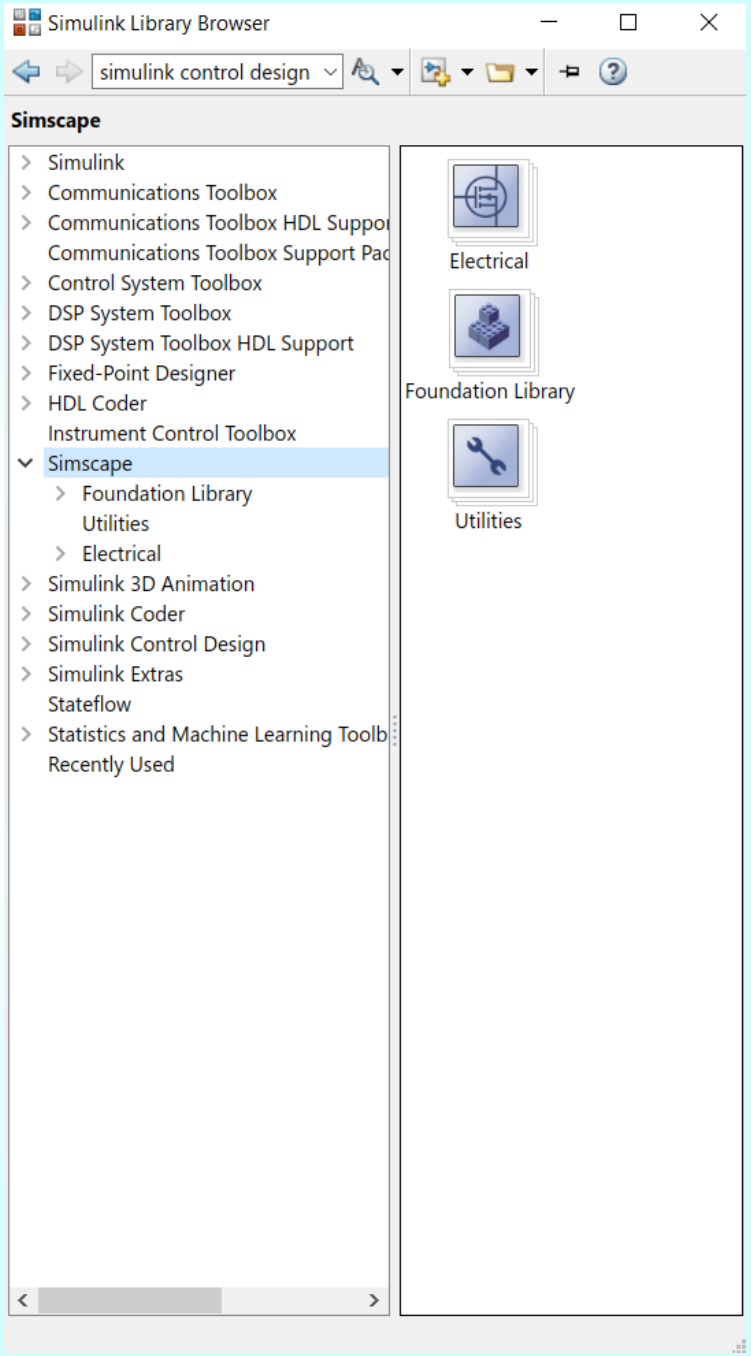
MATLAB Simulink

- Алатка за нумеричку анализу математичких модела
- Систем се дефинише графички, уметањем блокова и њиховим повезивањем у мрежу



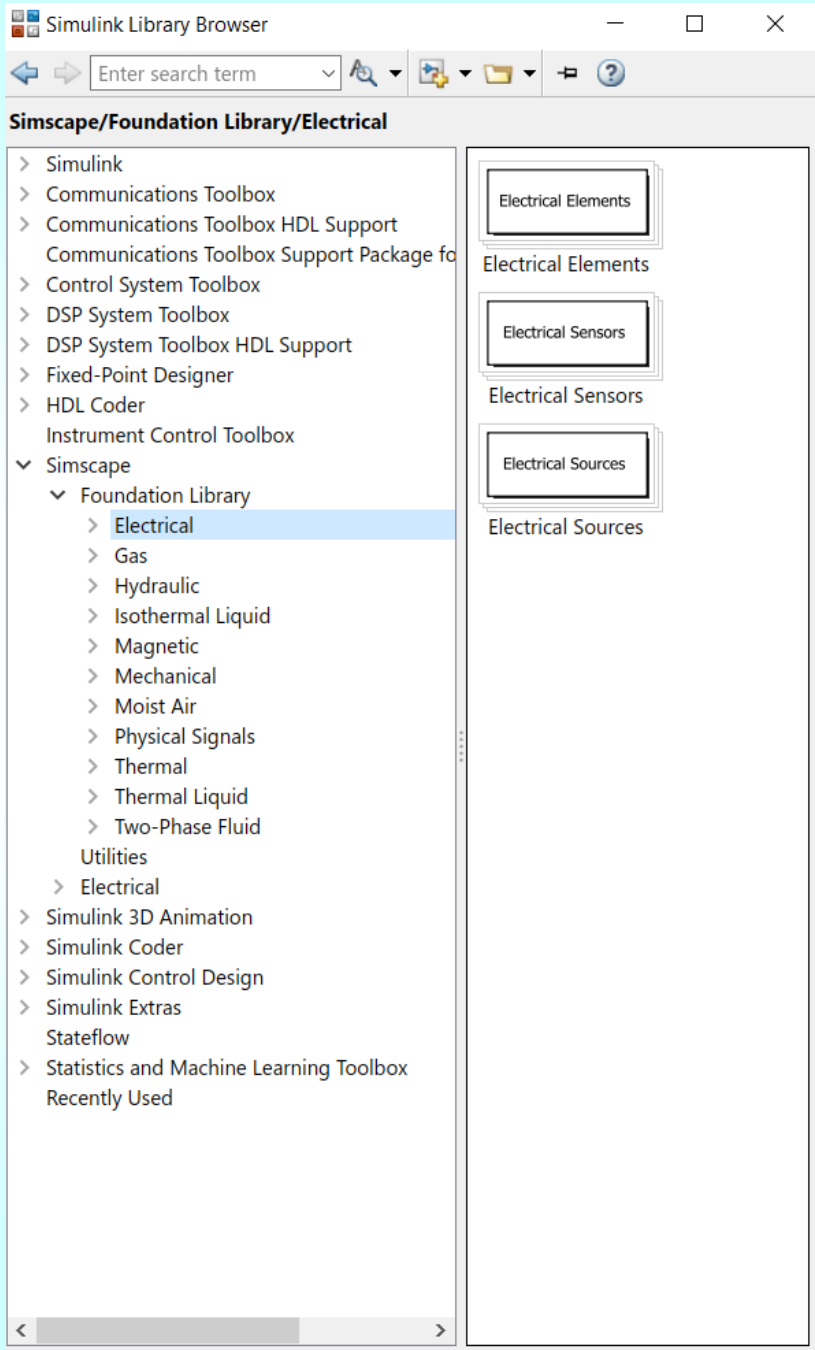
Simulink > Simscape

- Библиотека за симулацију и нумеричку анализу мултидоменских проблема (механика, електродинамика, оптика)



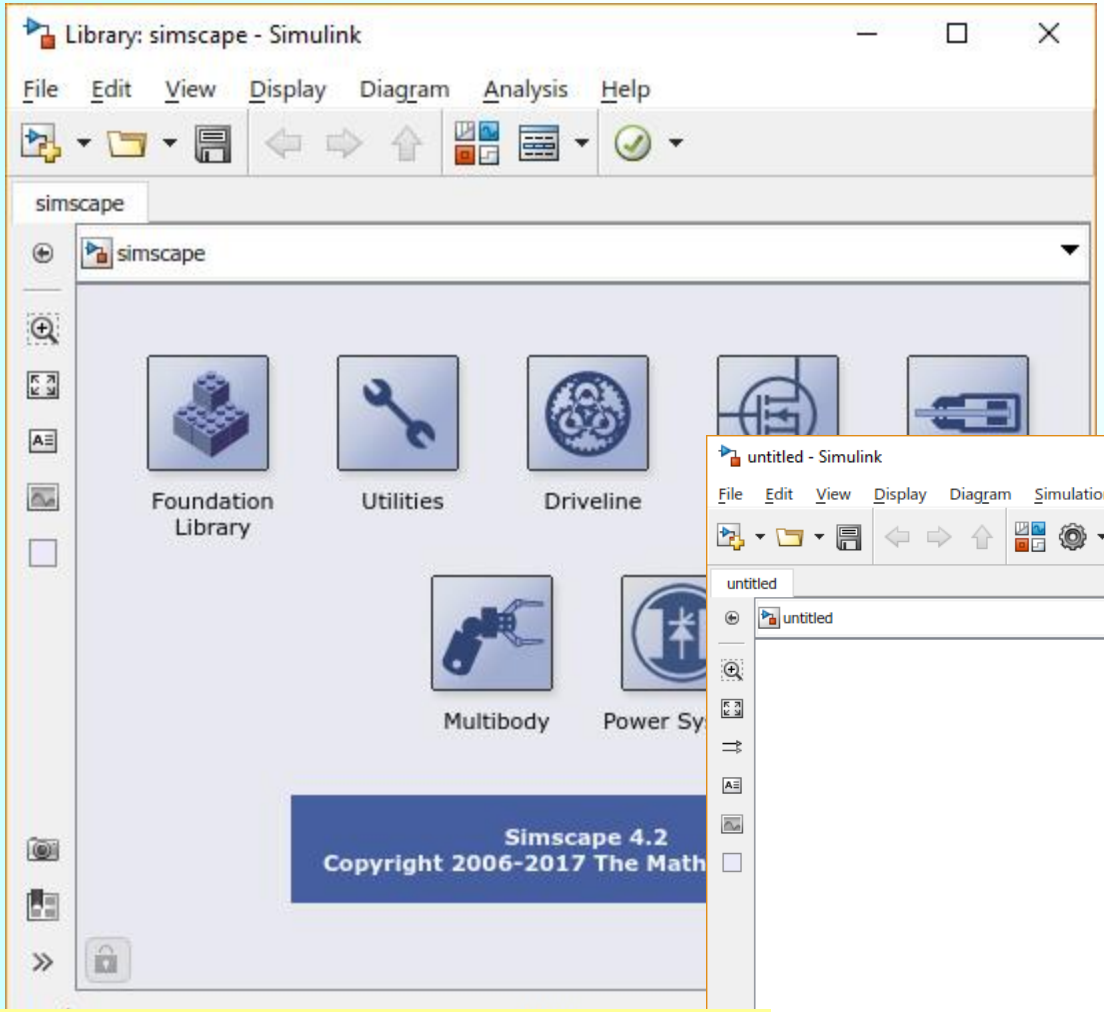
Simulink > Simscape > Foundation Library > Electrical

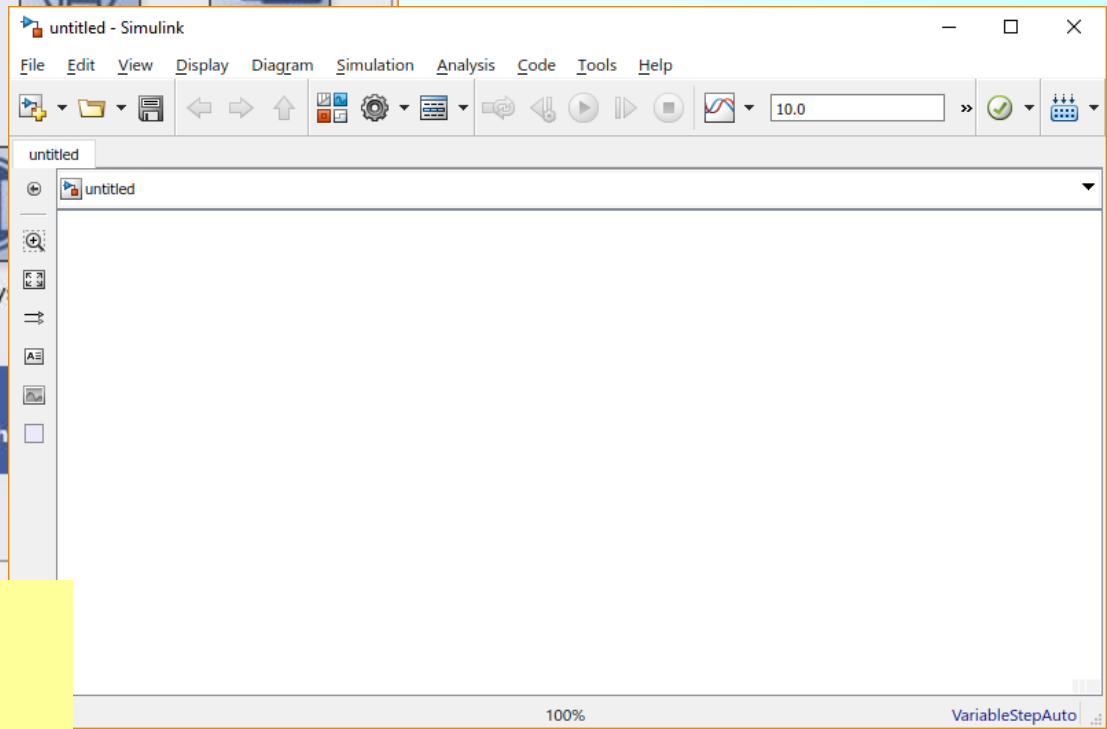
- Симулација линеарних и нелинеатних електричних кола



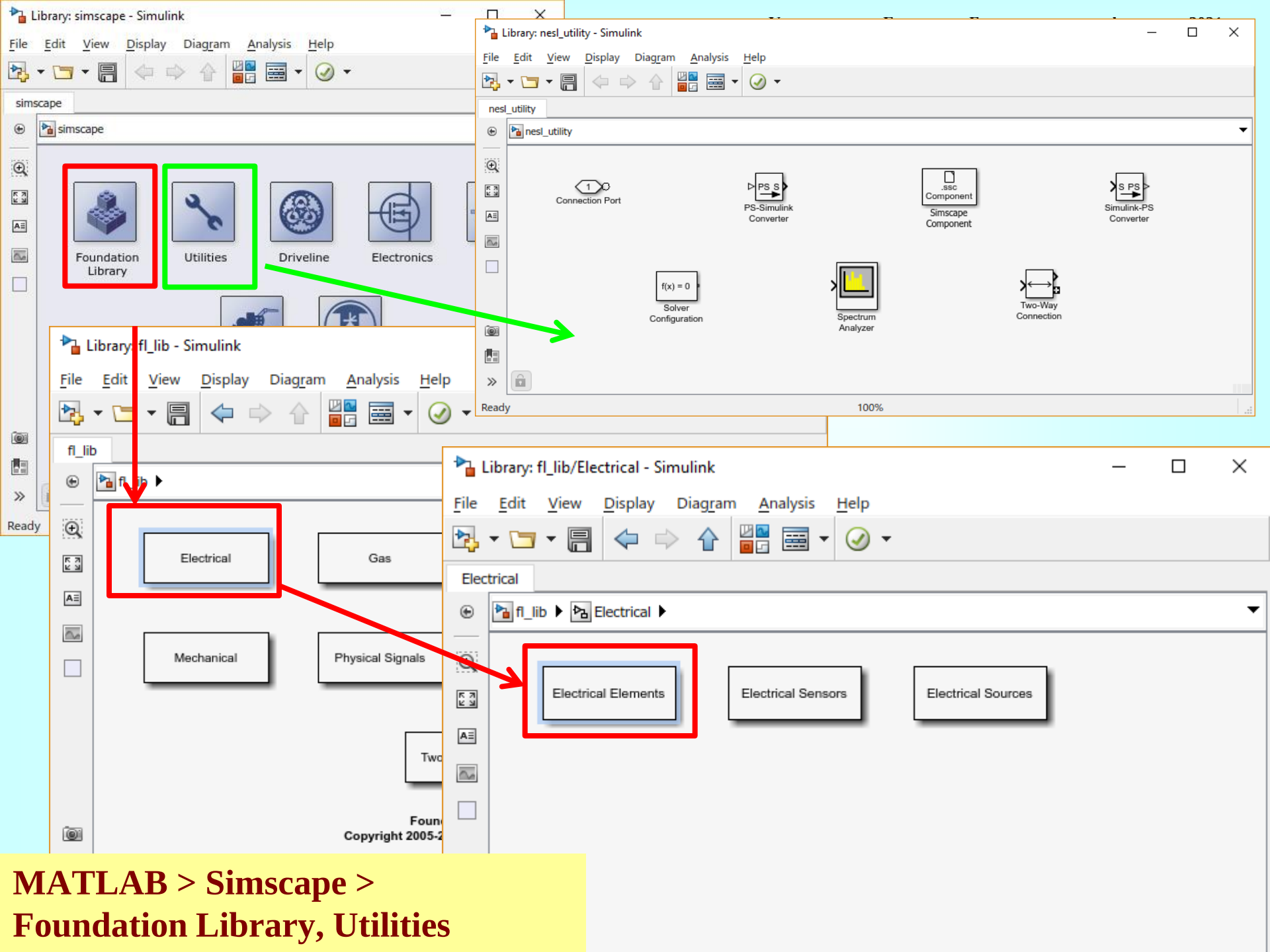
MATLAB > Simscape > Foundation Library, Utilities

Симулација електричног кола коришћењем **Simscape > Foundation Library** библиотеке

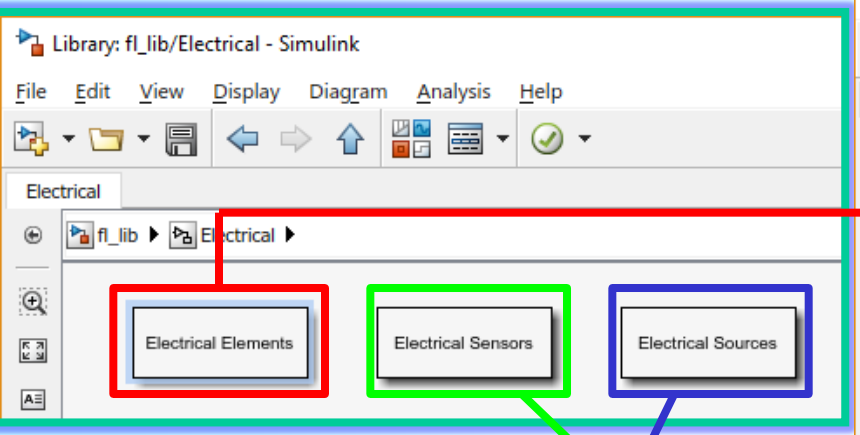




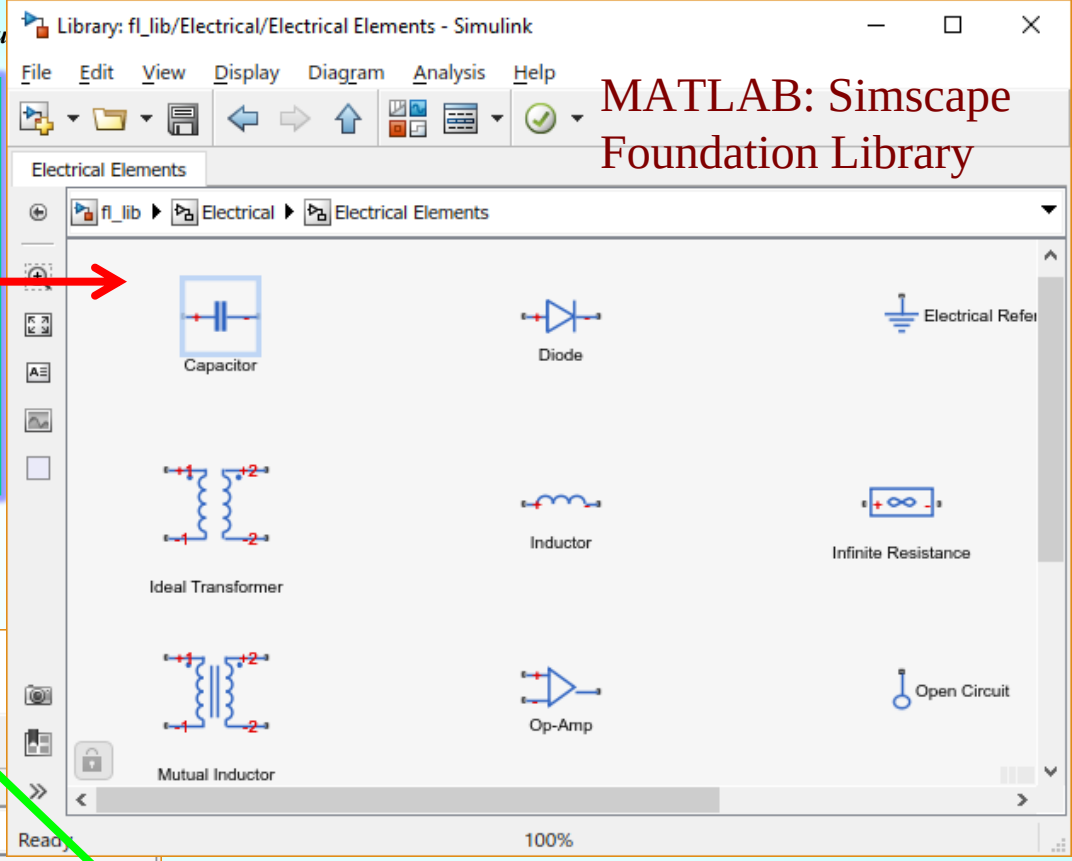
MATLAB > Simscape > Foundation Library, Utilities

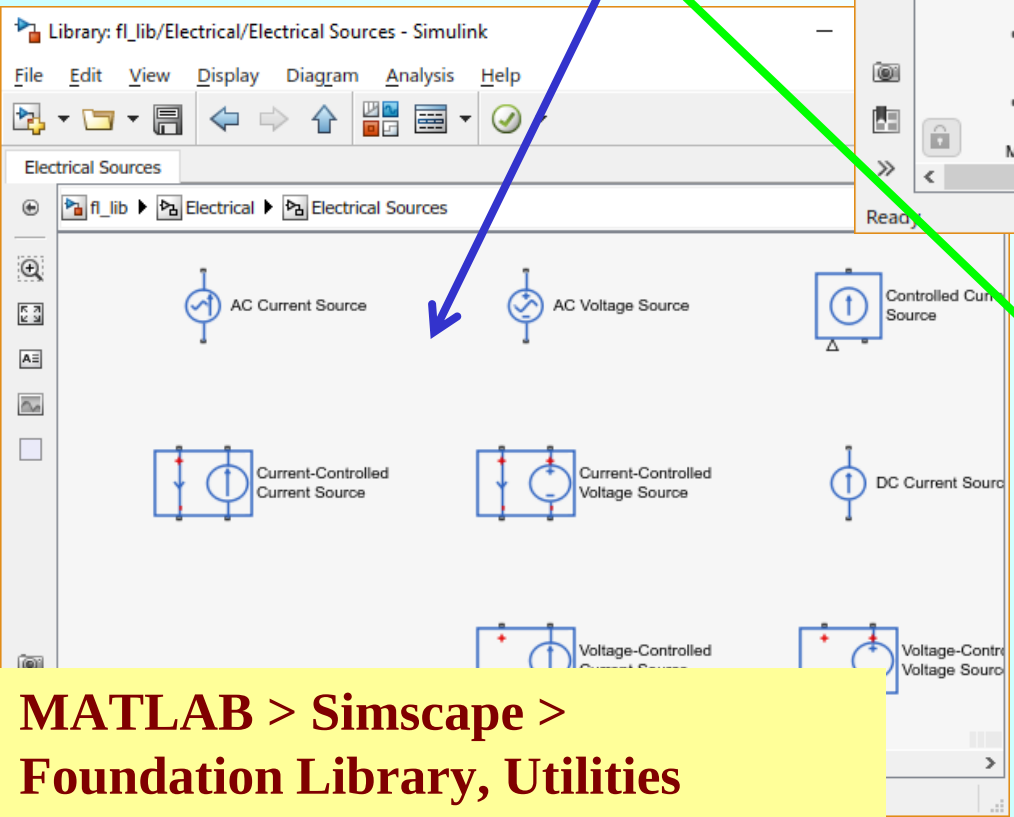


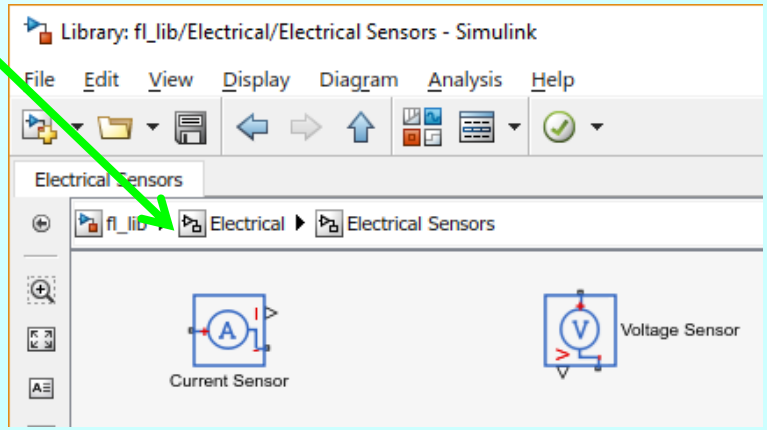
MATLAB > Simscape > Foundation Library, Utilities



MATLAB: Simscape Foundation Library

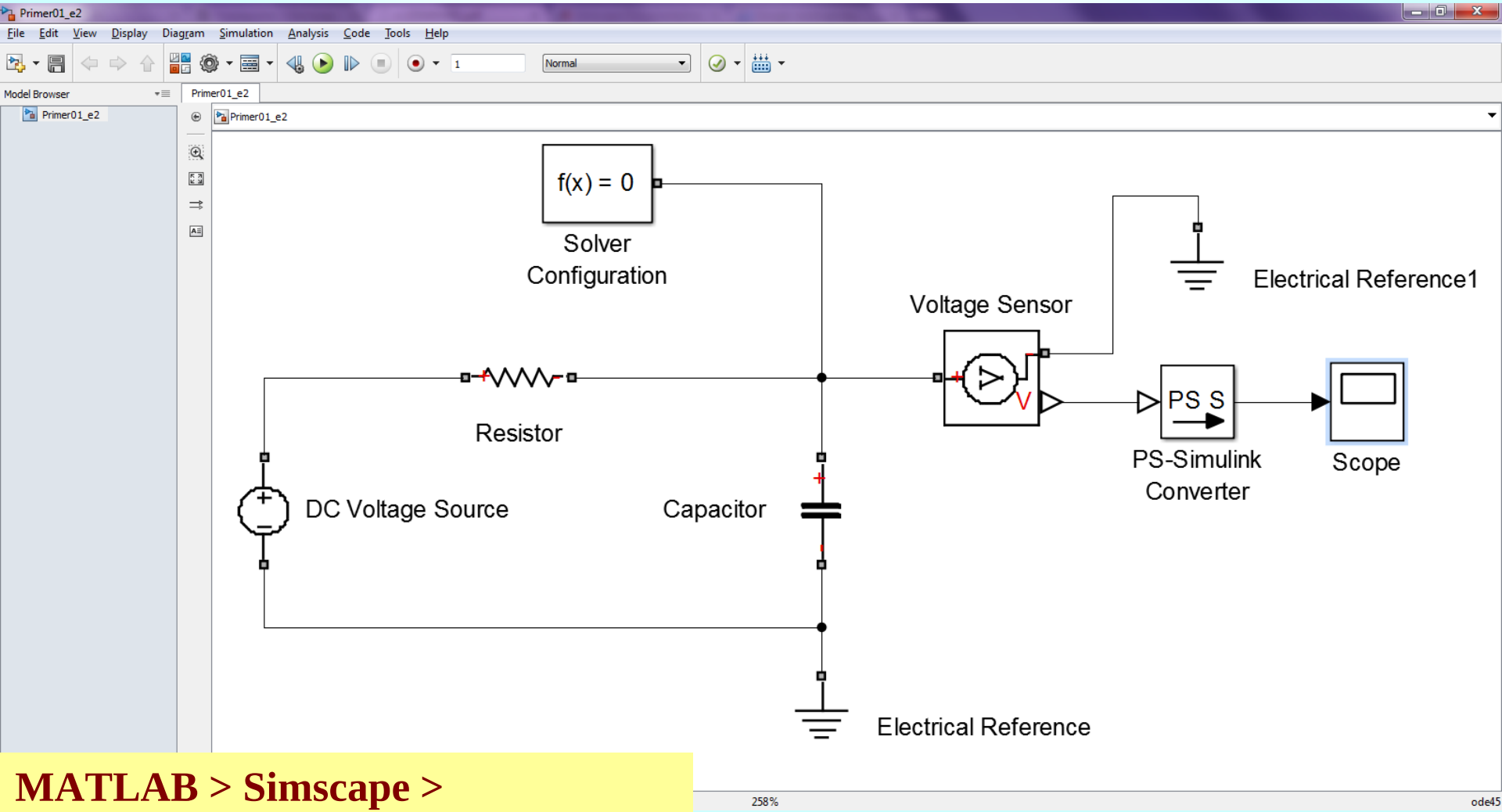




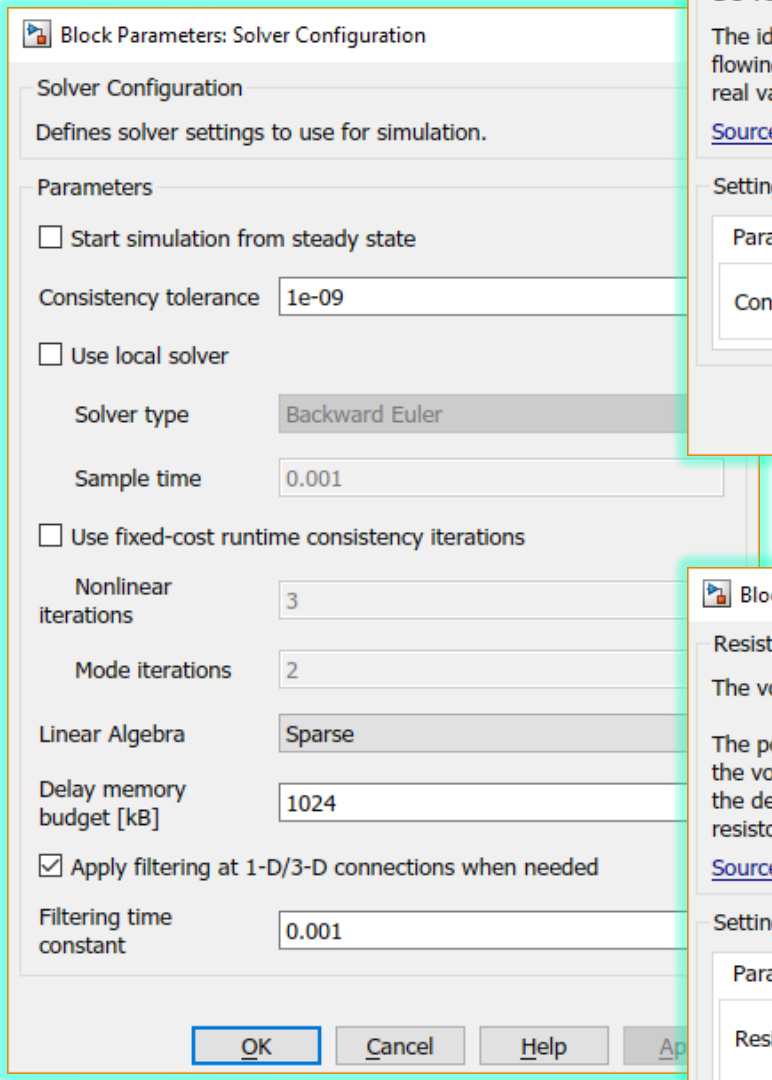


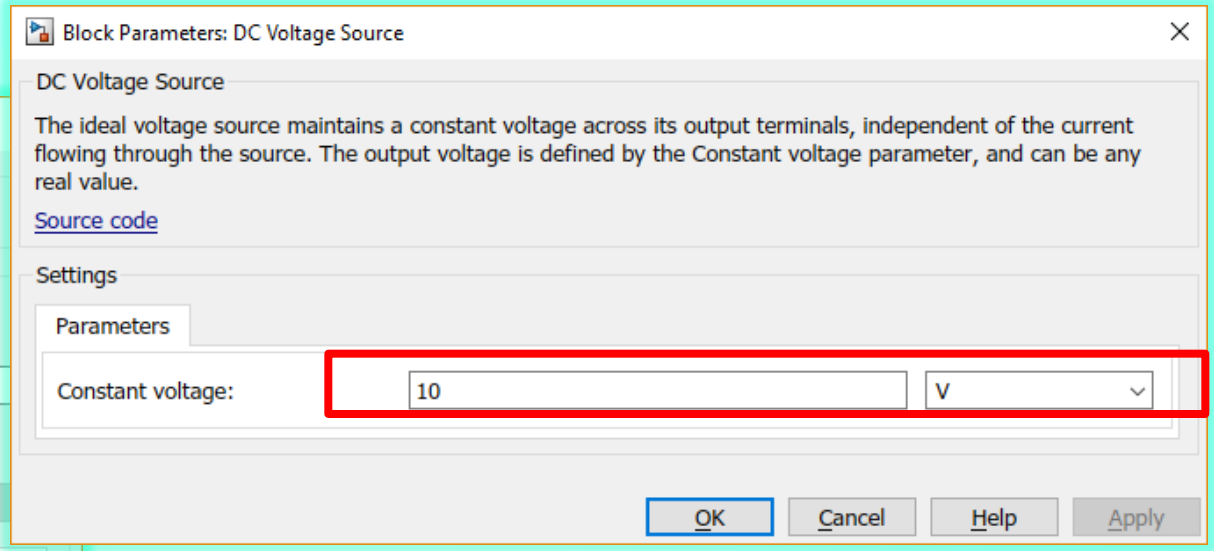
MATLAB > Simscape > Foundation Library, Utilities

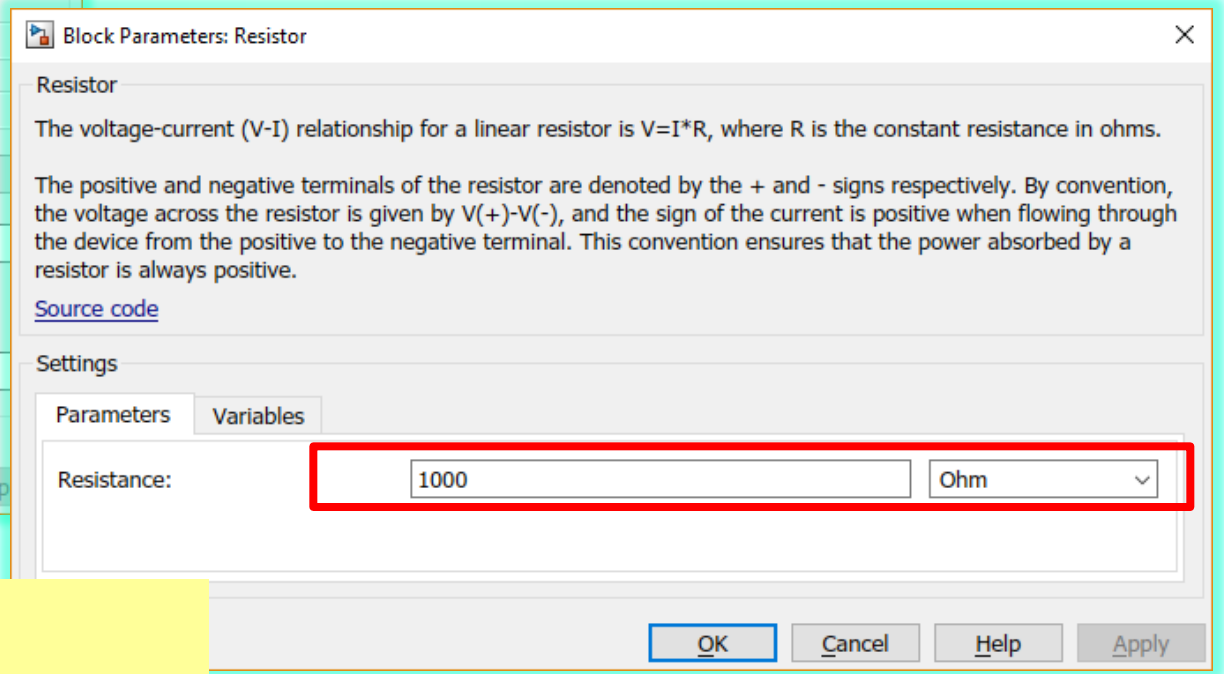
Симулација електричног кола коришћењем **Simscape > Foundation Library** библиотеке



MATLAB > Simscape > Foundation Library, Utilities







MATLAB > Simscape > Foundation Library, Utilities

Почетни услов...

Block Parameters: Capacitor

Capacitor

Models a linear capacitor. The relationship between voltage V and and current I is I=C*dV/dt where C is the capacitance in farads.

The Series resistance and Parallel conductance represent small parasitic effects. The parallel conductance can be used to model dielectric losses and the series resistance used to represent the effective series resistance (ESR) of the capacitor. Simulation of some circuits may require the presence of the small series resistance. Consult the documentation for further details.

Source code

Settings

Parameters | Variables

Capacitance:	1e-6	F
Series resistance:	0	Ohm
Parallel conductance:	0	1/Ohm

OK | Cancel | Help | Apply

*dV/dt where C is the capacitance in farads.

arallel conductance can be used to model
nce (ESR) of the capacitor. Simulation of
mentation for further details.

Settings

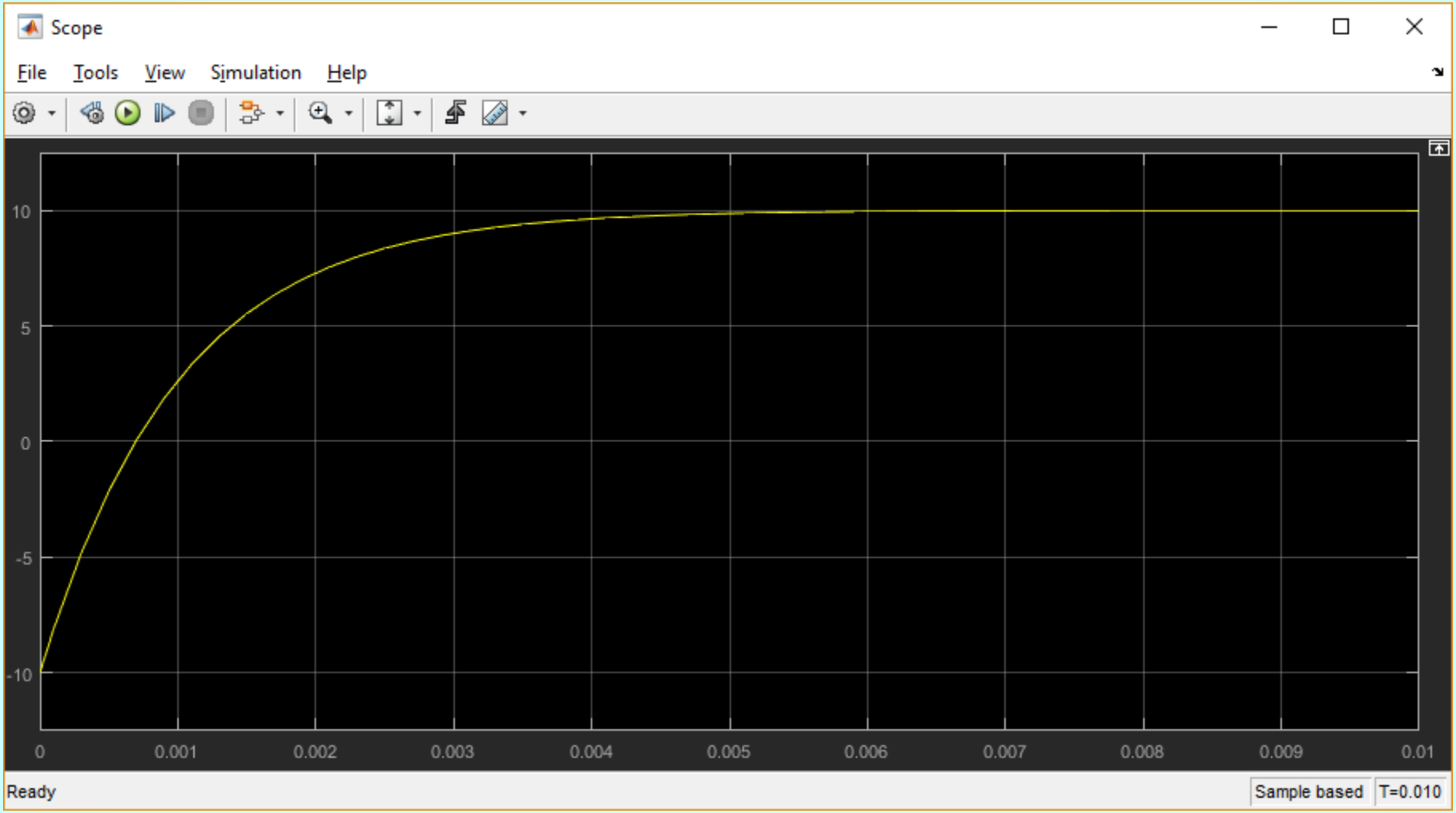
Parameters | Variables

Override	Variable	Priority	Beginning Value	Unit
☐	Current	None	0	A
☐	Voltage	None	0	V
☑	Capacitor voltage	High	-10	V

OK | Cancel | Help | Apply

MATLAB > Simscape > Foundation Library, Utilities

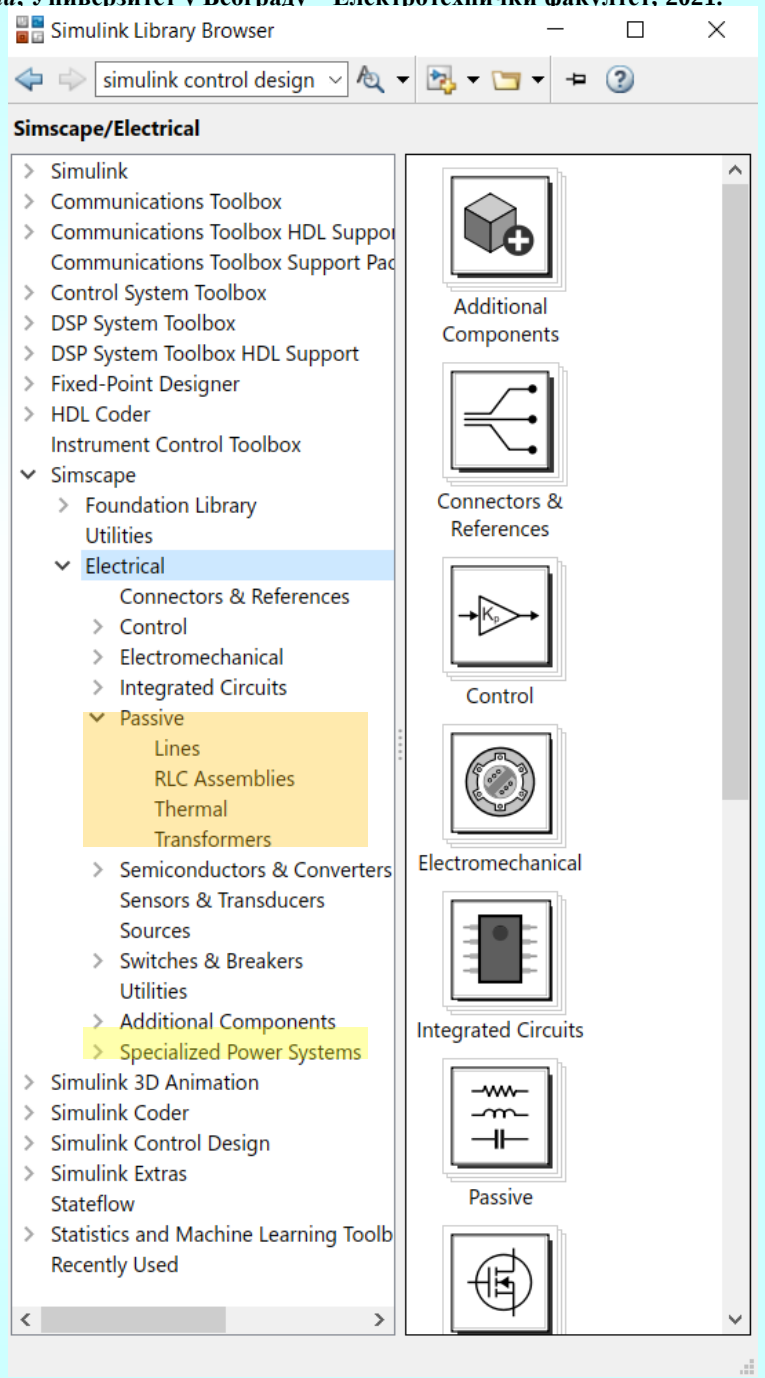
Налажење одзива...



MATLAB > Simscape > Foundation Library, Utilities

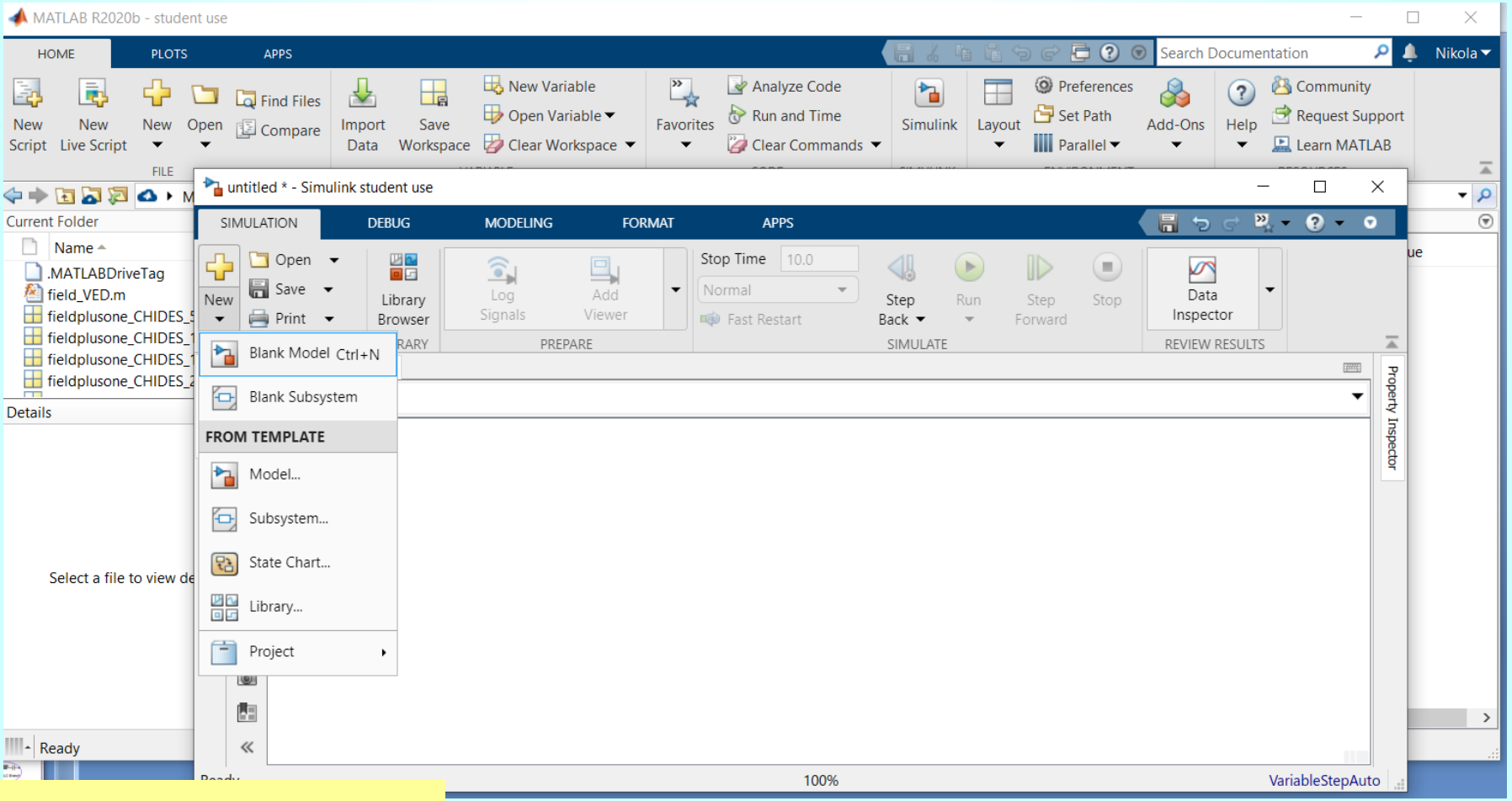
Simulink > Simscape > Electrical

- Библиотека за симулацију и нумеричку анализу електроенергетских система
- Трофазни извори, водови, трансформатори, потрошачи
- Термална анализа



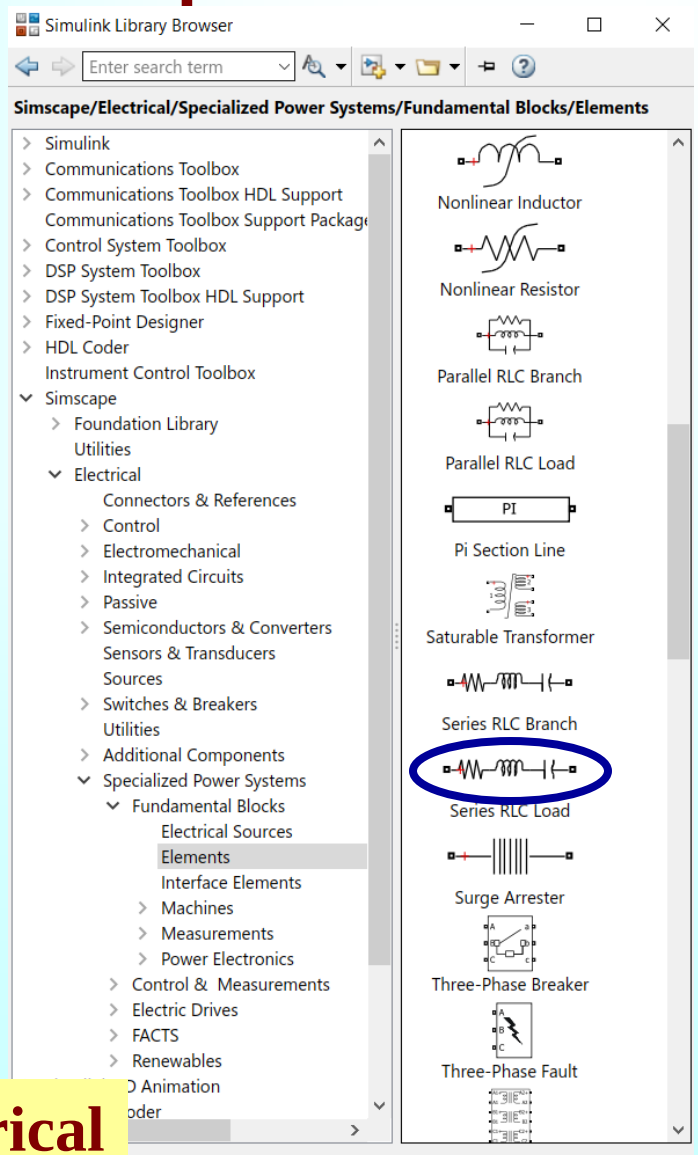
Simscape > Electrical

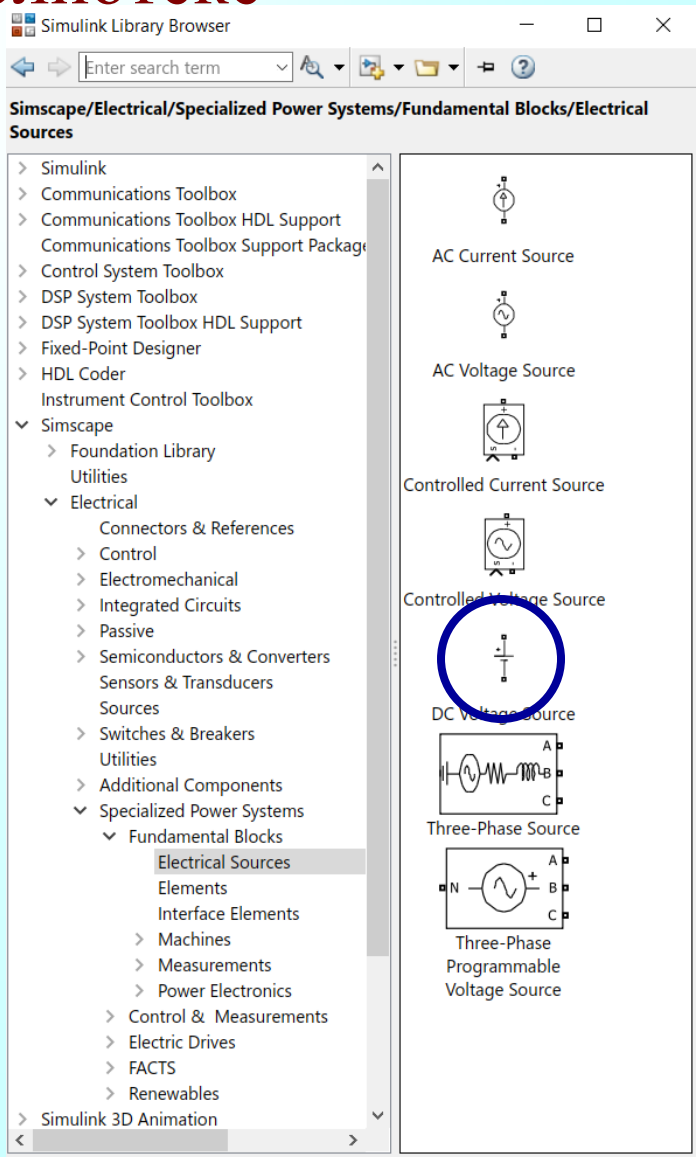
Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке



Simscape > Electrical

Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке

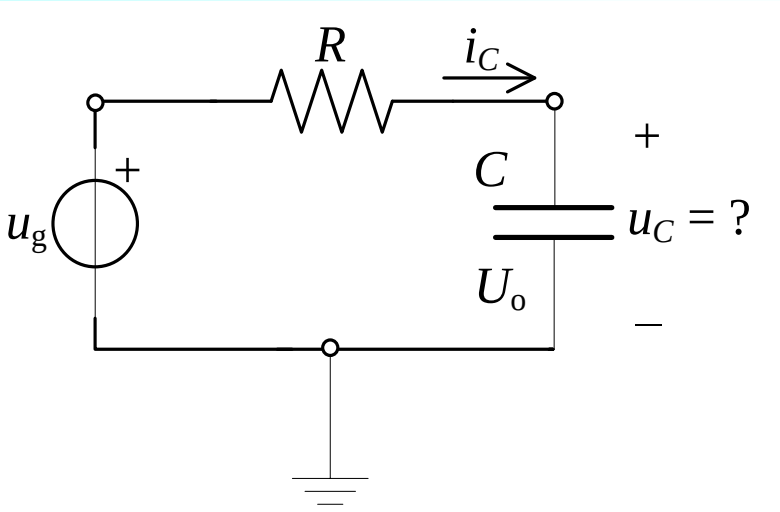


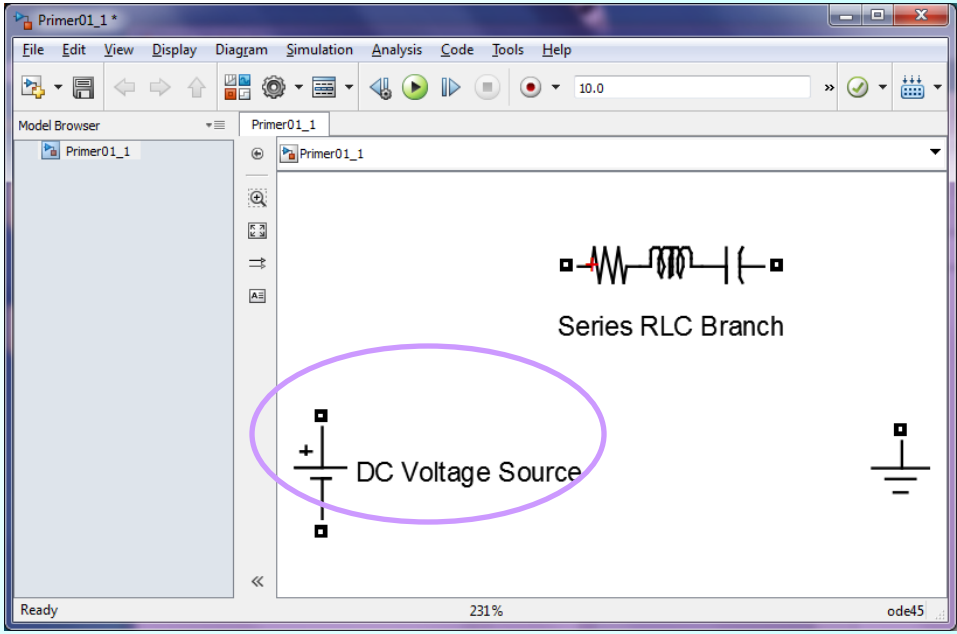


Simscape > Electrical

Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке

коло је образовано у тренутку $t_0 = 0$



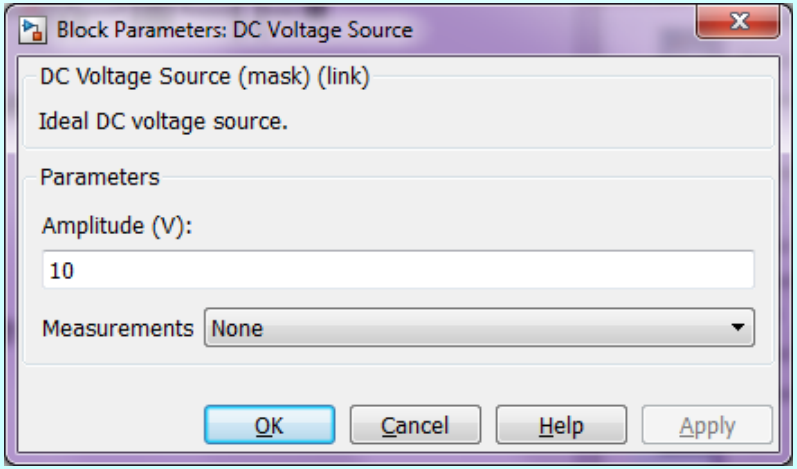


$R = 1\,\mathrm{k\Omega}$

$C = 1\,\mu\mathrm{F}$

$u_g(t) = U\,h(t), U = 10\,\mathrm{V}$

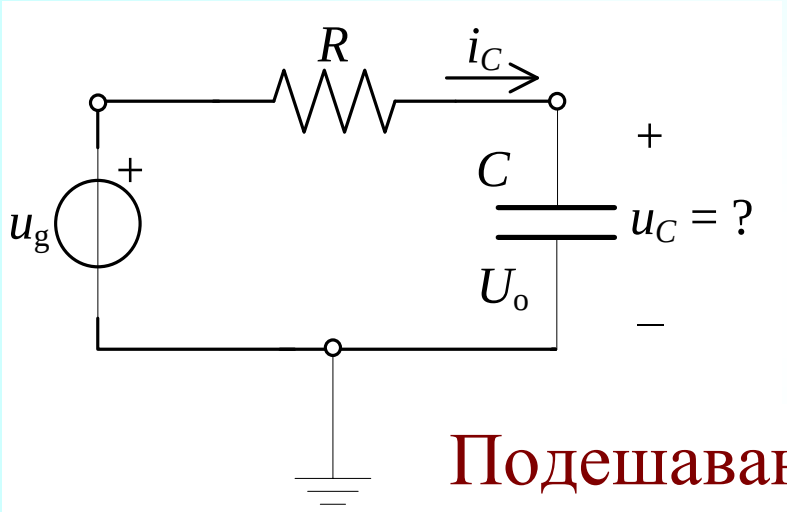
$u(t_0^-) = U_0 = -10\,\mathrm{V}$



Simscape > Electrical

Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке

коло је образовано у тренутку $t_0 = 0$



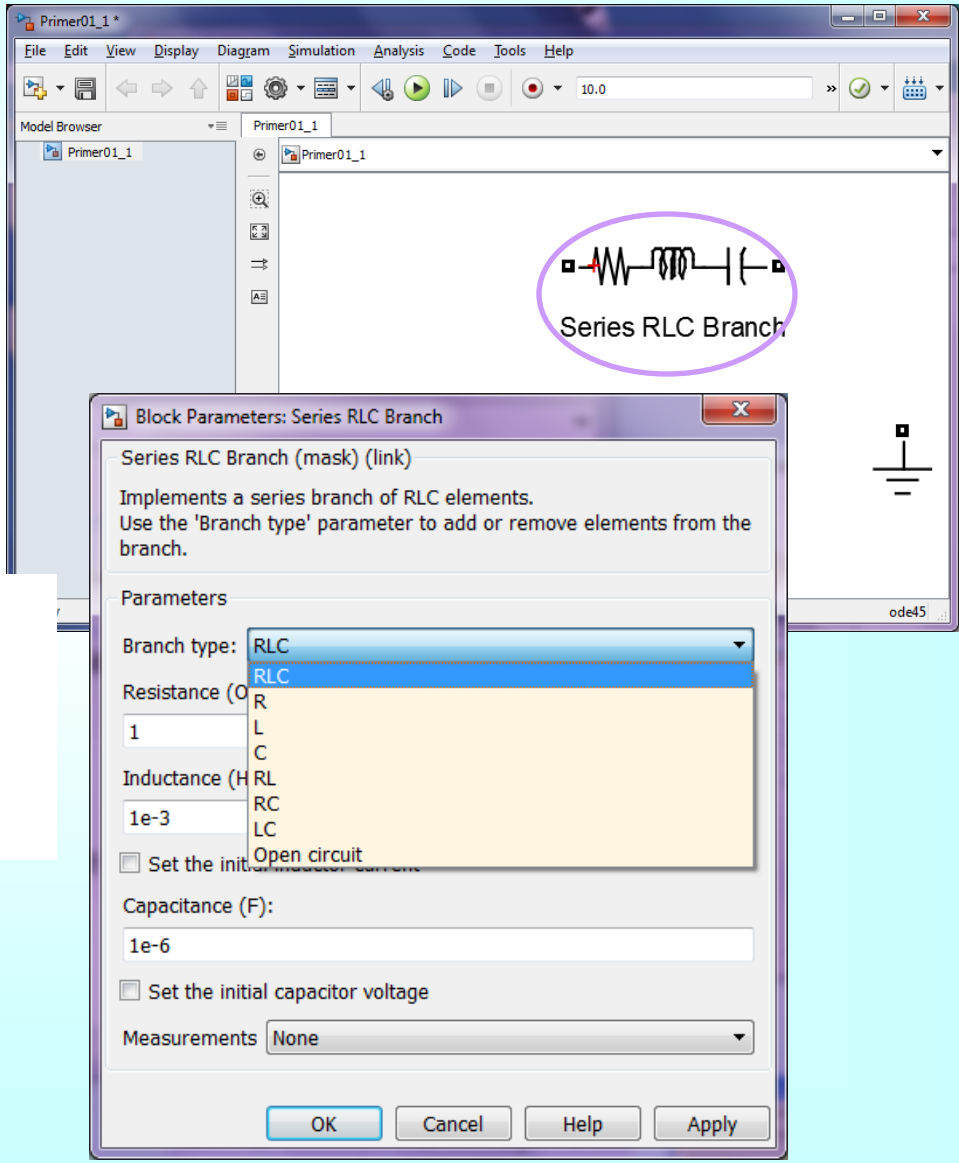
$R = 1\,\mathrm{k\Omega}$

$C = 1\,\mu\mathrm{F}$

$u_g(t) = U\,h(t), U = 10\,\mathrm{V}$

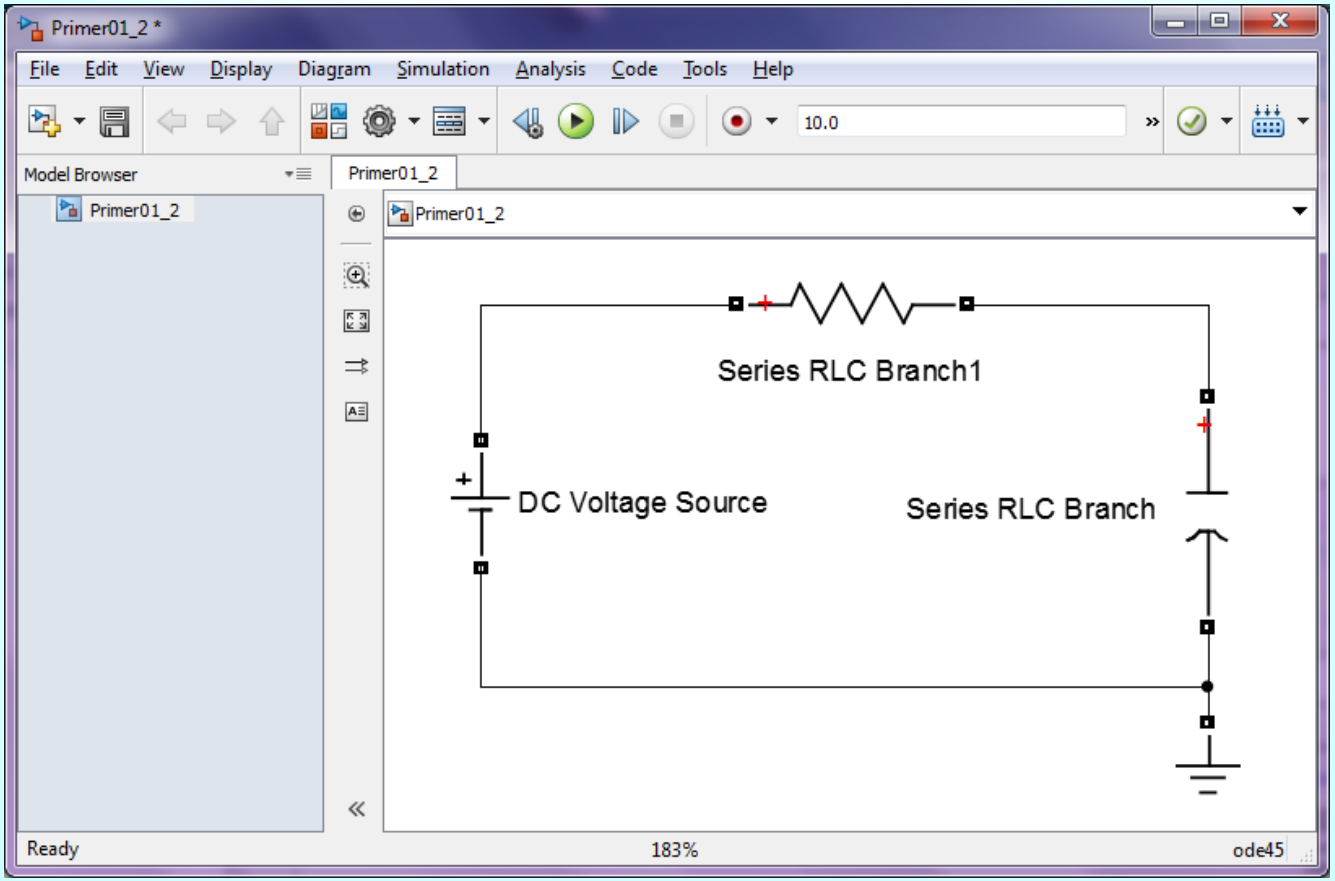
$u(t_0^-) = U_0 = -10\,\mathrm{V}$

Подешавање параметара симулације



Simscape > Electrical

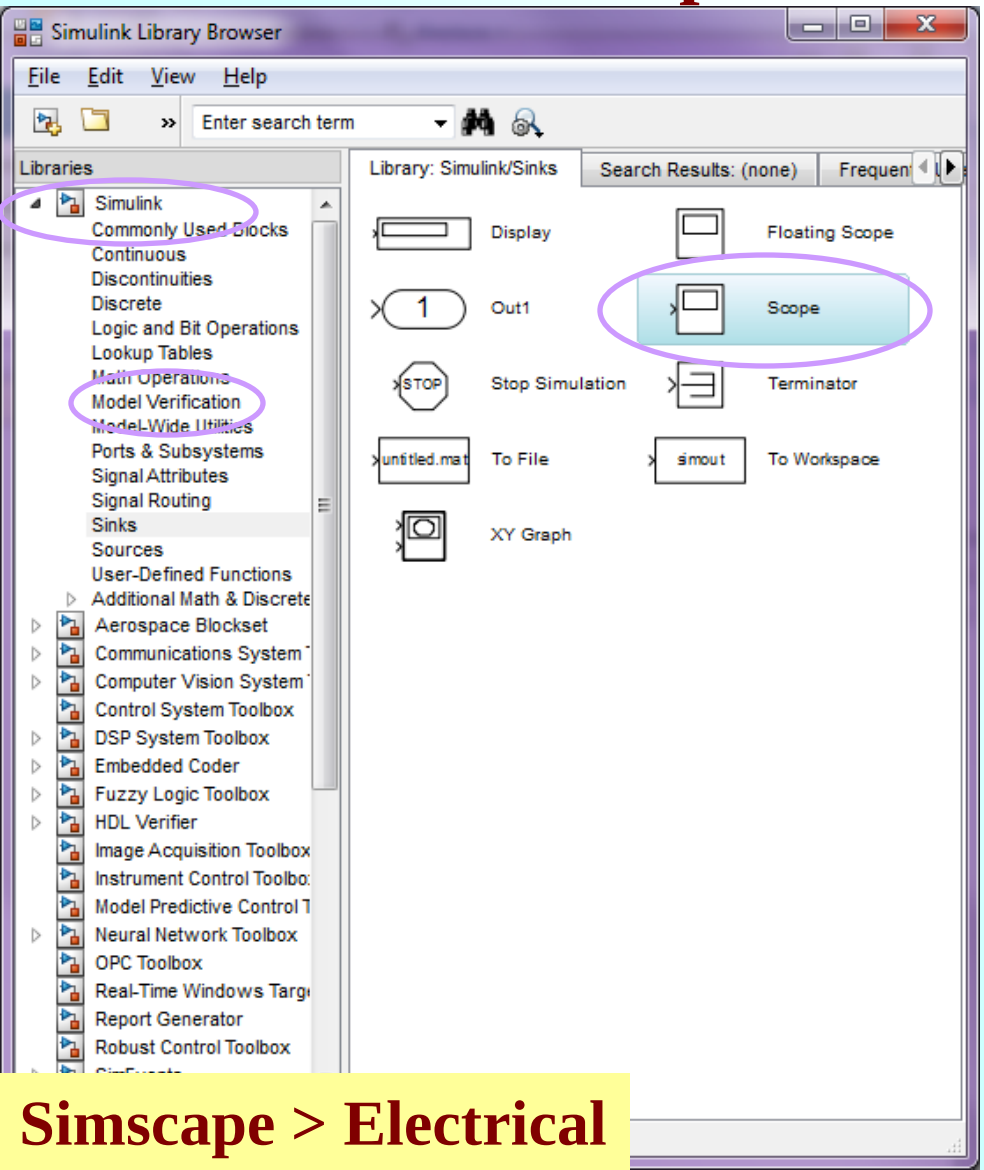
Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке

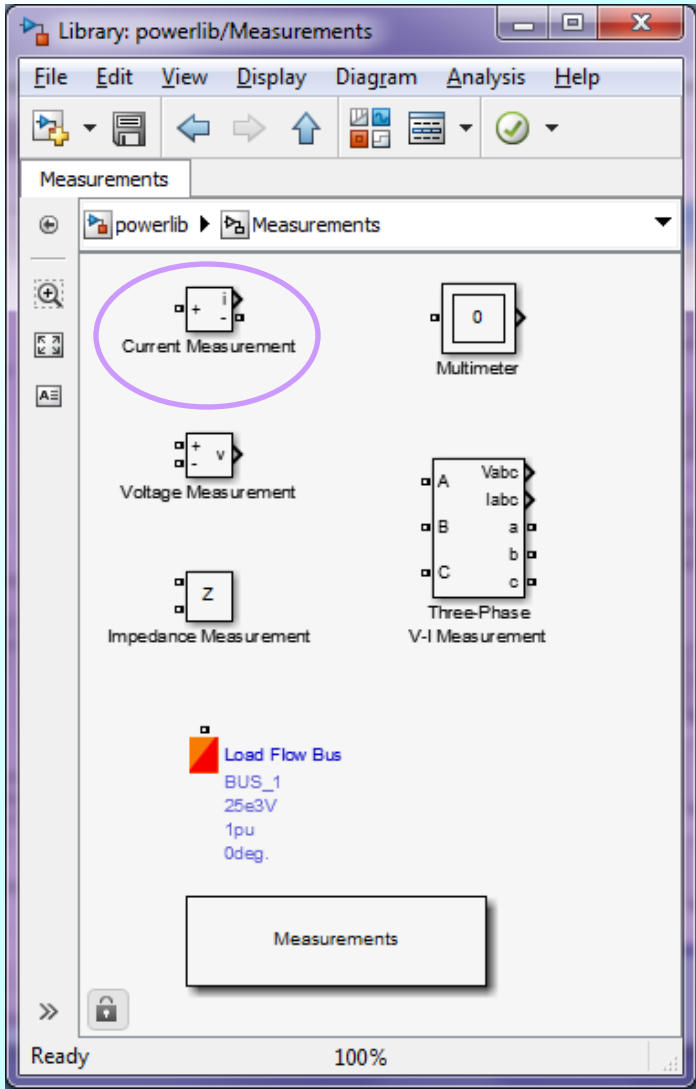


Како се мери напон???

Simscape > Electrical

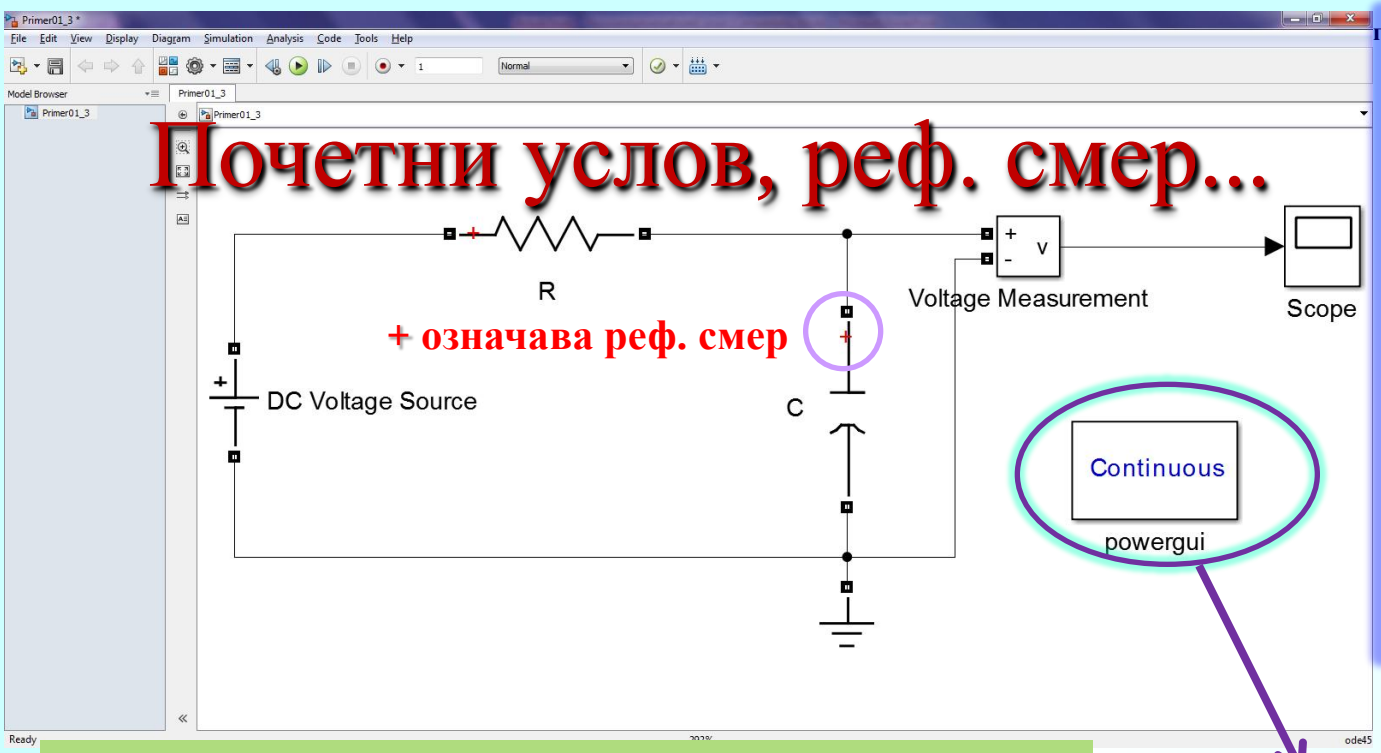
Симулација електричног кола коришћењем **Simscape>Electrical** библиотеке





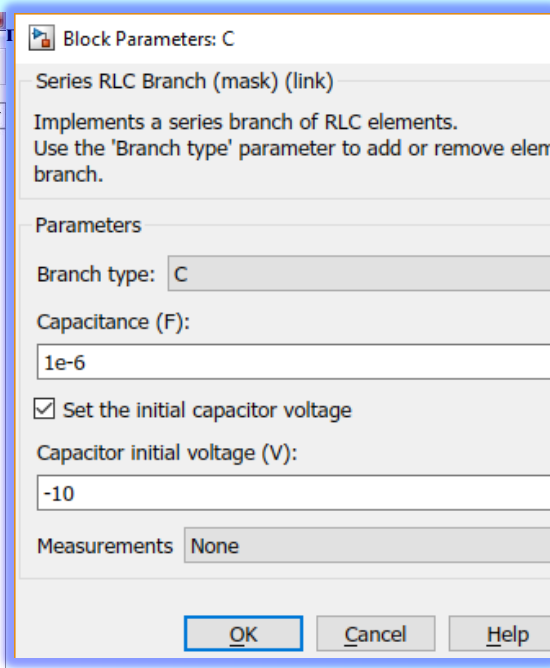
Simscape > Electrical

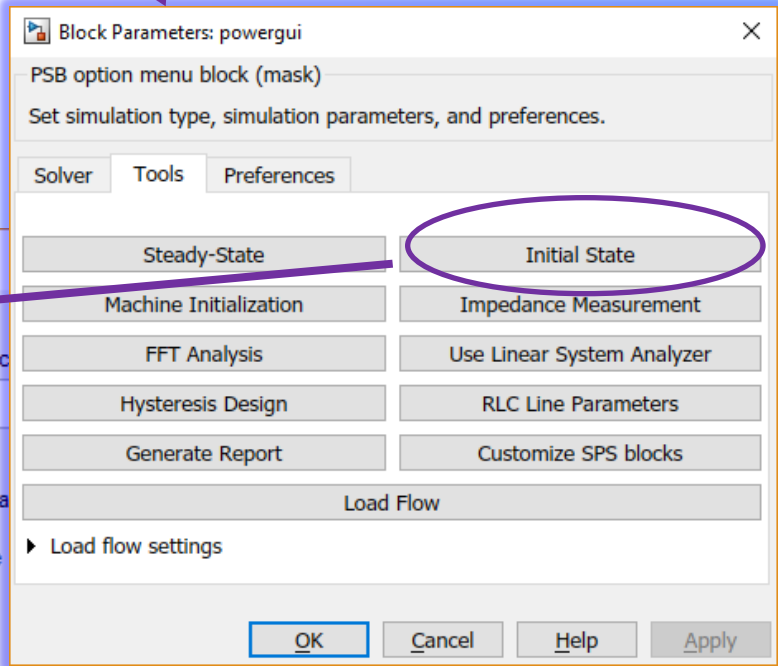
Почетни услов, реф. смер...

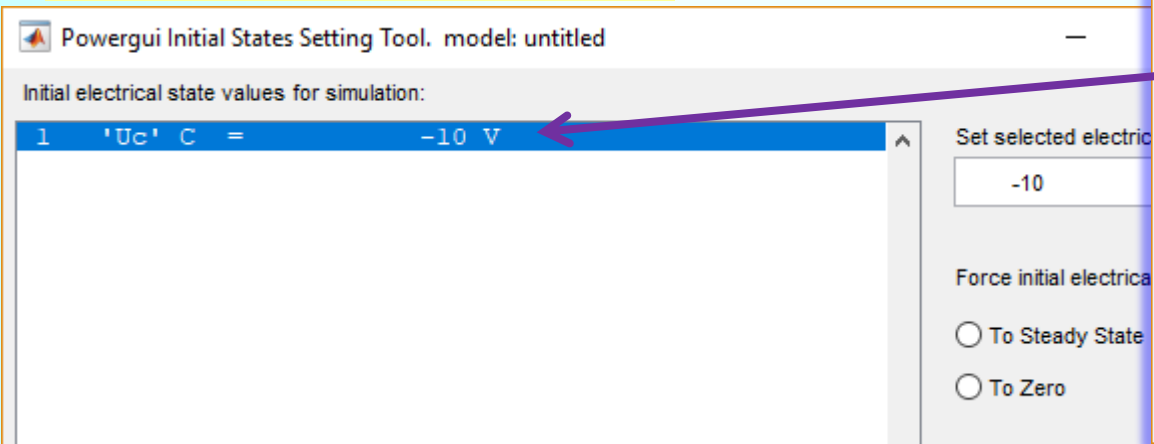


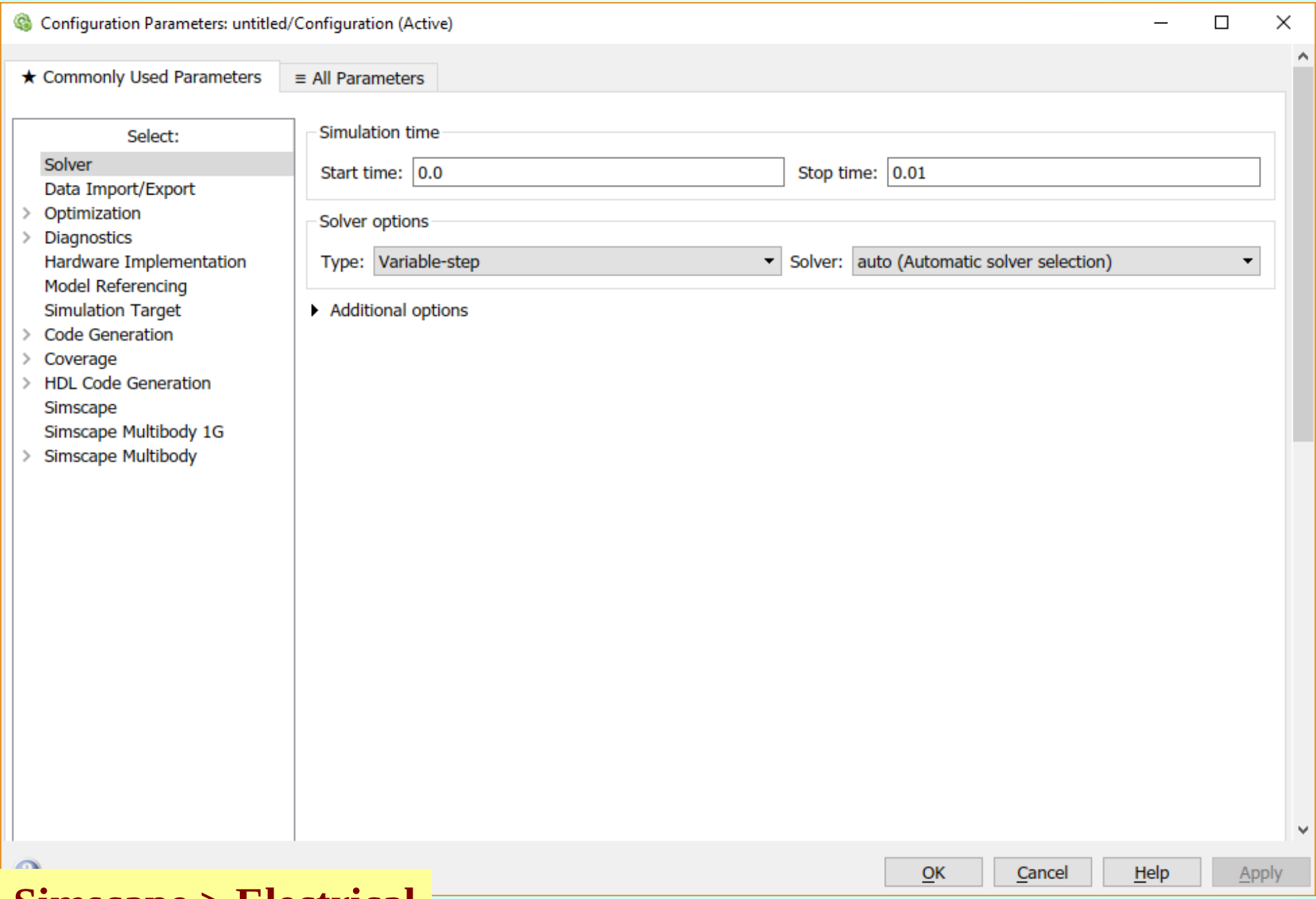
Плоче кондезатора се разликују!!!

Simscape > Electrical

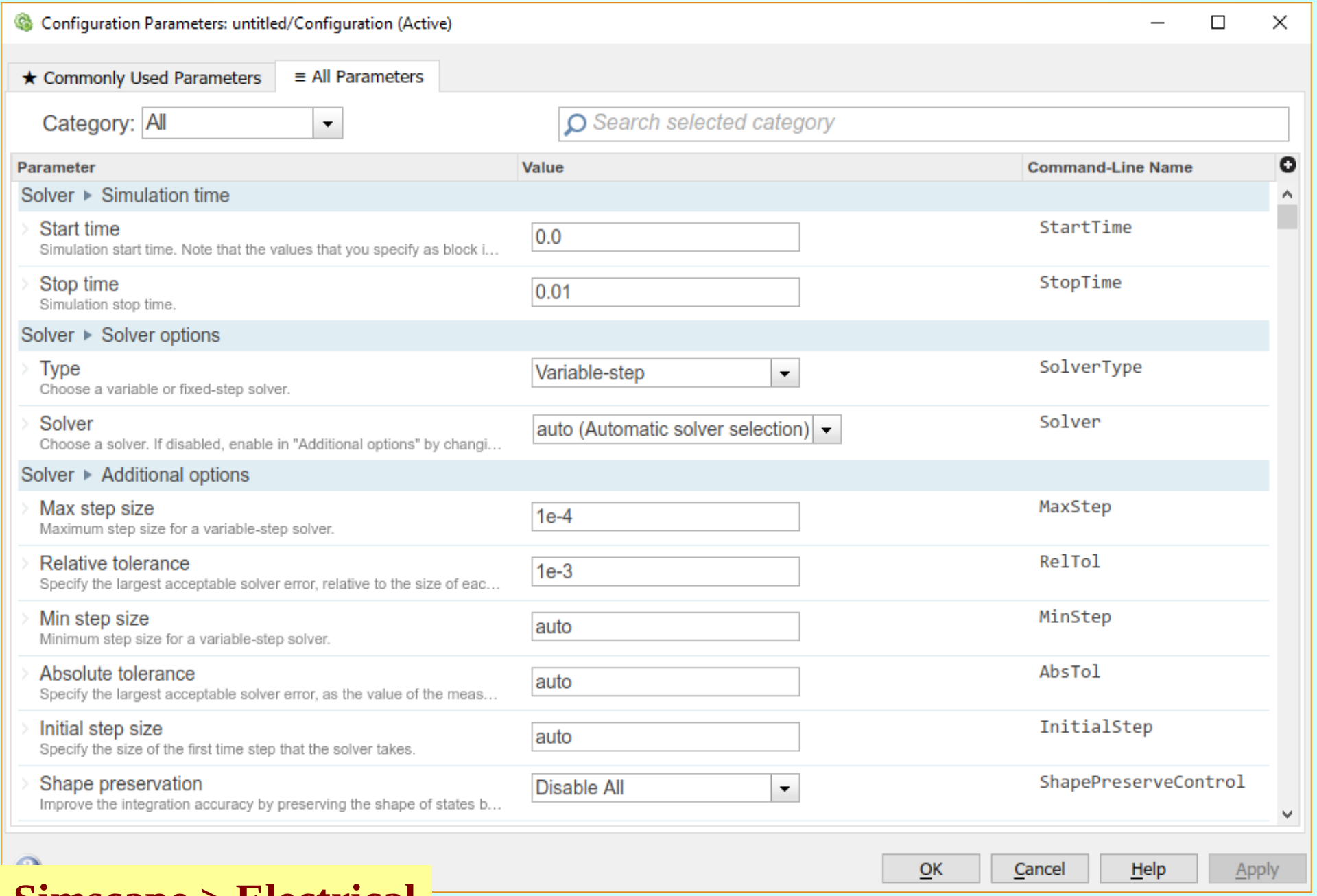






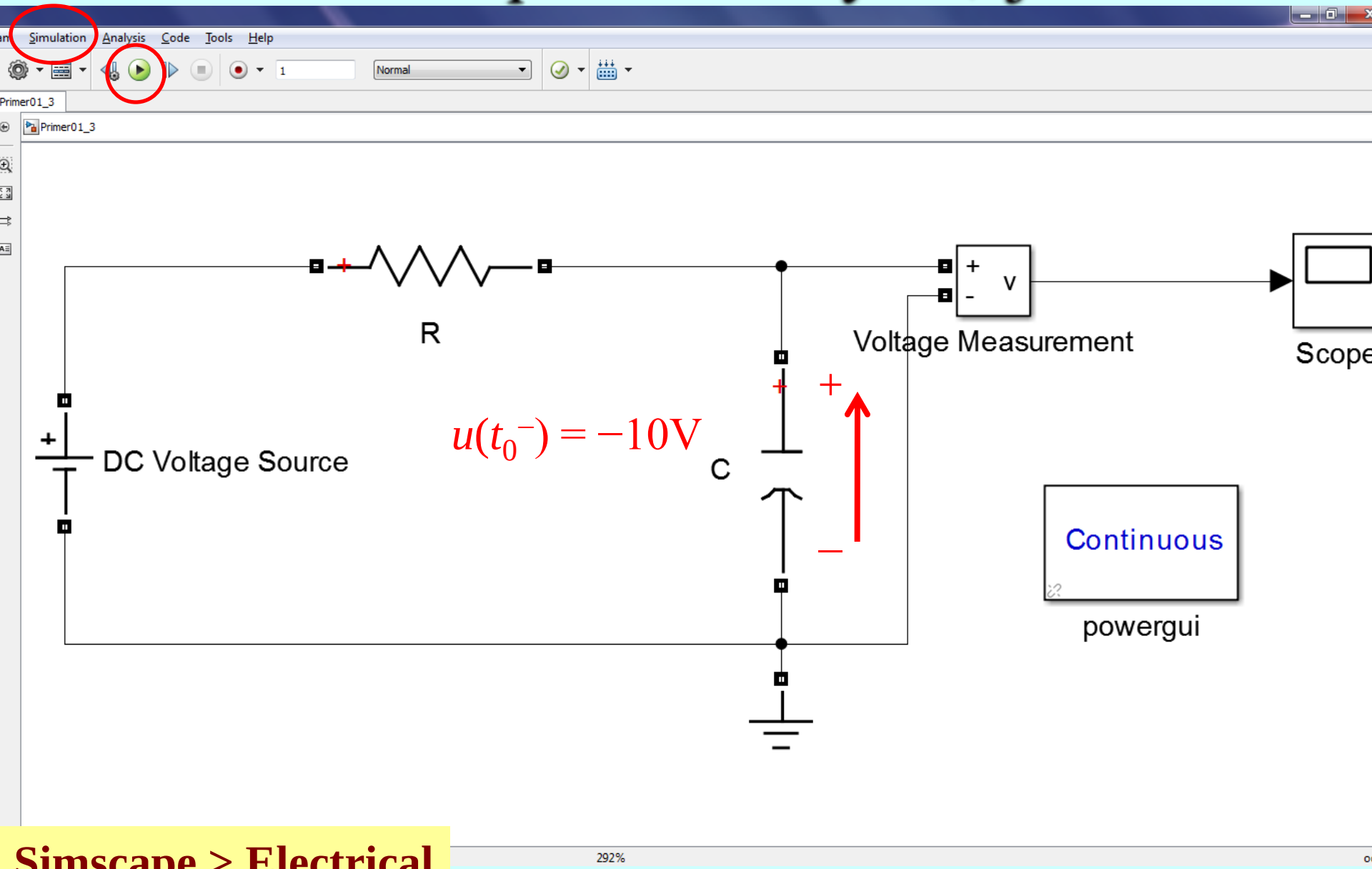


Simscape > Electrical



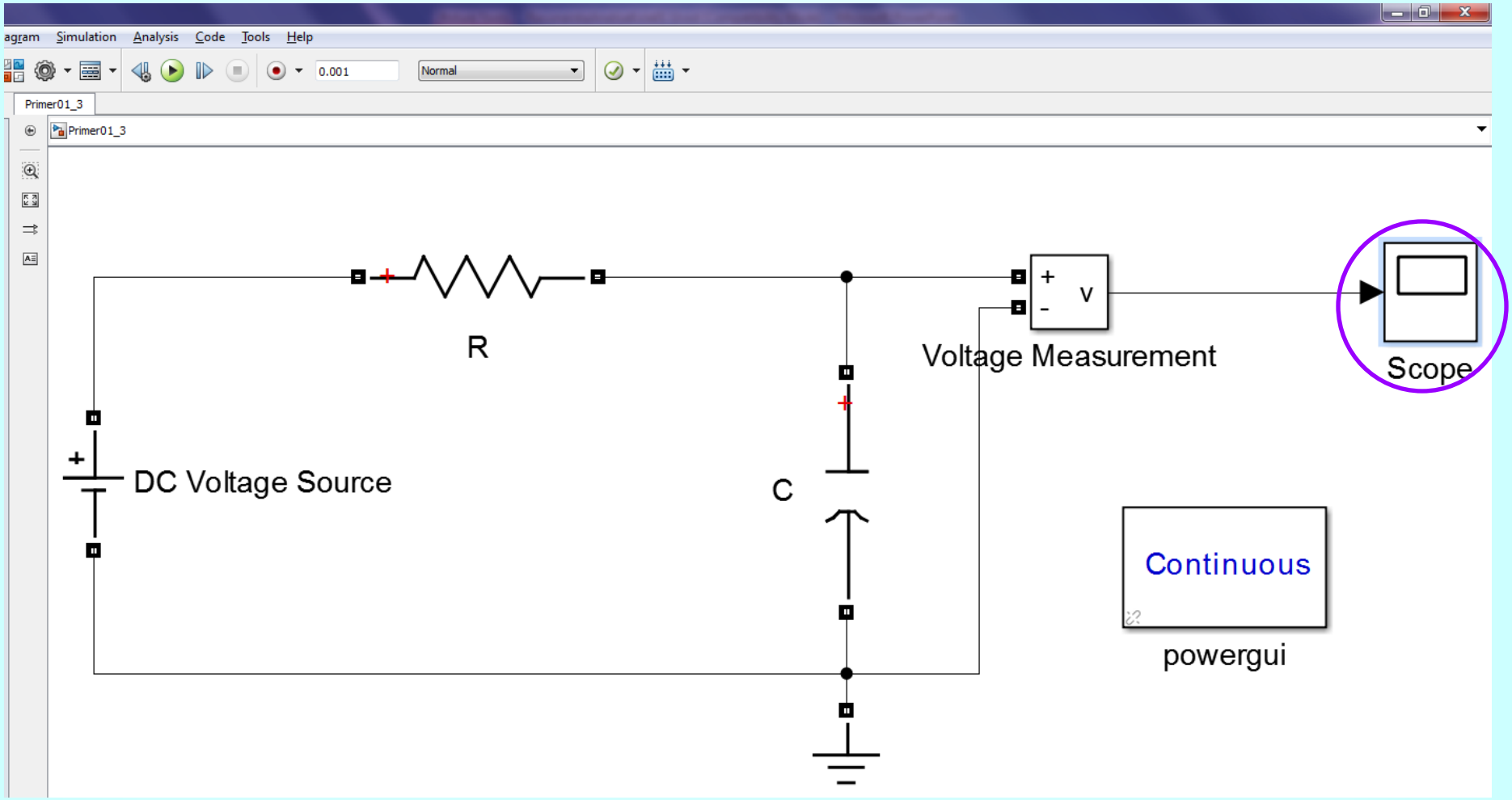
Simscape > Electrical

Покретање симулације...

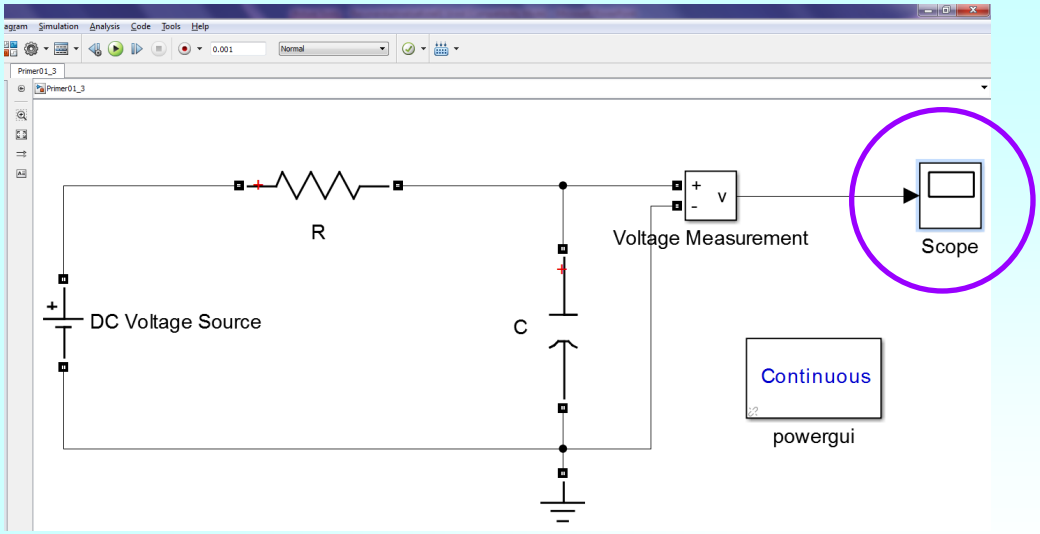


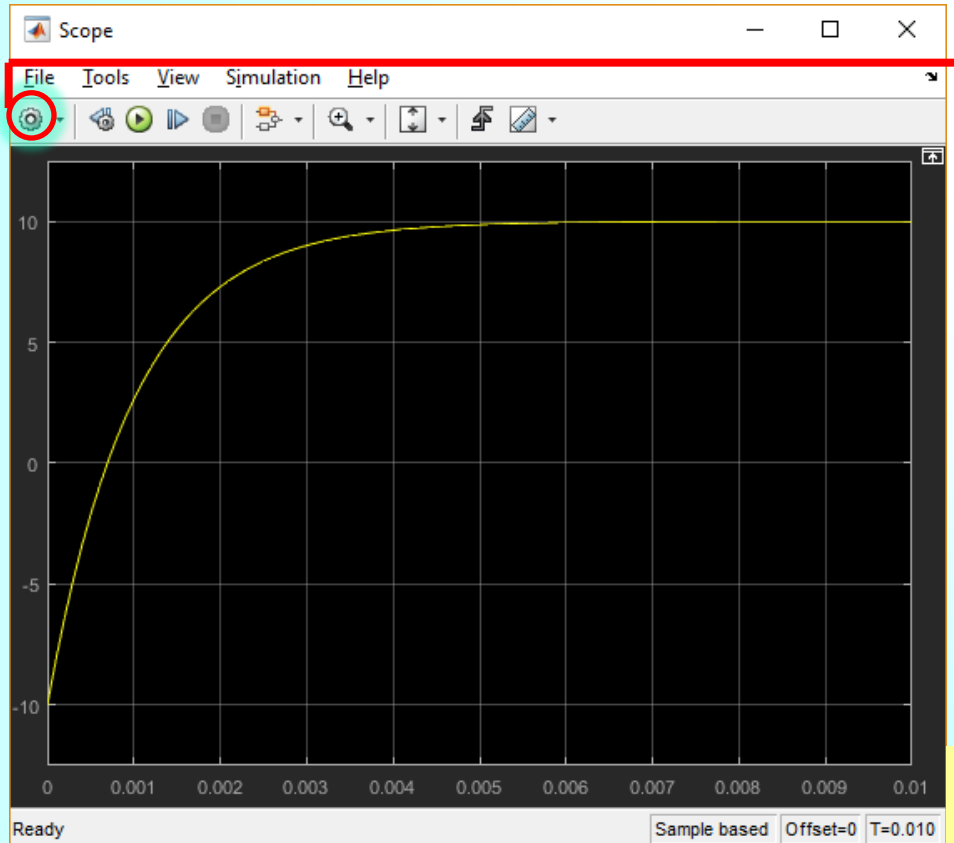
Simscape > Electrical

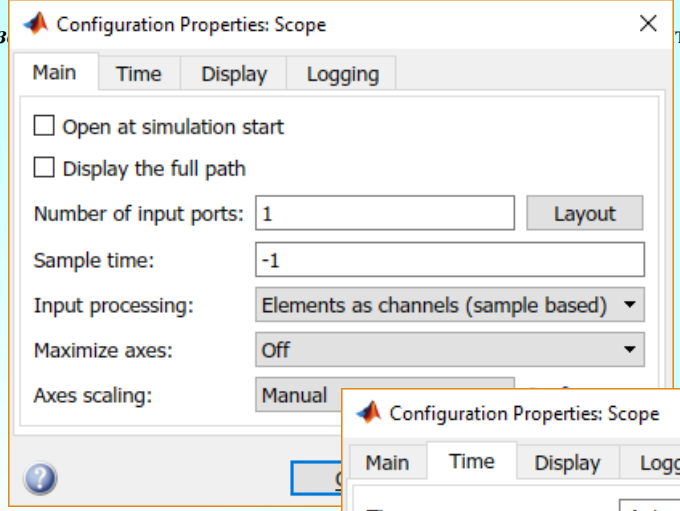
Налажење одзива... Осцилоскоп

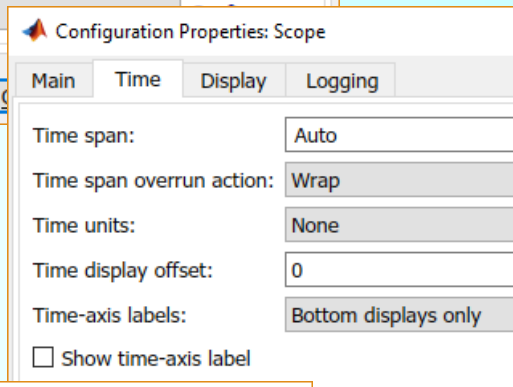


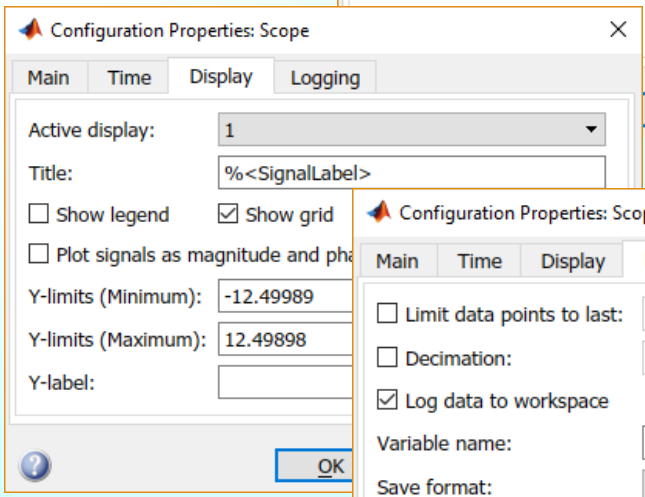
Simscape > Electrical

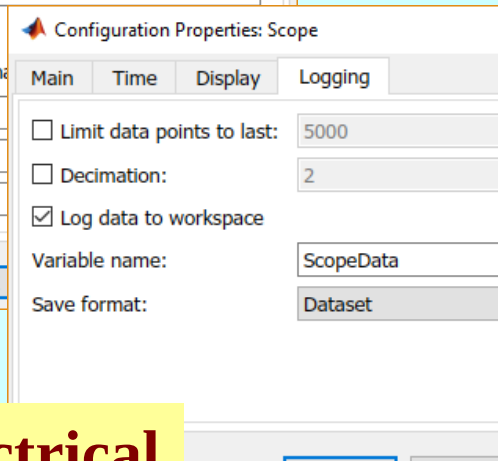




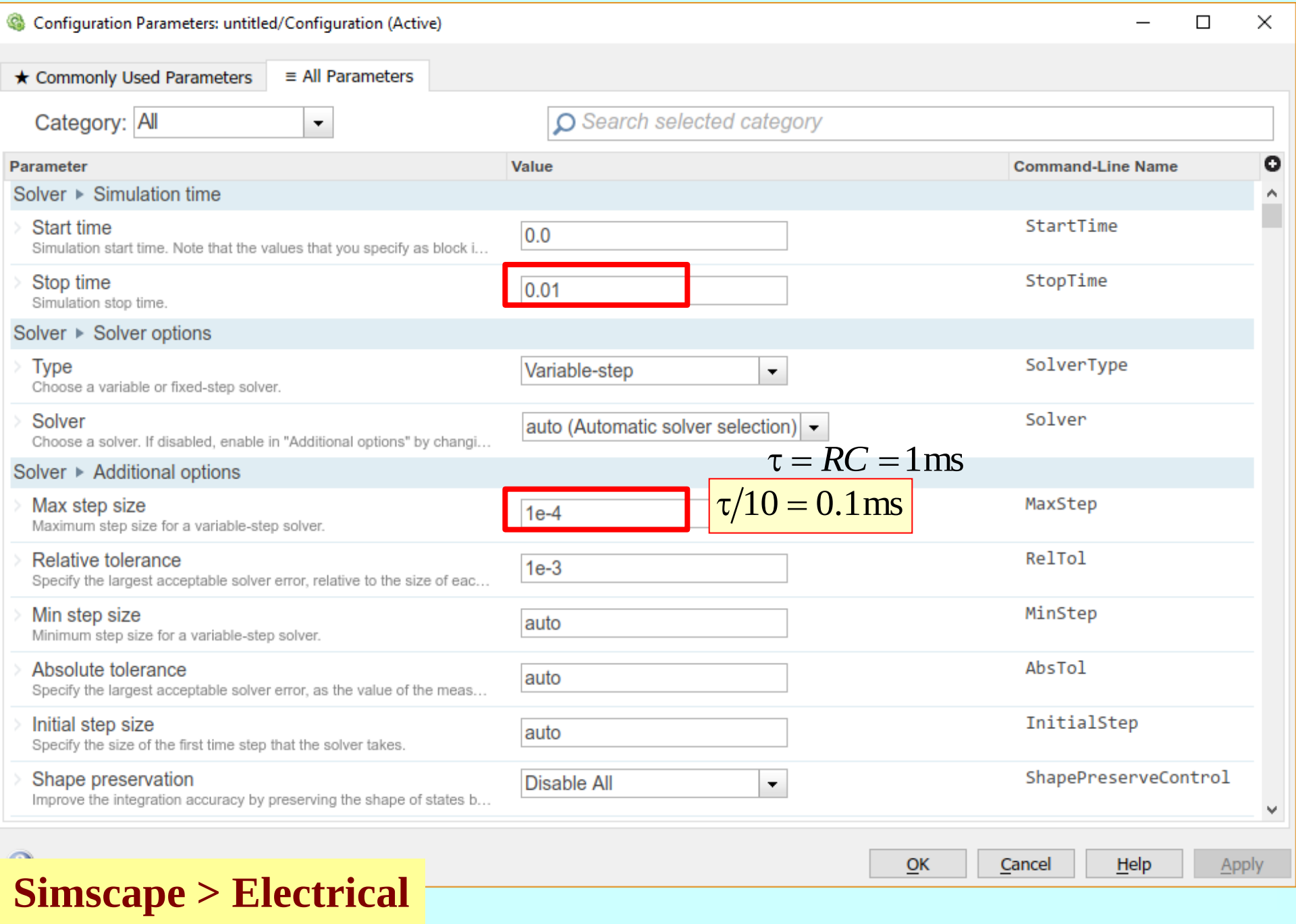








Simscape > Electrical



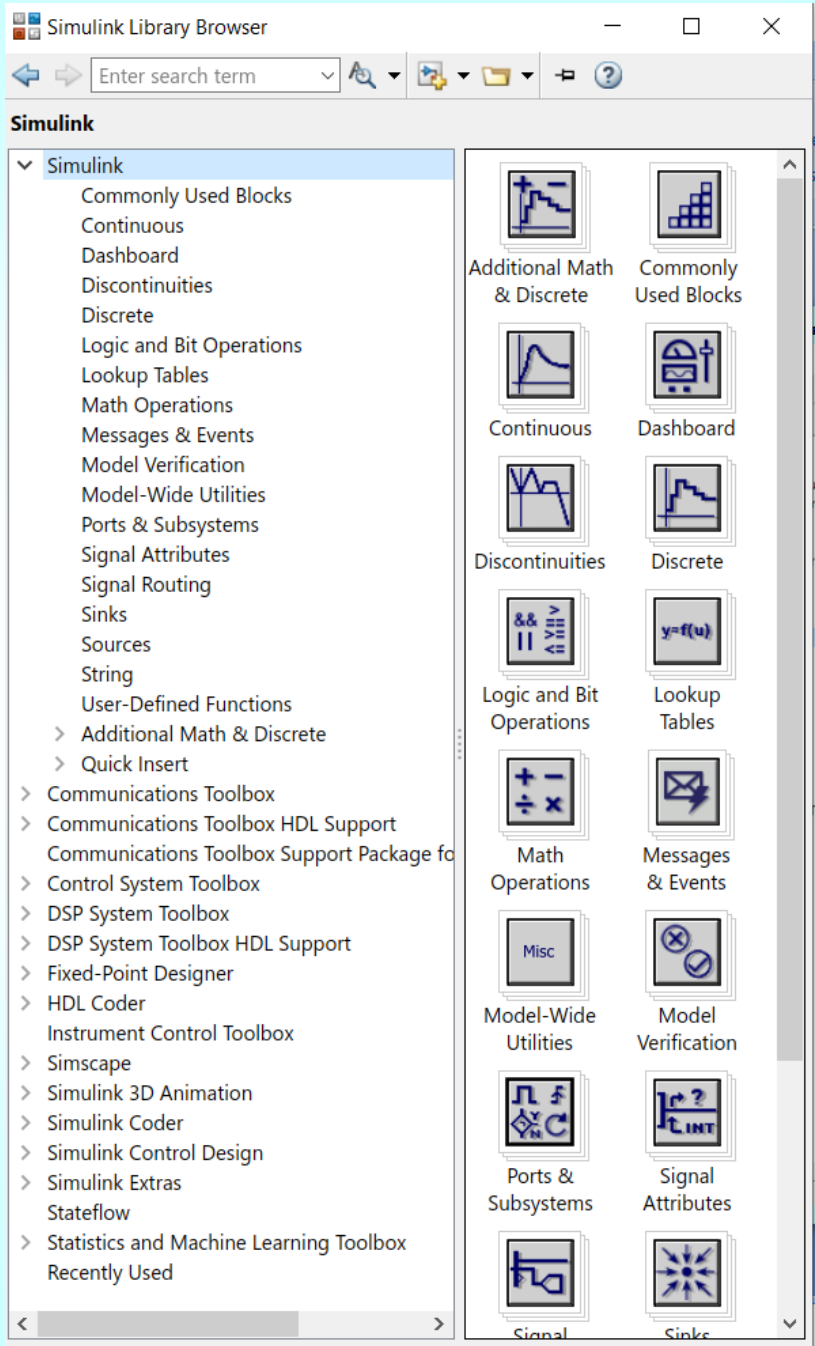
$\tau = RC = 1\text{ms}$

$\tau/10 = 0.1\text{ms}$

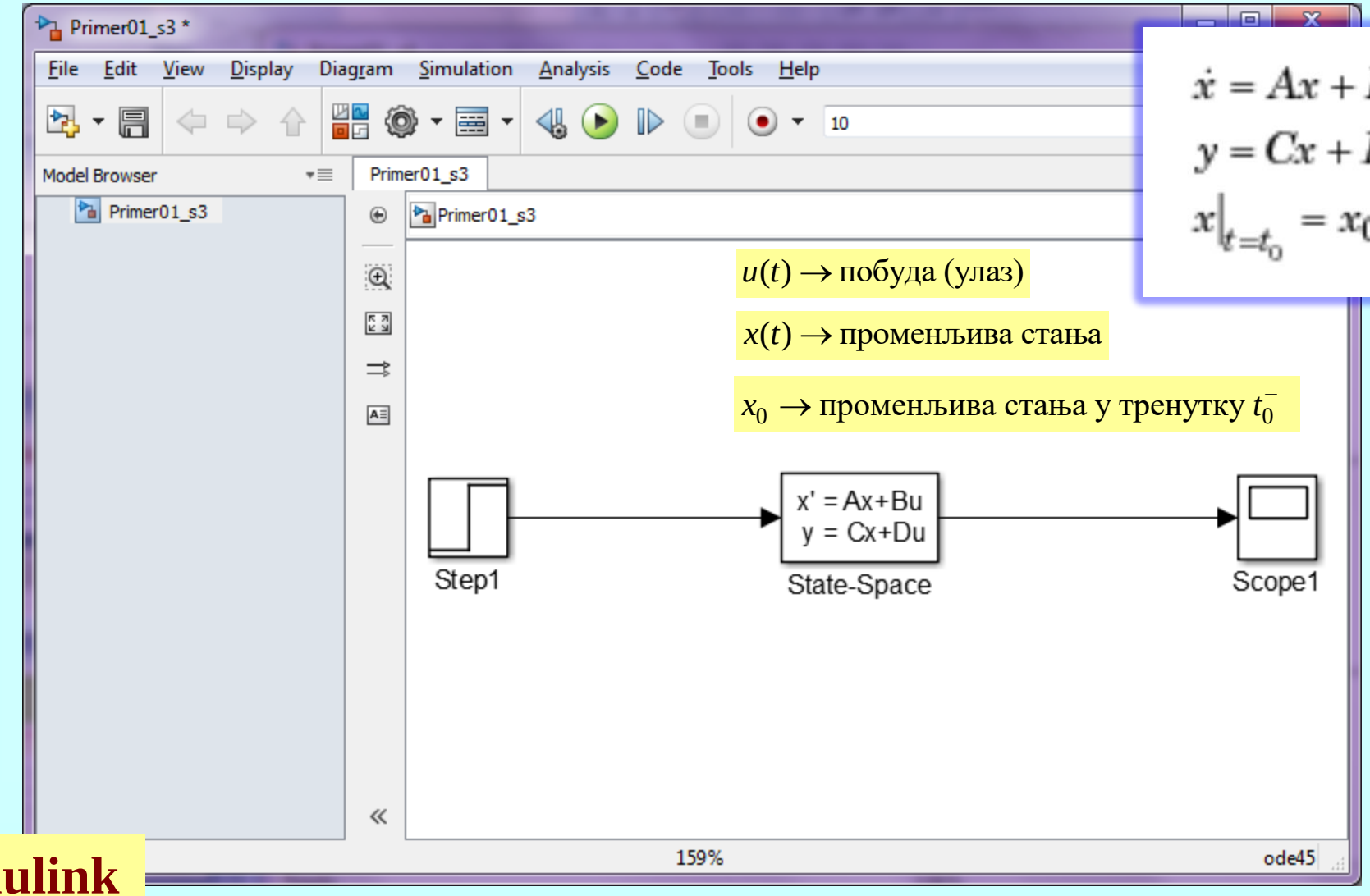
Simscape > Electrical

Simulink

- Решавање кола може се извршити његовим свођењем на систем линеарних диференцијалних једначина
- Дати систем може се представити једноставним блоком са више улаза и излаза

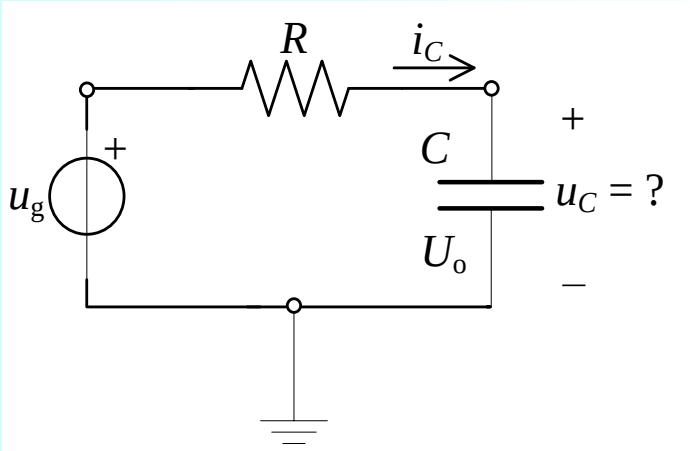


Симулација електричног кола као уопштеног система коришћењем **Simulink**-а



Simulink

Симулација електричног кола као уопштеног система коришћењем **Simulink**-а



$u = u_g(t) \rightarrow$ побуда (улаз)

$x = u_C(t) \rightarrow$ напон кондензатора (промењљива стања и одзив - излаз)

$x(t_0^-) = U_0 \rightarrow$ напон кондензатора у тренутку t_0^-

$$\dot{x} = Ax + Bu$$
$$y = Cx + Du$$
$$x\big|_{t=t_0} = x_0,$$

коло је образовано у тренутку $t_0 = 0$

$$u_g(t) = Ri_C + u_C(t)$$
$$i_C = C\frac{du_C}{dt}$$
$$u_C(t_0^-) = U_0 = -10\,\mathrm{V}$$

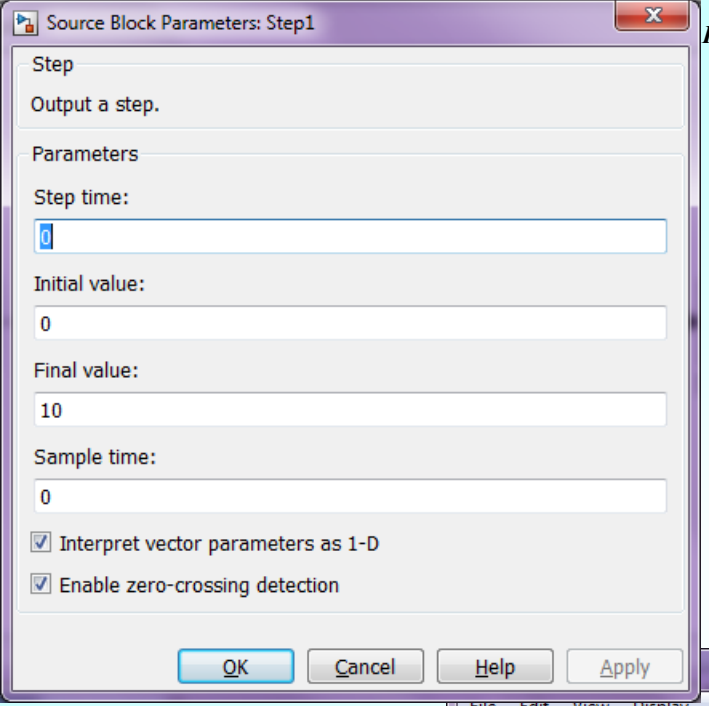
$$R = 1\,\mathrm{k\Omega}$$
$$C = 1\,\mu\mathrm{F}$$
$$u_g(t) = U\,h(t), U = 10\,\mathrm{V}$$
$$u_C(t_0^-) = U_0 = -10\,\mathrm{V}$$

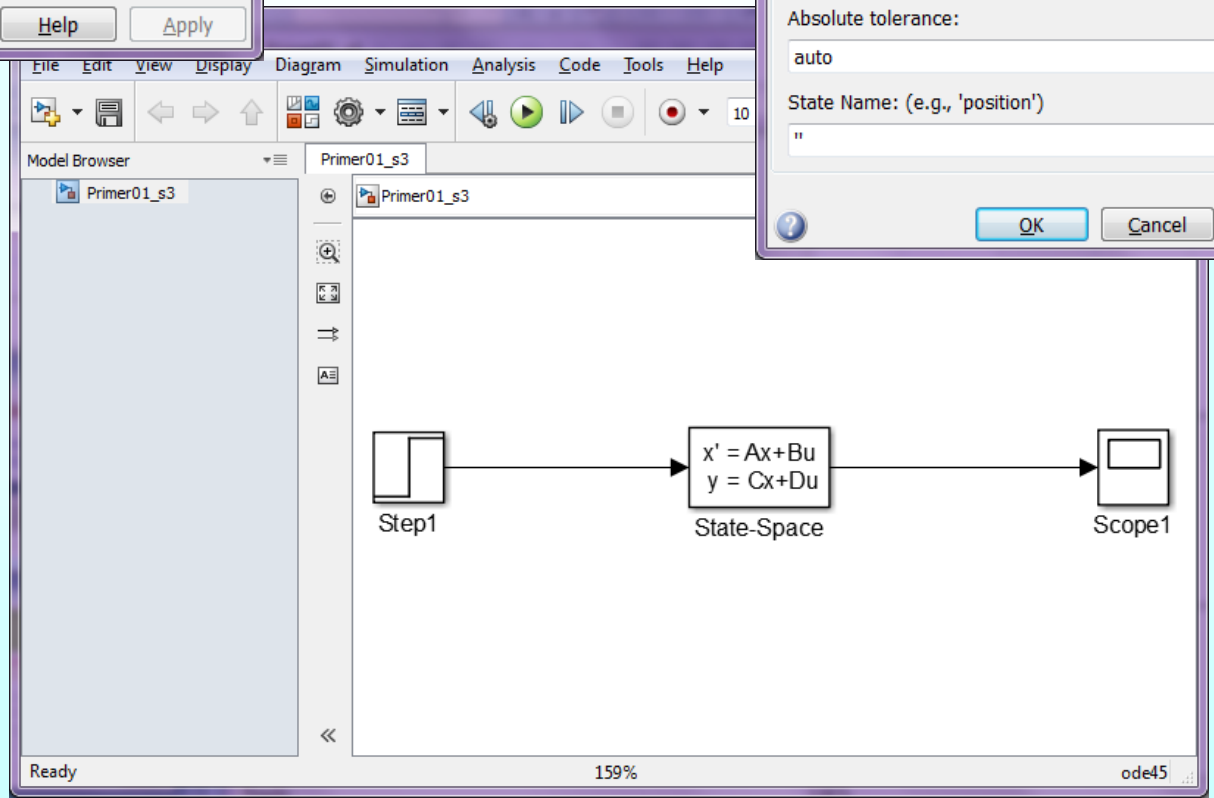
$$\frac{du_C(t)}{dt} = \frac{i_C}{C} = \frac{u_g(t) - u_C(t)}{RC} = -\frac{u_C(t)}{RC} + \frac{u_g(t)}{RC}$$
$$u_C(t_0^-) = U_0 = -10\,\mathrm{V}$$

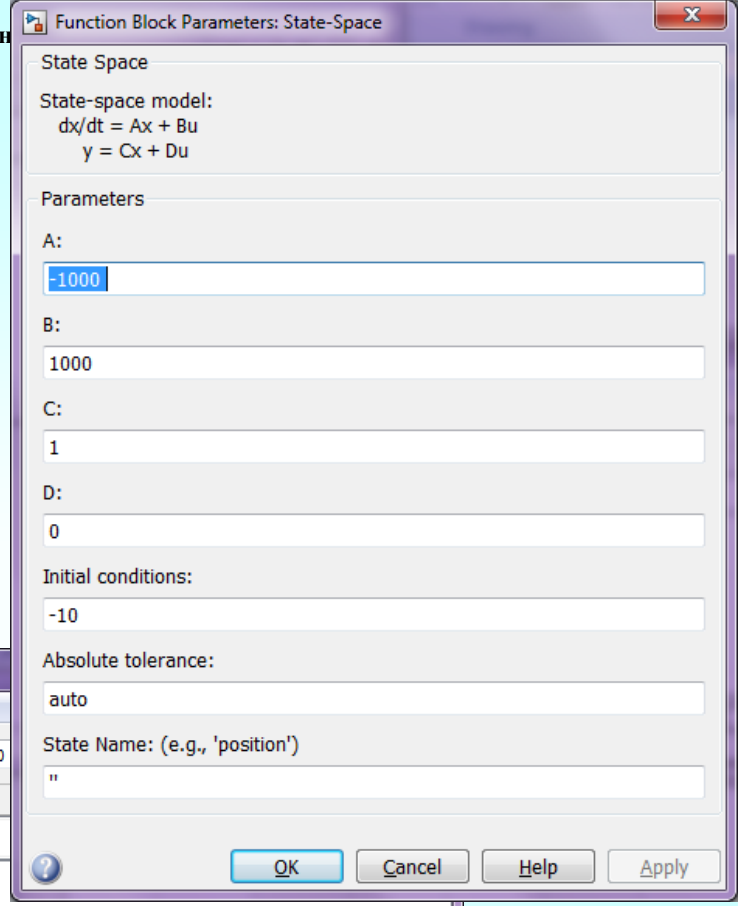
$$\frac{du_C(t)}{dt} = Au_C(t) + Bu_g(t) \Rightarrow A = -\frac{1}{RC} = -1000, B = 1000$$
$$u_C(t) = Cu_C(t) \Rightarrow C = 1, D = 0$$
$$u_C(t_0^-) = U_0 = -10\,\mathrm{V}$$

Simulink

Симулација електричног кола као уопштеног система коришћењем **Simulink**-а

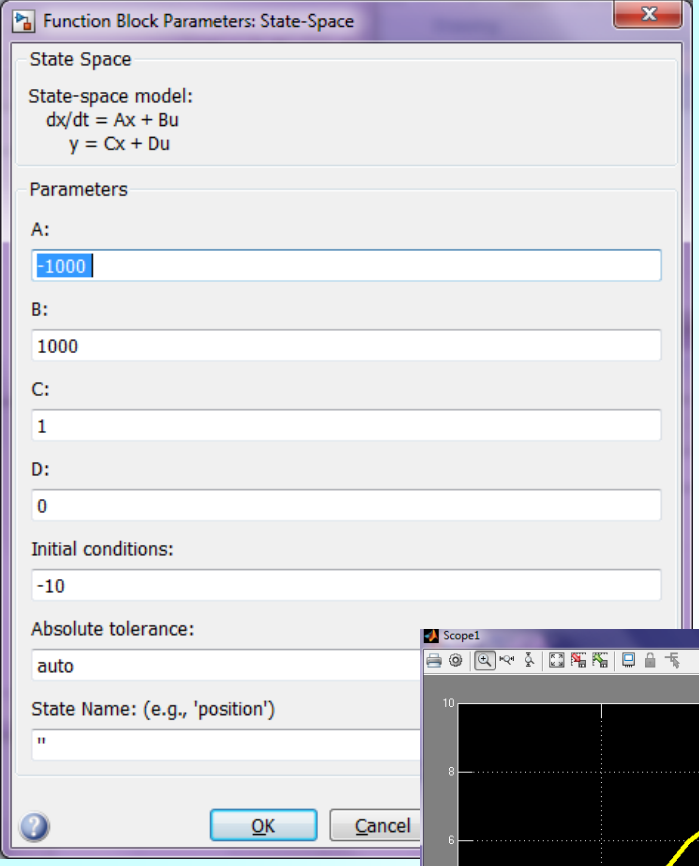


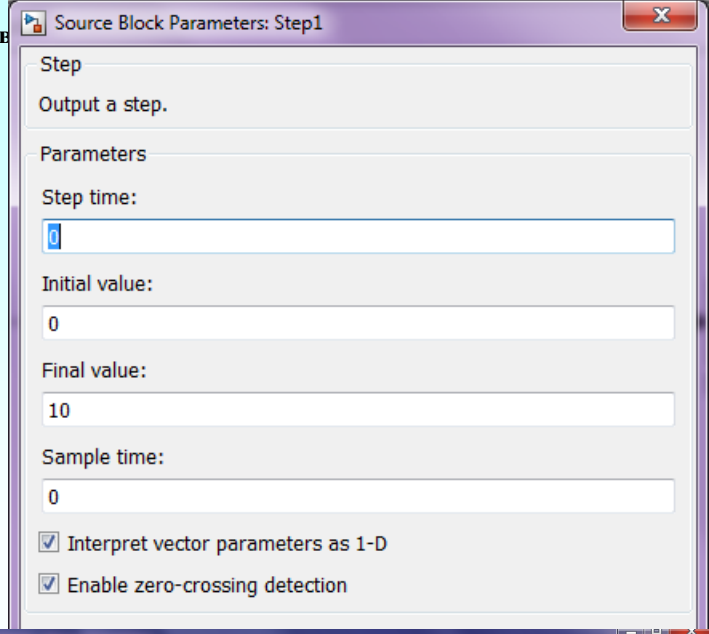


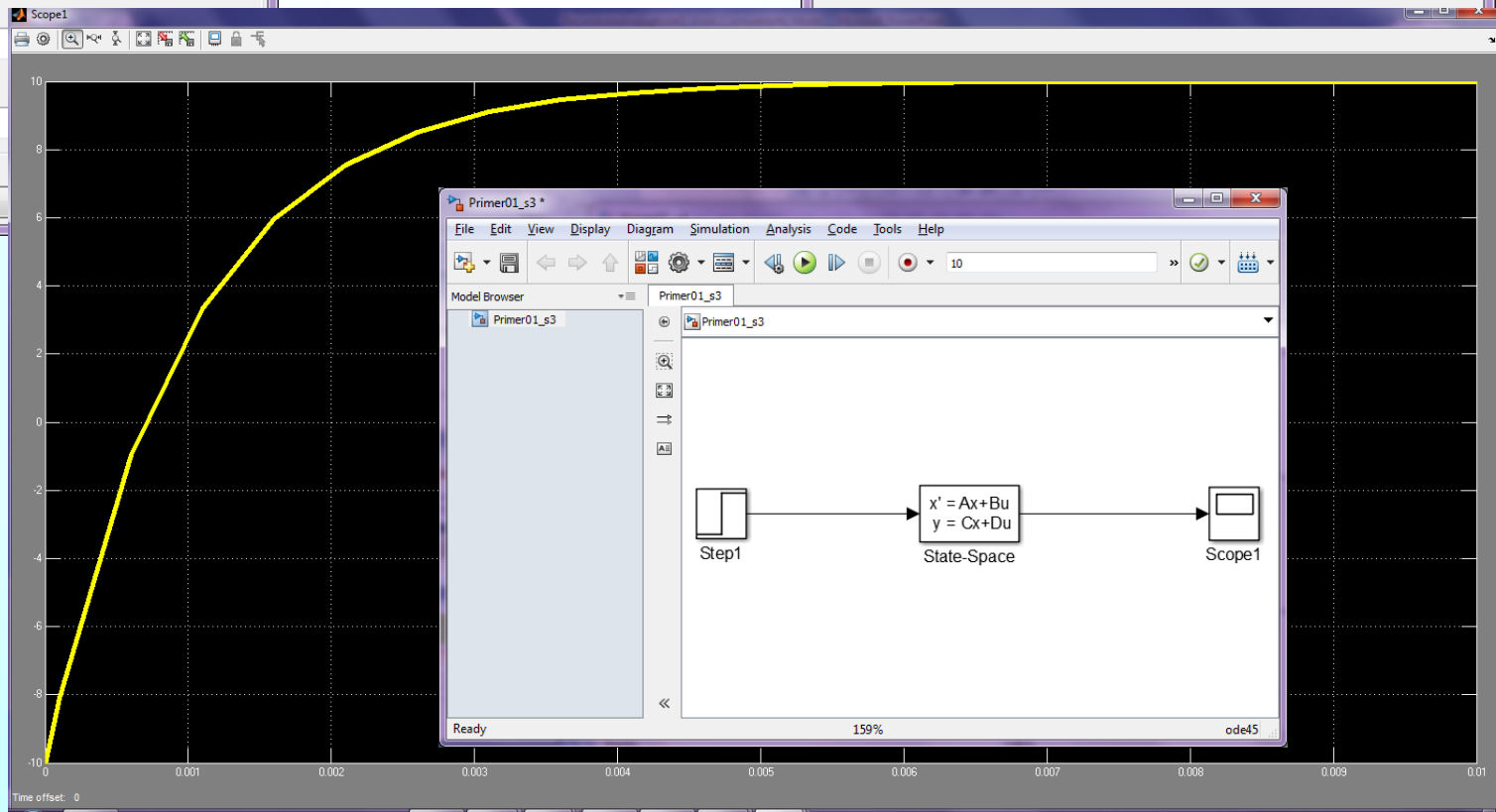


Simulink

Симулација електричног кола као уопштеног система коришћењем **Simulink**-а

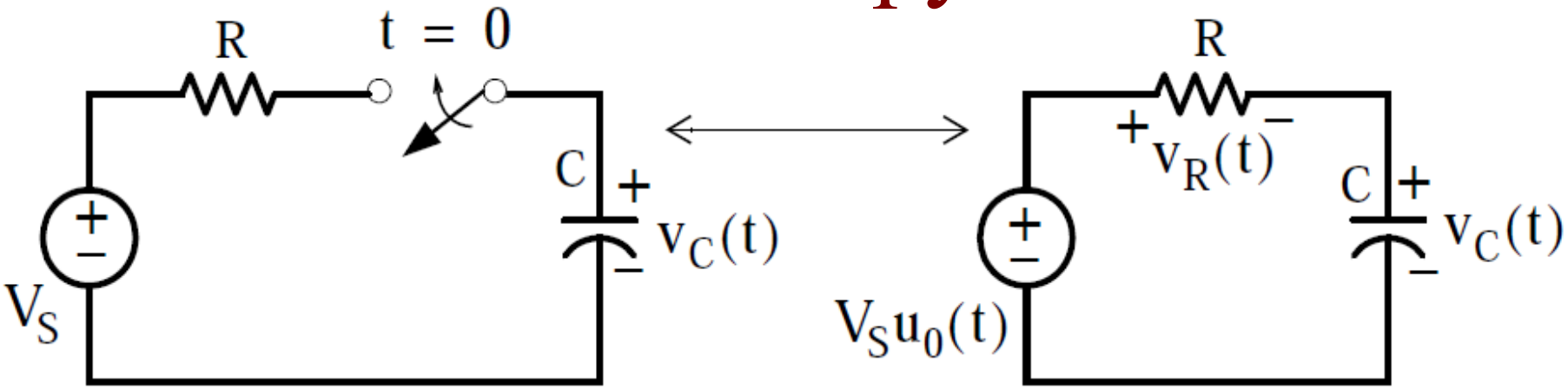






Simulink

Решимо “ручно”



$$v_R + v_C = V_S u_0(t)$$

$$i = i_C = C\frac{dv_C}{dt}$$

$$v_R = Ri = RC\frac{dv_C}{dt}$$

$$RC\frac{dv_C}{dt} + v_C = V_S u_0(t)$$

$t > 0$

$$RC\frac{dv_C}{dt} + v_C = V_S$$

$$v_C(t) = (V_S - V_S\, e^{-(1/RC)t})u_0(t)$$

MATLAB
Symbolic Math Toolbox (Live Script)

- Библиотека за симболички рачун (алгебарске и диференцијалне једначине)
- Омогућава нумеричку анализу резултата прорачуна

Live Editor - D:\Engineering_Work\ETF\Teaching\PRK\RCKolo_v0.mlx

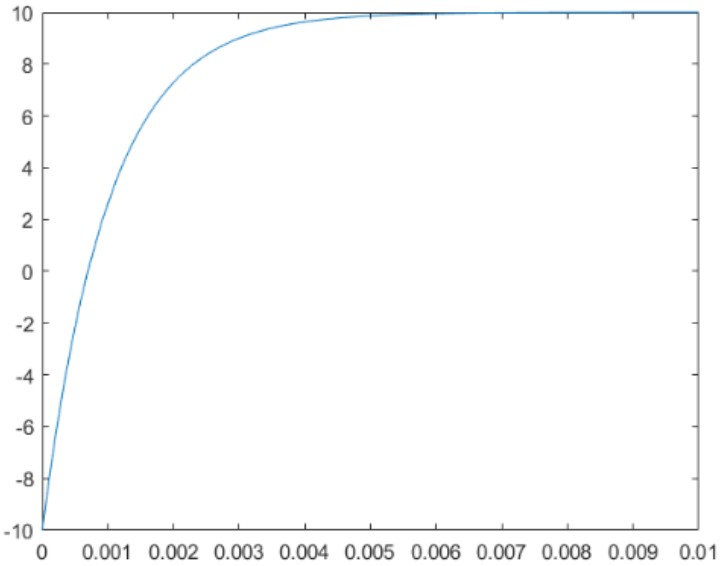
pluto1.m | template1.m | RCKolo_v0.mlx

```
1  syms v_c(t)
2  ode = diff(v_c,t) ==-1000*v_c+10000
3  cond = v_c(0)==-10
4  v_csol = dsolve(ode,cond)
5
6  fplot(v_csol, [0 0.01])
```

$$\frac{\partial}{\partial t} v_c(t) = 10000 - 1000\, v_c(t)$$

cond = $v_c(0) = -10$

v_csol = $10 - 20\, e^{-1000\, t}$



Simbolic Math Toolbox