

# MATLAB/Simulink tutorial



ECEN 2060

Spring 2008



# MATLAB/Simulink

---

- MathWorks tools for technical computing and simulations, widely used across various engineering and science disciplines
- MATLAB
  - Programming language and interactive environment well suited for computing, algorithms, data processing and visualization
- Simulink
  - Environment for graphical, model-based simulation of dynamic systems
- Version 2007a (or 2007b) available in all computer labs in the Engineering building
- Personal copy (full version, but for students only) can be purchased at [www.mathworks.com](http://www.mathworks.com) for \$99. This is not required for ECEN2060
- Tutorial objectives: very basic introduction to the tools at the level sufficient to understand ECEN2060 simulation models and do homework assignments

# Start MATLAB, then start Simulink

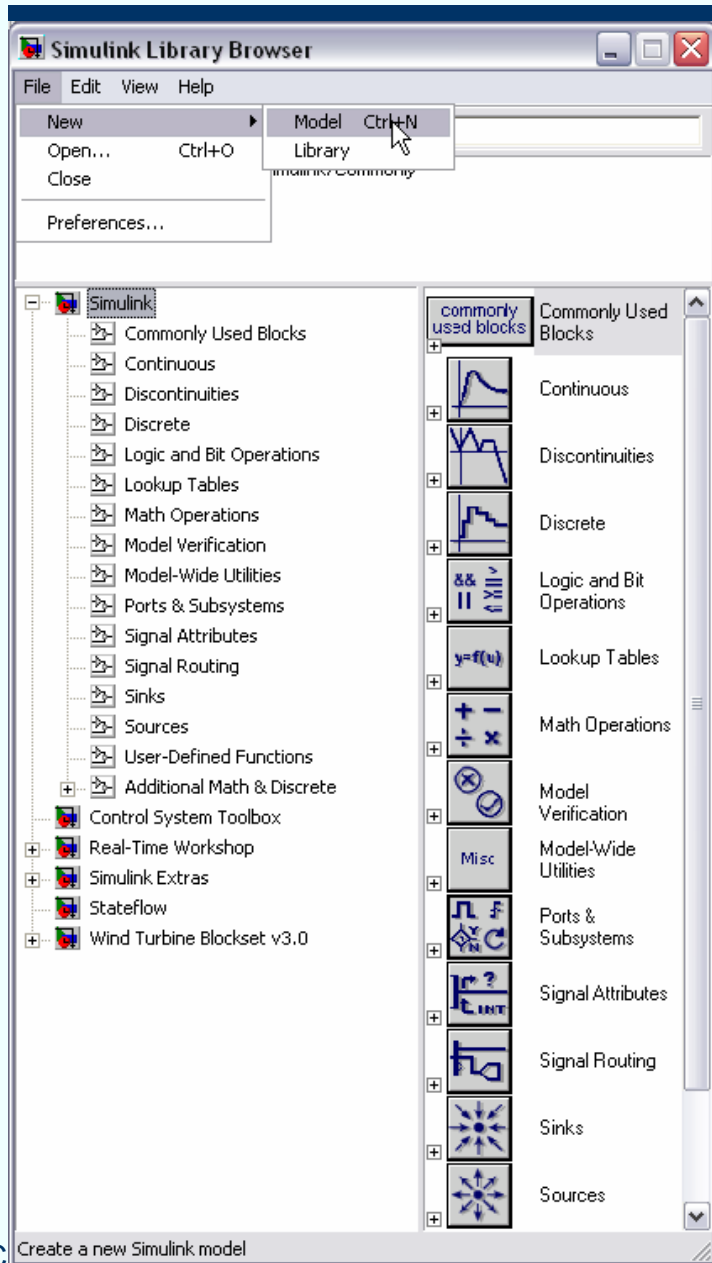
Click here to start Simulink

Choose Current Directory

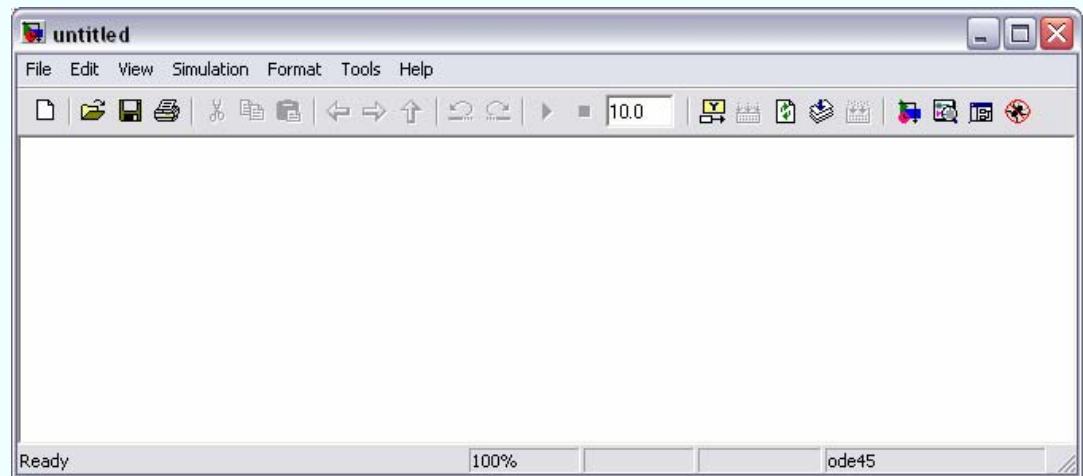
The screenshot displays the MATLAB 7.3.0 (R2006b) software interface. The main window has a menu bar (File, Edit, Debug, Desktop, Window, Help) and a toolbar. A blue circle highlights the Simulink icon in the toolbar, with an arrow pointing to the text "Click here to start Simulink". Another blue circle highlights the "Current Directory" dropdown menu in the top right, with an arrow pointing to the text "Choose Current Directory". The interface is divided into several panes: "Current Directory" (showing files like findMPP.m and pv1.mdl), "Command Window" (showing the execution of the command >> 1+1 and the result ans = 2), and "Command History" (showing the command 1+1). The "Start" button is visible in the bottom left corner.

MATLAB commands and scripts can be executed interactively in the "Command Window"

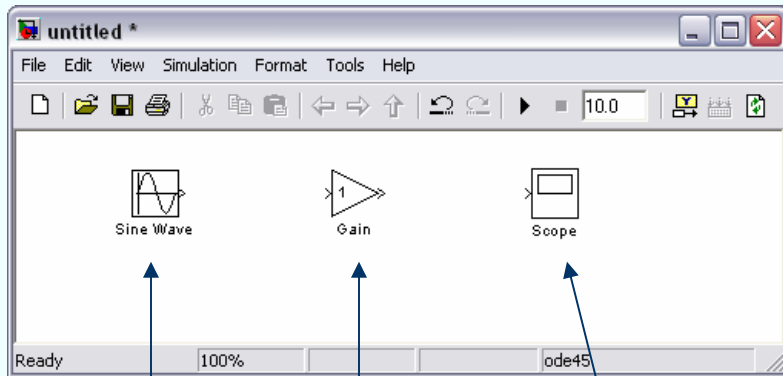
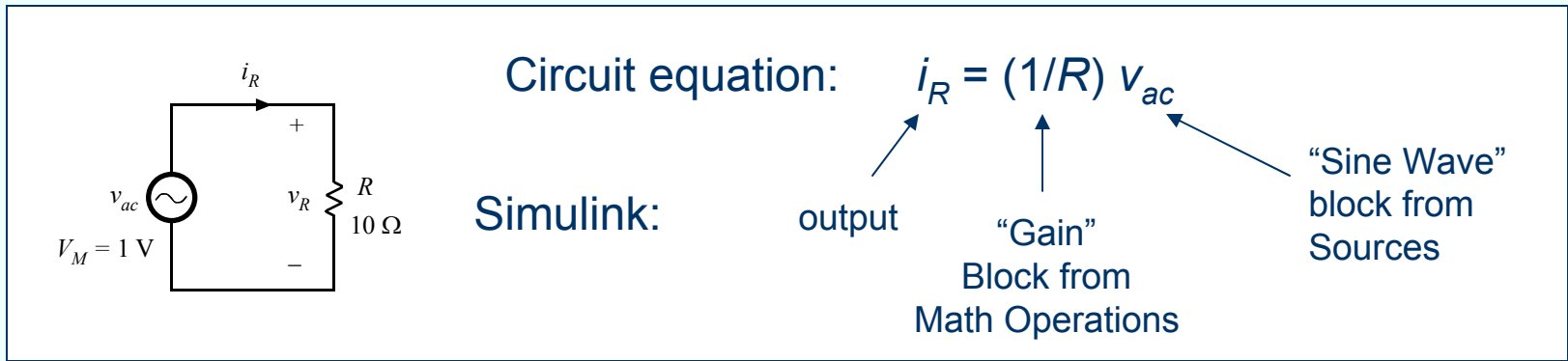
# Simulink Library Browser



- Library Browser gives access to various standard or additional blocks that are used to build more complicated models
- ECEN2060 models will be constructed using standard Simulink blocks from the Simulink library
- Click File - New – Model (or Ctrl-N) to start a new model window



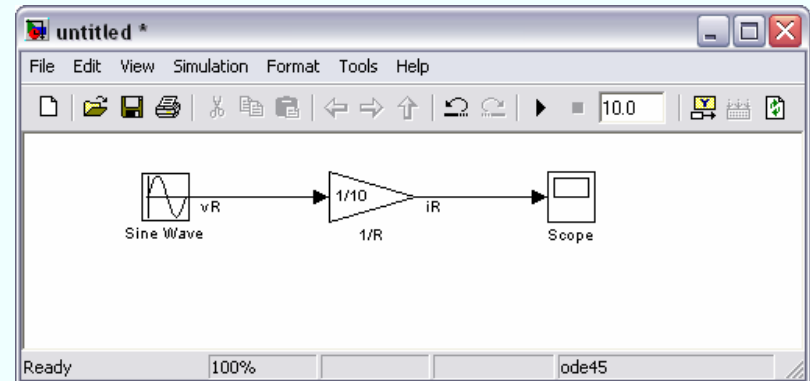
# Constructing and simulating a simple circuit model



Drag an drop Sine Wave block from Sources in the Library

Drag an drop Gain block from Math Operations in the Library

Drag an drop Scope block from Sinks in the Library



Connect block inputs and outputs according to the circuit equation

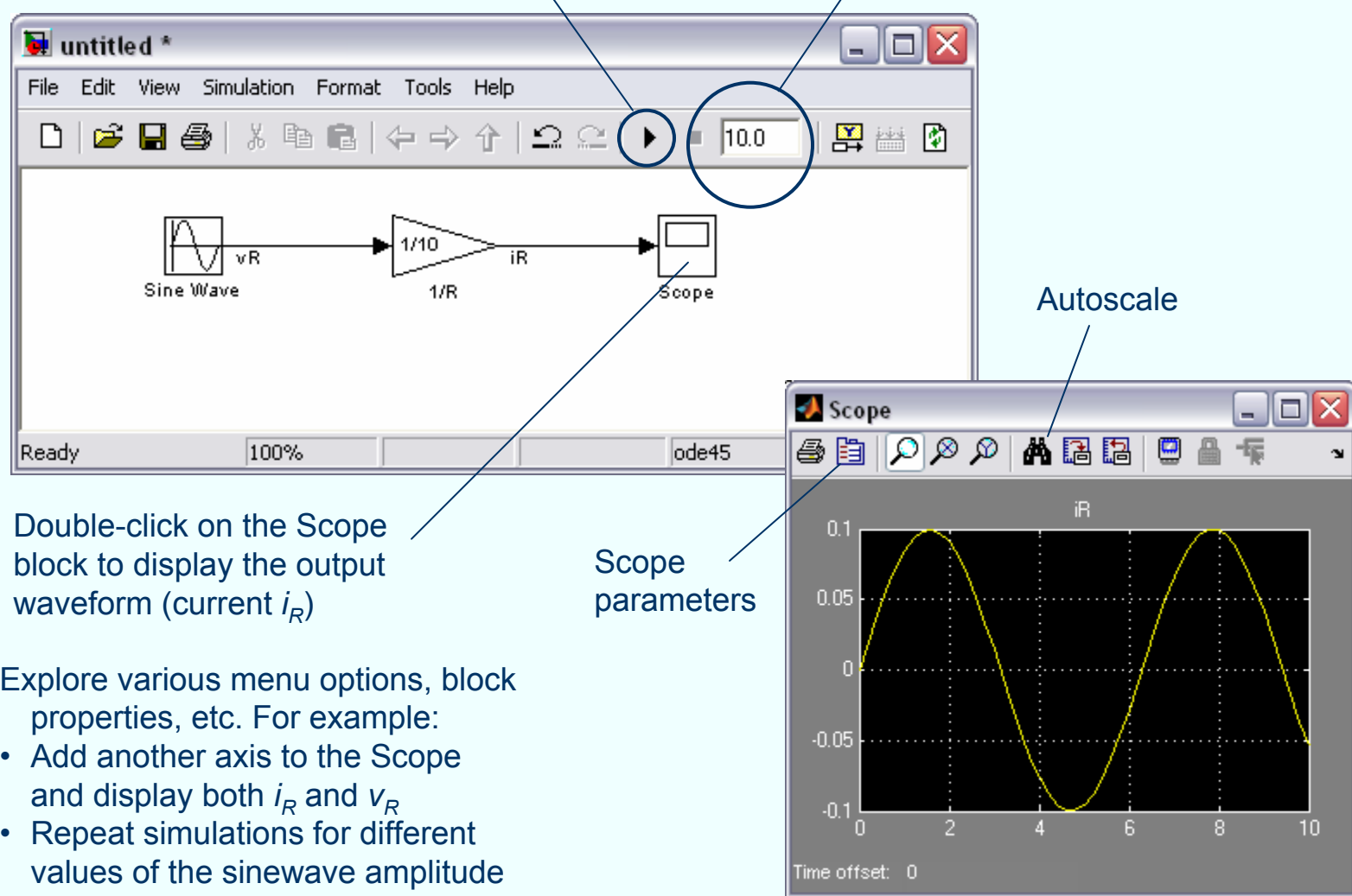
Double-click a block to change parameter values

To better document the model, double-click on the block name or on the connection line to change block and signal names as desired

# Constructing and simulating a simple circuit model

Click here to start simulation

Total simulation time (10 seconds in this example) can be changed here



The image shows two windows from a simulation software. The top window, titled 'untitled \*', contains a circuit diagram. It starts with a 'Sine Wave' block labeled  $v_R$ , which is connected to a gain block labeled  $1/10$  and  $1/R$ . The output of the gain block is labeled  $i_R$  and is connected to a 'Scope' block. The software's toolbar includes a play button (circled in blue) and a text box containing '10.0' (also circled in blue). The bottom window, titled 'Scope', displays a plot of the current  $i_R$  over time. The plot shows a yellow sine wave oscillating between approximately 0.1 and -0.1 on the y-axis, and 0 to 10 on the x-axis. The plot area is labeled 'Autoscale'.

Double-click on the Scope block to display the output waveform (current  $i_R$ )

Scope parameters

Autoscale

Explore various menu options, block properties, etc. For example:

- Add another axis to the Scope and display both  $i_R$  and  $v_R$
- Repeat simulations for different values of the sinewave amplitude or frequency, or different  $R$

# Notes

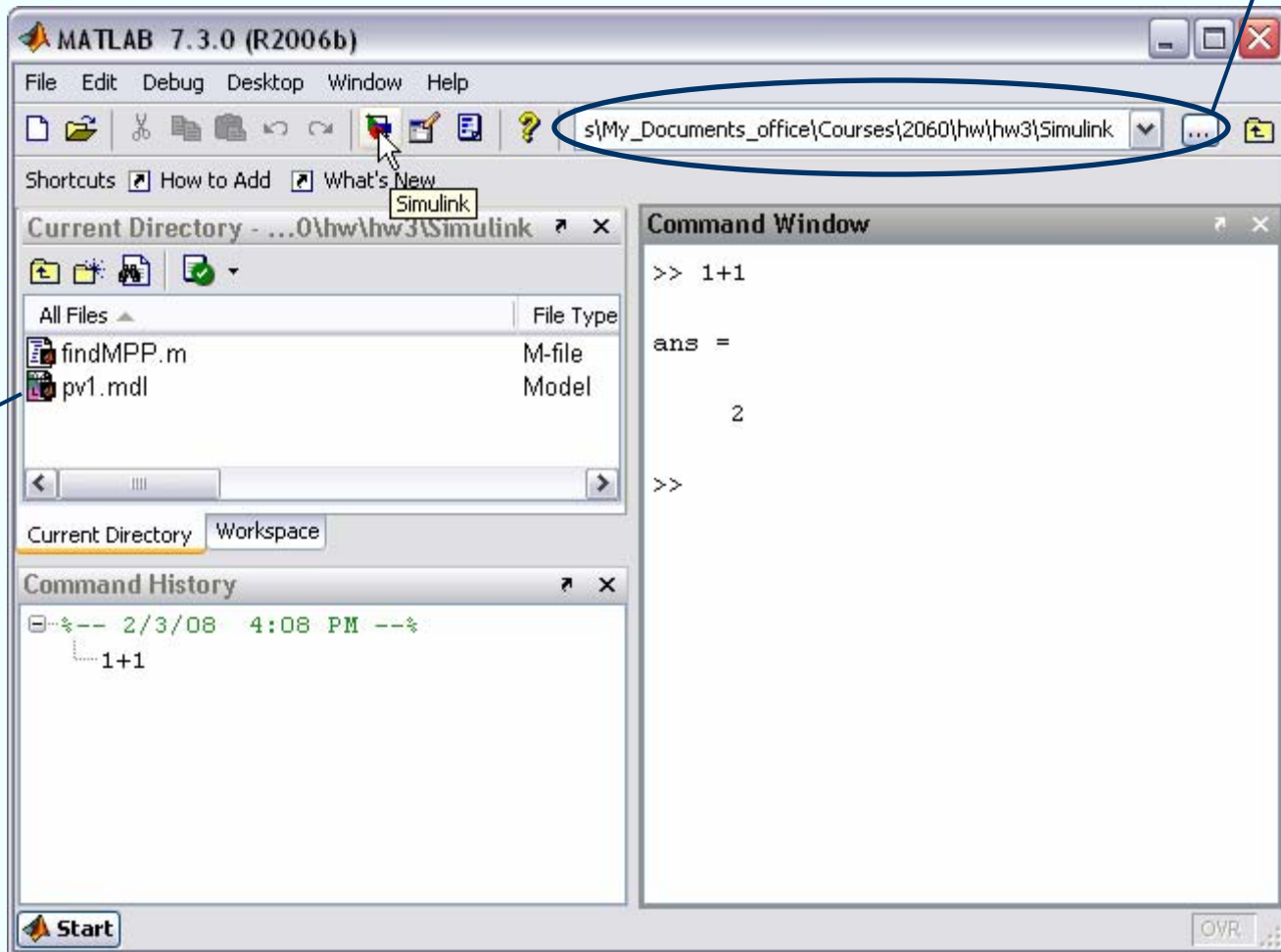
---

- Simulink blocks have zero, one or more inputs and zero, one or more outputs
- Inputs and outputs are all considered “signals”
- Unlike circuit-oriented simulators (e.g. Spice), Simulink has no knowledge of whether a signal is voltage, current, power, torque, speed, ..., which
  - makes it a very general tool for simulations of various dynamical systems, but
  - requires a user to decide which signals are inputs and which signals are outputs, and make block connections to correctly model system equations; as a result, Simulink block diagrams may not be as intuitive as circuit diagrams

# Another example: HW 3 Simulation problem

Make sure you have downloaded **pv1.mdl** and **findMPP.m** from HW3 assignment to a working folder

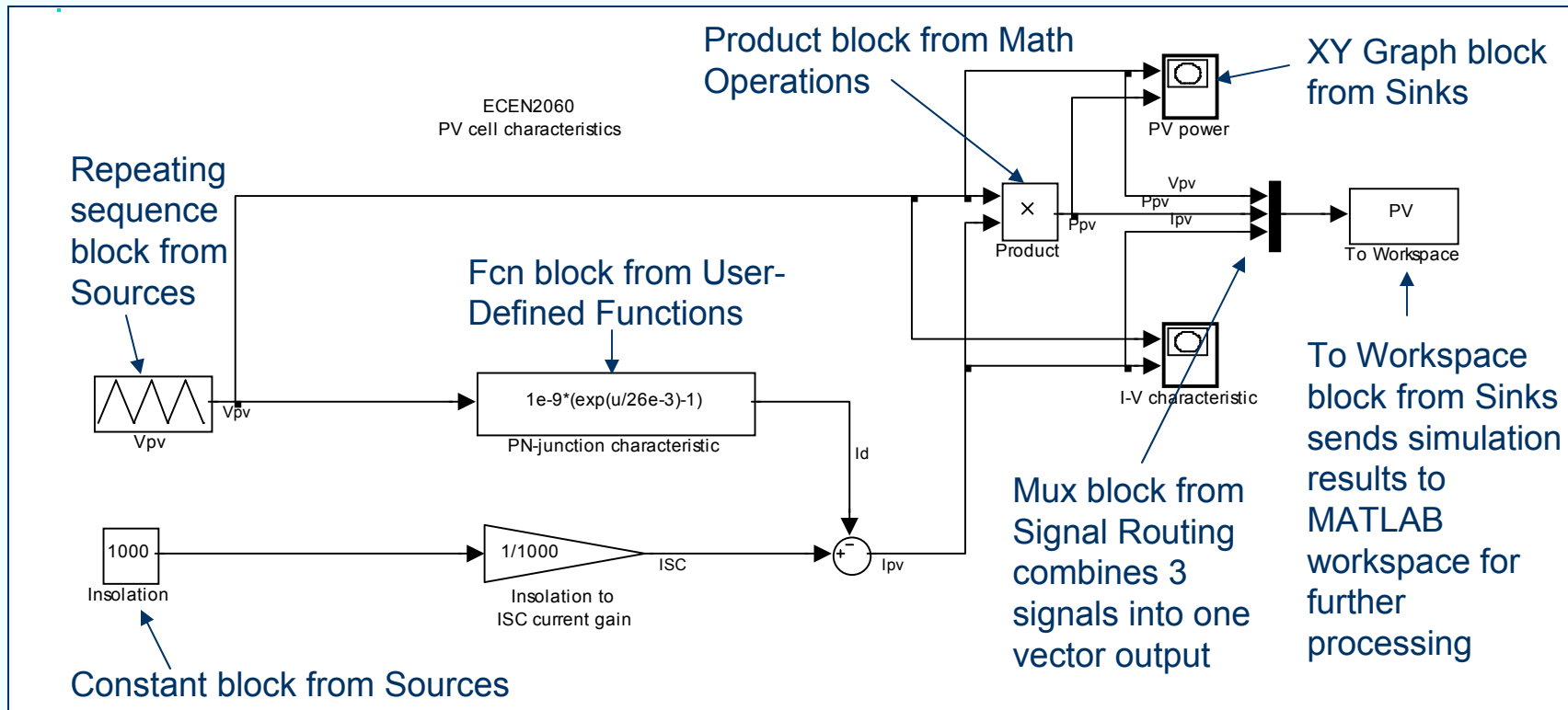
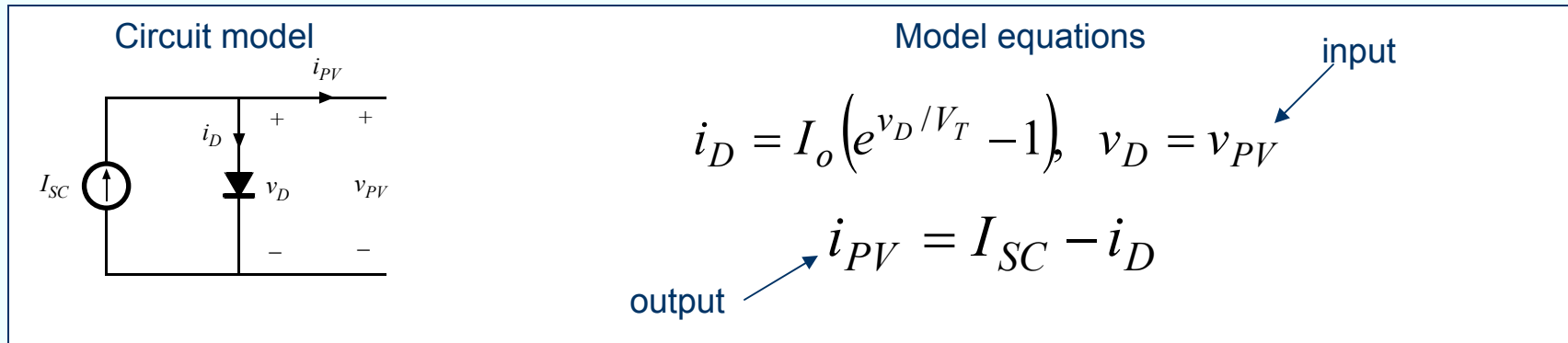
Set MATLAB Current Directory to the folder where you downloaded pv1.mdl and findMPP.m



Double-click pv1.mdl to open the PV cell Simulink model



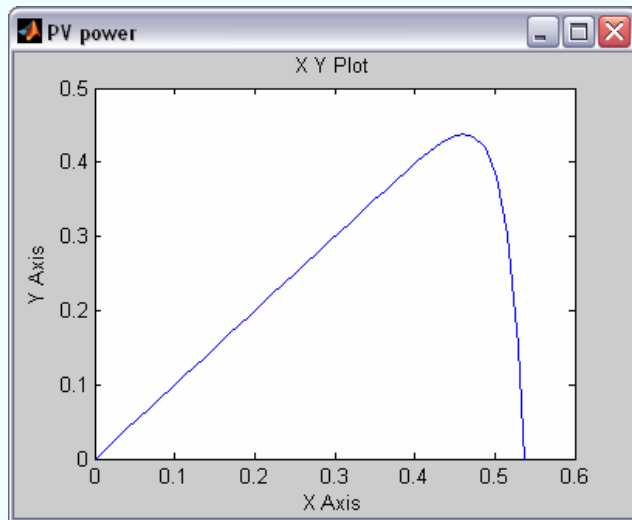
# Simple PV cell Simulink model pv1.mdl



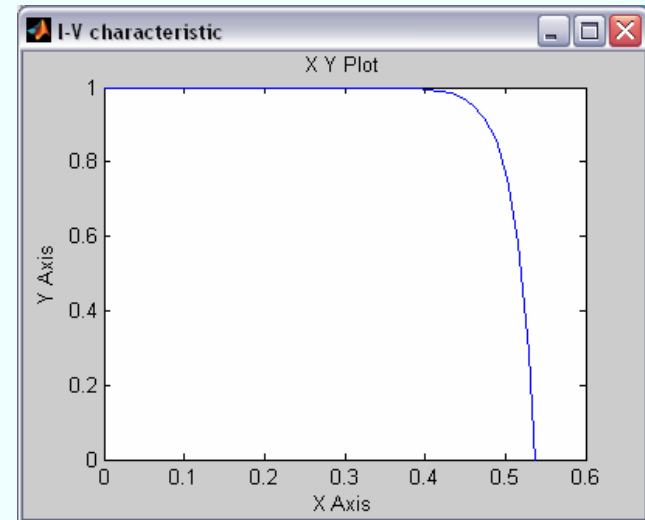
pv1.mdl Simulink model from ECEN2060 HW3 problem

# Run pv1 simulation

Output power  $P_{pv}$  (i.e the product of  $i_{PV}$  and  $v_{PV}$ ) as a function of  $v_{PV}$  is immediately displayed in a X-Y Plot window



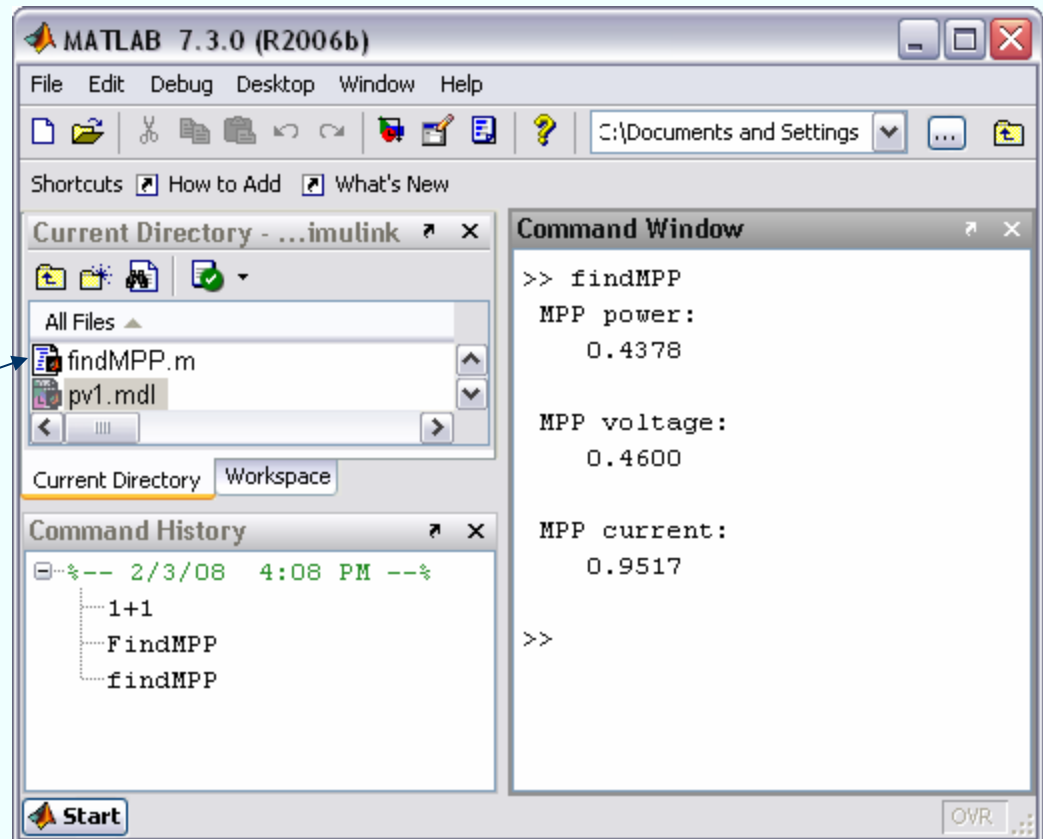
Output current  $i_{PV}$  as a function of  $v_{PV}$  is immediately displayed in another X-Y Plot window



Output power  $P_{pv}$ , current  $i_{PV}$ , voltage  $v_{PV}$ , and simulation time are stored in a “structure” variable PV, which is made available (using the “To Workspace” block) for further processing in the MATLAB Command Window

# Example of MATLAB processing of simulation results

Type “findMPP” and Enter in the MATLAB Command Window. This runs the MATLAB script **findMPP.m**, which takes the simulation results and finds the maximum power point (power, voltage and current) and again plots power  $P_{pv}$  and  $I_{pv}$  as functions of  $V_{pv}$ .



Double-click on the findMPP.m file to open the script and examine the MATLAB code

# More notes

---

- Simulink model and MATLAB Figure windows include a “Copy” function in the “Edit” menu. This is useful for reporting results: you can simply copy and paste your models or graphical results into a Word or PowerPoint document
- You may want to explore other options in the MATLAB Figure window. For example, find out how add a grid, change the line type, thickness or color, change the x-axis or y-axis scales, etc.
- This tutorial is very limited in scope, just to get you started with the tools we will be using to model and test various electrical or electro-mechanical energy systems in ECEN2060. You may want to browse through Simulink/MATLAB HELP documentation to further explore the tool capabilities