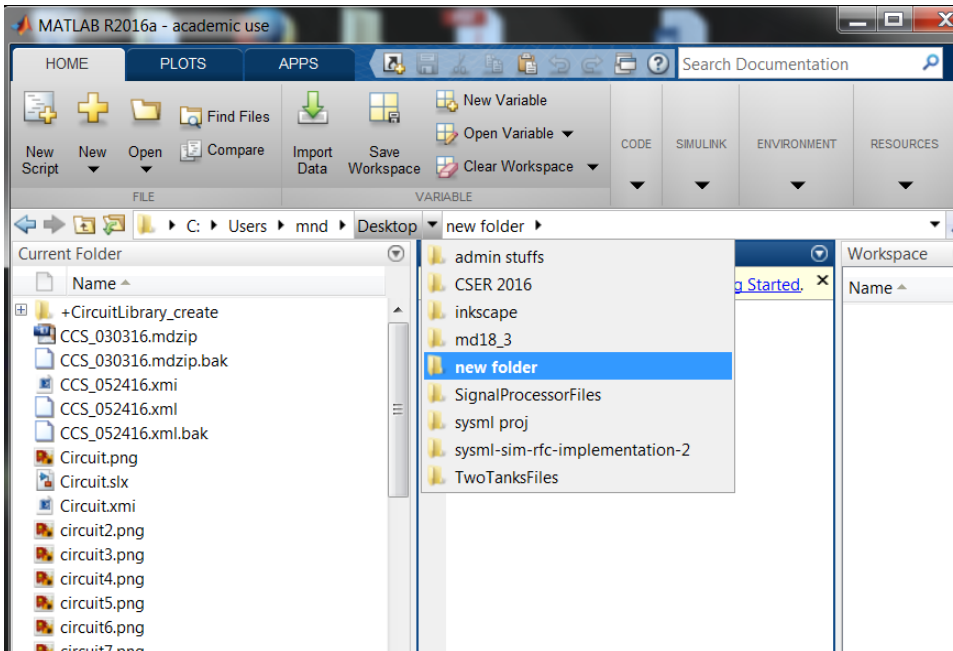


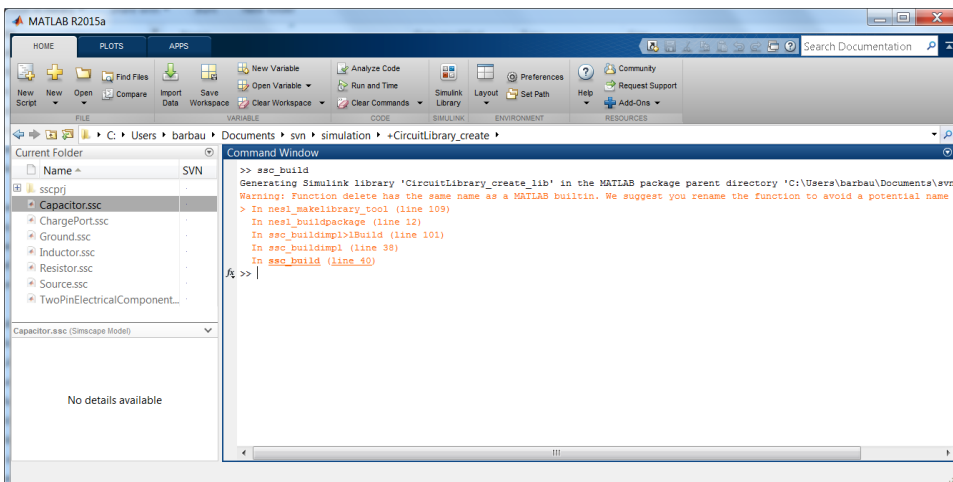
# Electrical Circuit example to Simulink and Simscape model R2016a

To generate a Simulink model directly from the modeling tool

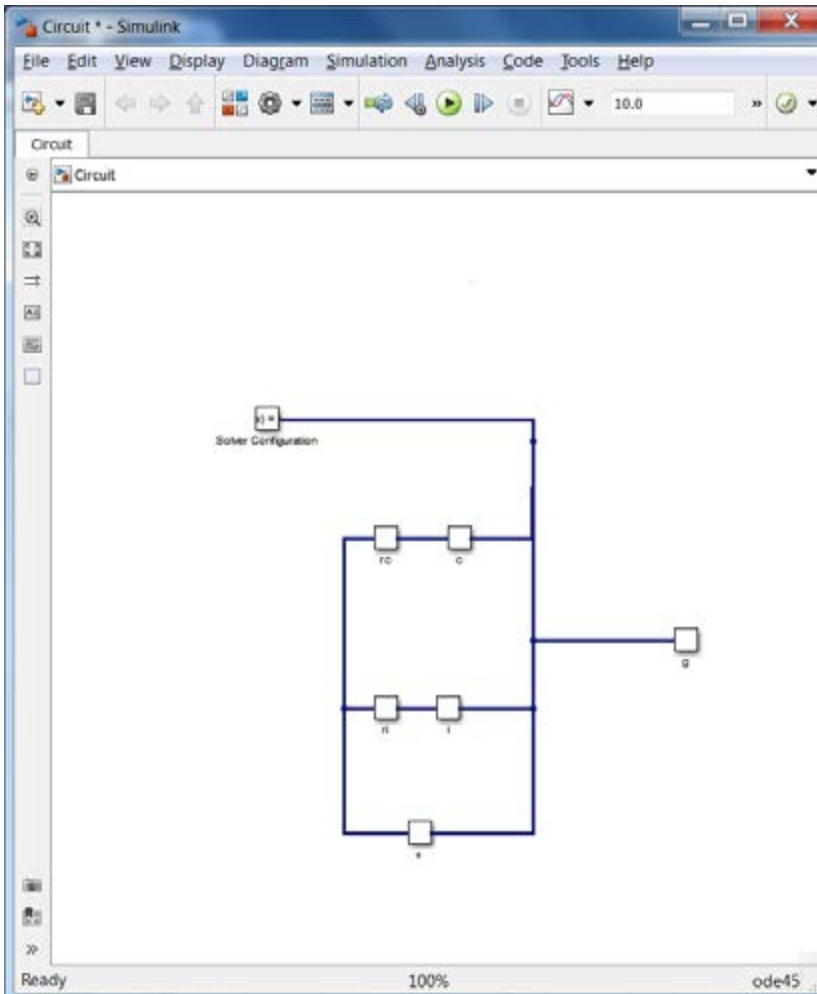
1. Right-click the *Block TestCaseCircuit::Circuit* and select **Tools > Export to Simulink**. Please see [Generating a simulation file](#) for more information.
2. Set the options listed below and click **OK**:
  - **Format:** XML (.sdl)
  - **S-Function or Simscape:** Simscape
  - **Simscape port libraries:** Create new port types
3. Launch Matlab with Simulink and Simscape extensions.
4. In Matlab's **Current Directory** navigation bar, search for the file directory where the Simulink/Simscape files (as well as the SysML files) are saved, and set it as the current directory. The **Current Folder** panel should display the generated files.



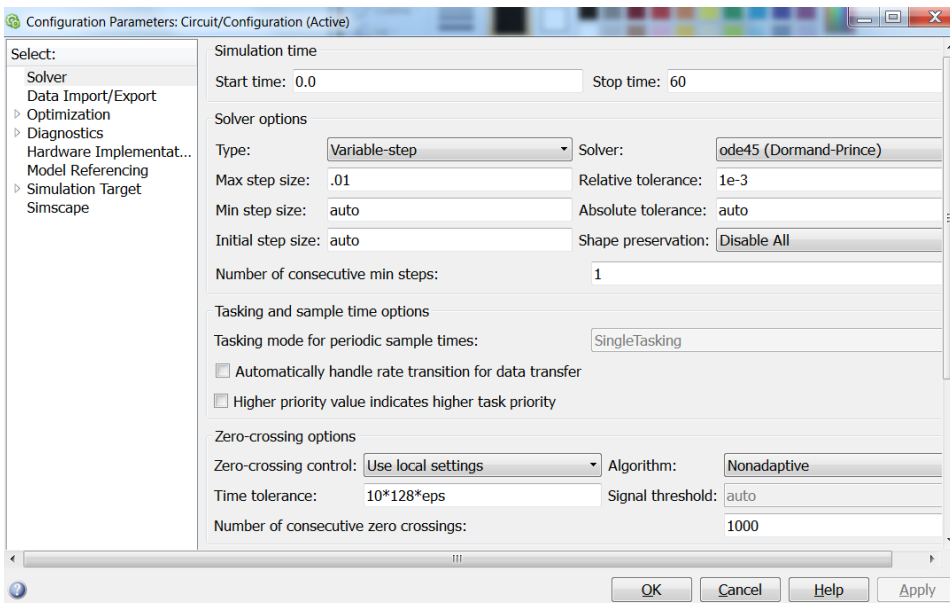
5. In the **Current Folder** panel, double-click on the folder named **+CircuitLibrary** (this sets the current directory to this folder). Type **ssc\_build** (or **sbuild** depending on the Matlab version) in the **Command Window** and press **Enter**. This will generate a folder **sscprj** in the **+CircuitLibrary** directory as well as a **CircuitLibrary\_lib.slx** file in the directory with the original **Circuit.slx** file. Next, once again, search for and go back to the file directory where the Simulink/Simscape files (as well as the SysML files) are saved, and then set it as the current directory.



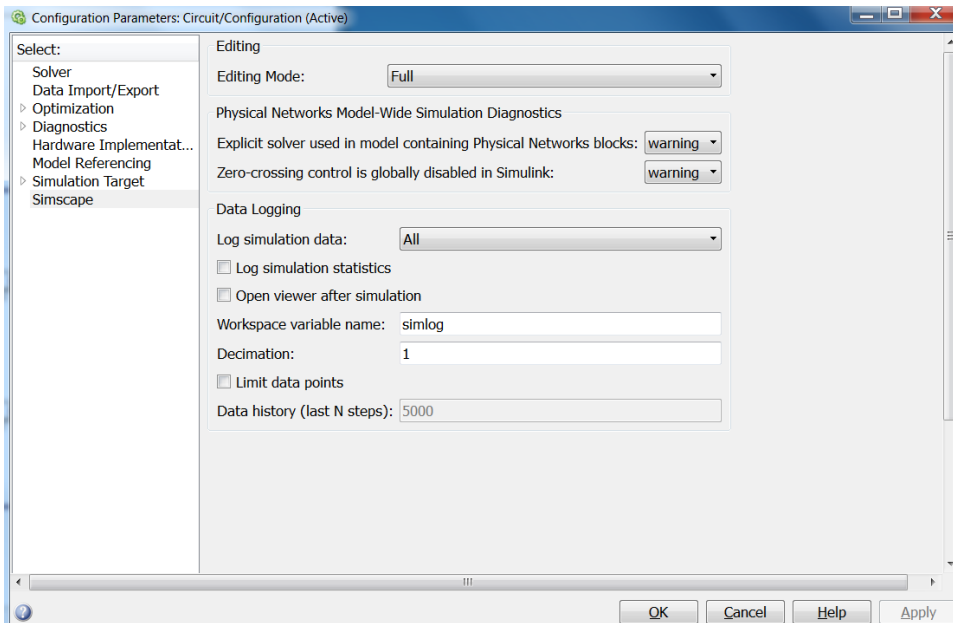
6. In the **Current Folder** panel, double-click on **Circuit.slx** to open the Simulink/Simscape model. Rearrange the blocks to get a better picture of the blocks and links.



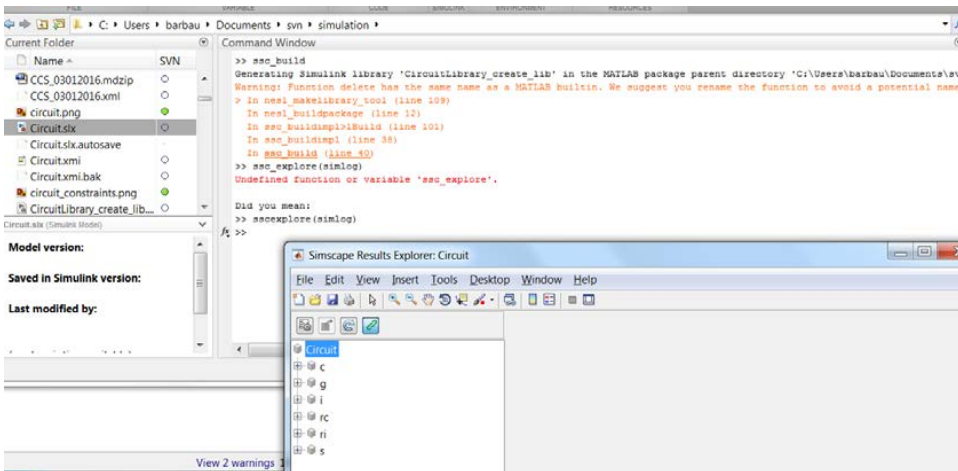
- Go to **Simulation > Model Configuration Settings**. This allows to select the types of solver and the runtime of the simulation. Select a **Start Time** of **0**, and a **Stop Time** of **60** (or any other reasonable number of seconds). Change the **Max step size** to **0.01**. Under **Solver Options**, keep the solver **Type** to **Variable-step** (or any other desirable solver that is suitable). Press **Apply**.



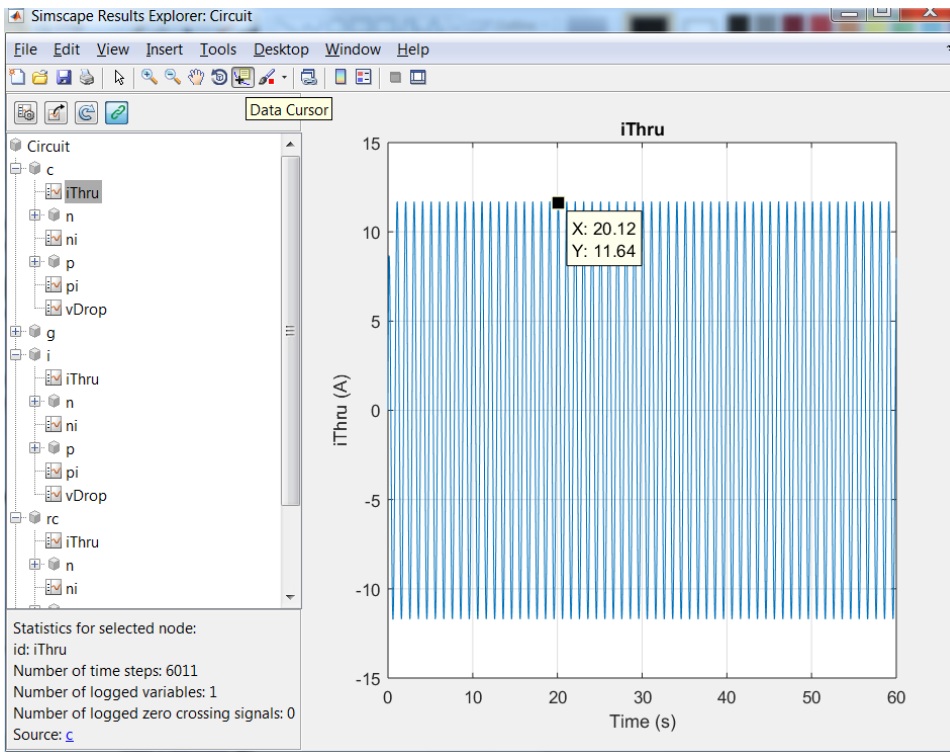
- In **Model Configuration Settings**, select **Simscape** in the **Commonly Used Parameters** panel. Change **Log simulation data** to **All**. Make sure **Limit data points** is unchecked. Press **Apply**. Press **OK**.



9. In the **Circuit** Simulink window, go to **Simulation > Run**.
10. Once the model has compiled, go to Matlab's **Command Window**, type `ssc_explore(simlog)` (make sure that either the version of Matlab has the function `ssc_explore`, or a file `ssc_explore.m` is in the same directory as **Circuit.slx** in the **Current Folder** panel) and press Enter. A **Simscape data logging explorer: Circuit** window will open up.



11. In the **Simscape data logging explorer**, the **Circuit** blocks can be expanded to see how their properties simulated through time. For the circuit components **c** (capacitor), **i** (inductor), **rc** (resistor in series with the capacitor), **ri** (resistor in series with the inductor), and **s** (source), the properties **iThru** (current) and **vDrop** (voltage) can be selected to see the flow of charge (current) and its potential to flow (voltage) through each component. To see the simulation's specific data points, the **Data cursor** icon needs to be selected and points on the plot need to be chosen.



12. To change the default value or initial value parameters of the model, double-click the blocks in the **Circuit Simulink** model. A window will pop out for block parameters that can be changed. Run through Steps 4 through 8 to run the simulation with (new) block parameters again.

