# Signal Processor example to Simulink model R2014a

To generate a Simulink model directly from the modeling tool

1. Right-click the *Block* **TestCaseSignalProcessor::TestBed** 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**: S-Function version 2
3. Launch Matlab with Simulink and Simscape extensions.
4. In Matlab's **Current Directory** navigation bar, search for the file directory where the Simulink files (as well as the SysML files) are saved, and set it as the current directory. The **Current Folder** panel should display the generated **.slx** and **M-files**.
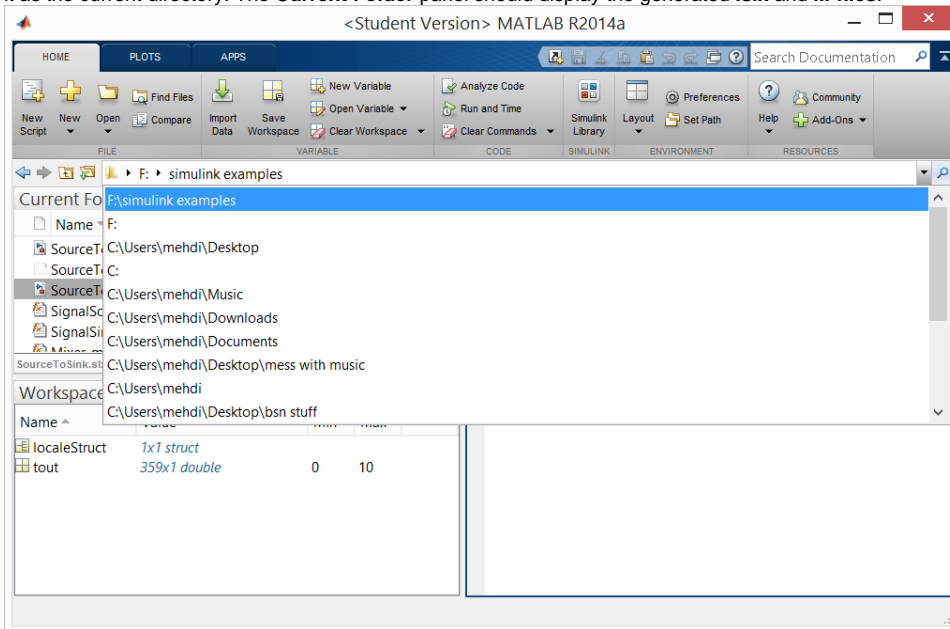


5. In the **Current Folder** panel, double-click on the file named **SourceToSink.slx**. The **SourceToSink** model will open up in a new Simulink window. Rearrange the blocks to reflect the signal flow arrowheads.
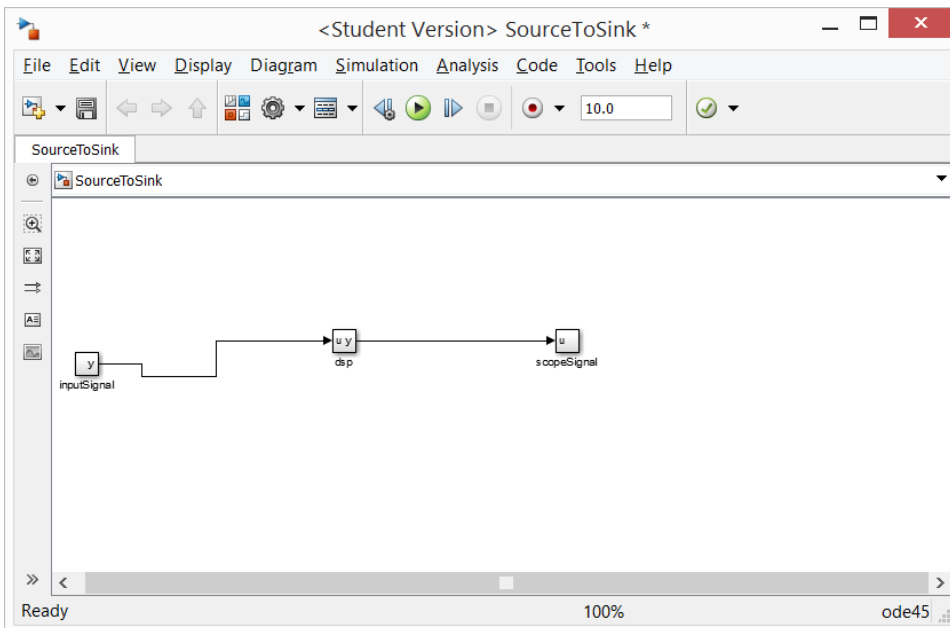


6. Open the **Library Browser** either by using the **4-block** icon or by going to **Tools** > **Library Browser**. Find and click on **Sinks** among the list in the **Libraries** panel, which will be categorized **under Simulink**.
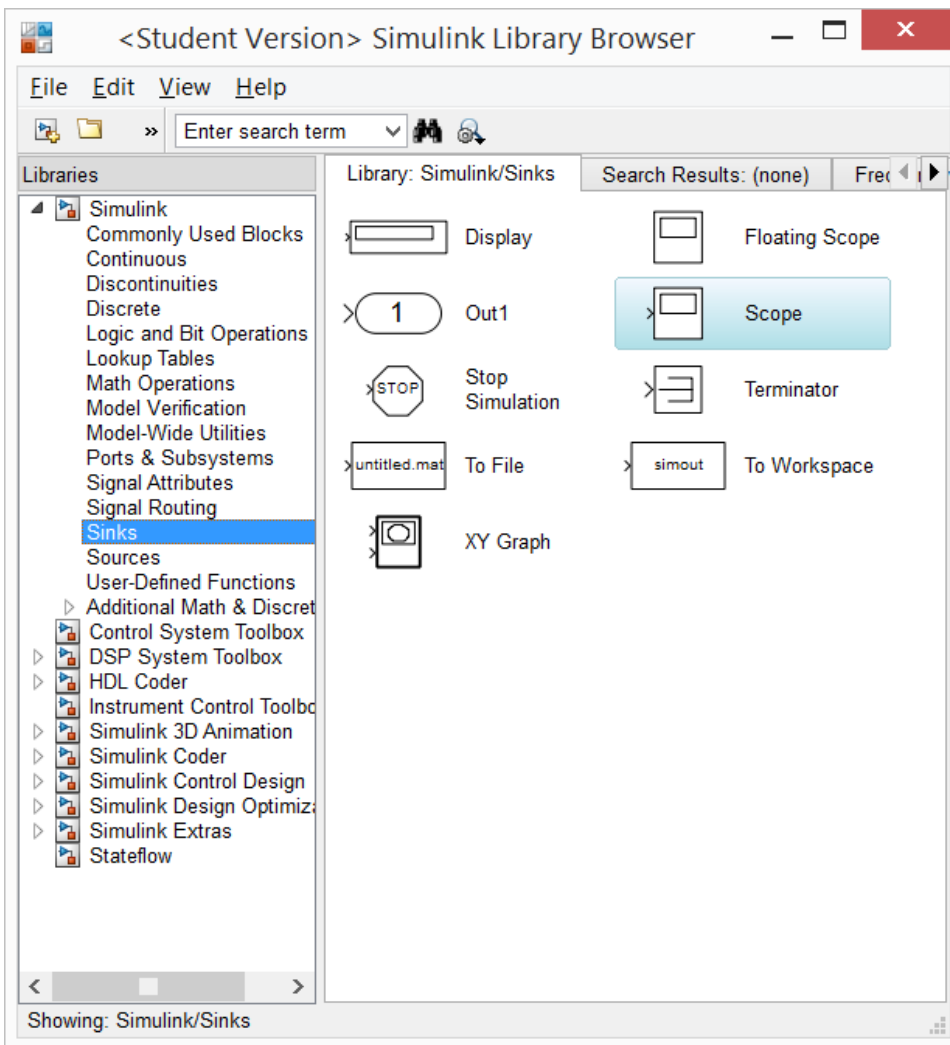
**7.** Pull-and-drag the **Scope** block from the **Sinks** library list into the **SourceToSink Simulink** model. Attach a signal line segment from the Scope port onto the signal line that connects the **dsp** block and the **scopeSignal** block. Repeat Steps 6 & 7 to add a **Scope** block and attach it to the signal line segment that connects the **inputSignal** block to the **dsp** block.
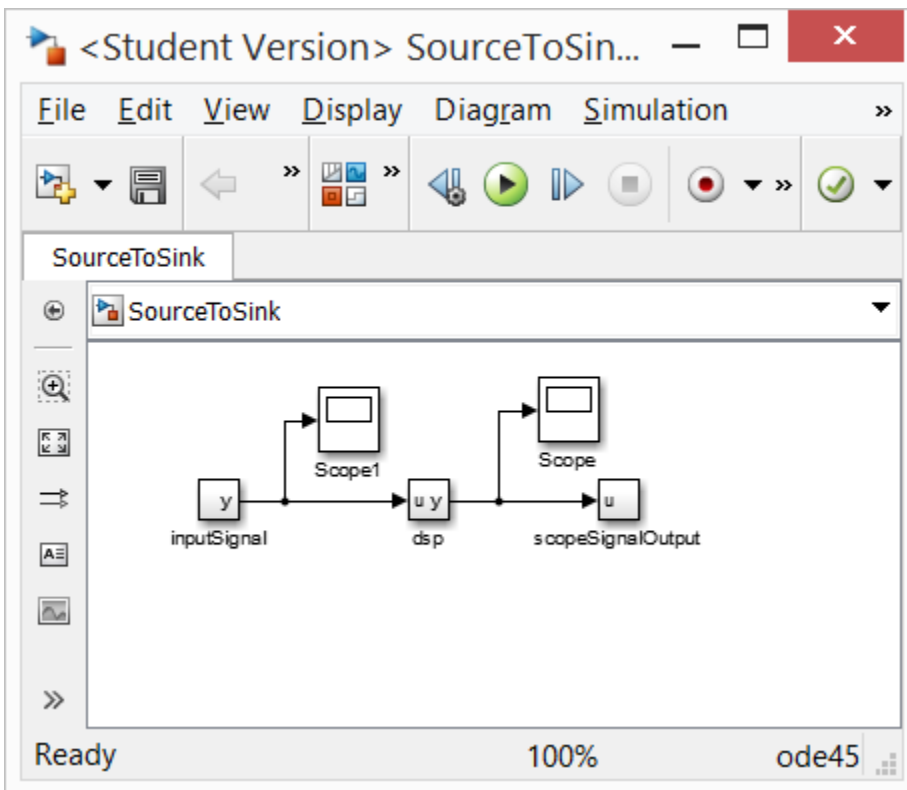
8. Go to **Simulation** > **Model Configuration Settings**. This allows selecting the types of solver and the runtime of the simulation. Select a **Start Time** of **0**, and a **Stop Time** of **20** (or any other reasonable number of seconds). Under **Solver Options**, change the solver **Type** to **Variable-step** (or any other desirable solver that is suitable). Set the **Max step size** to **0.01**. Press **Apply**, then press **OK**.
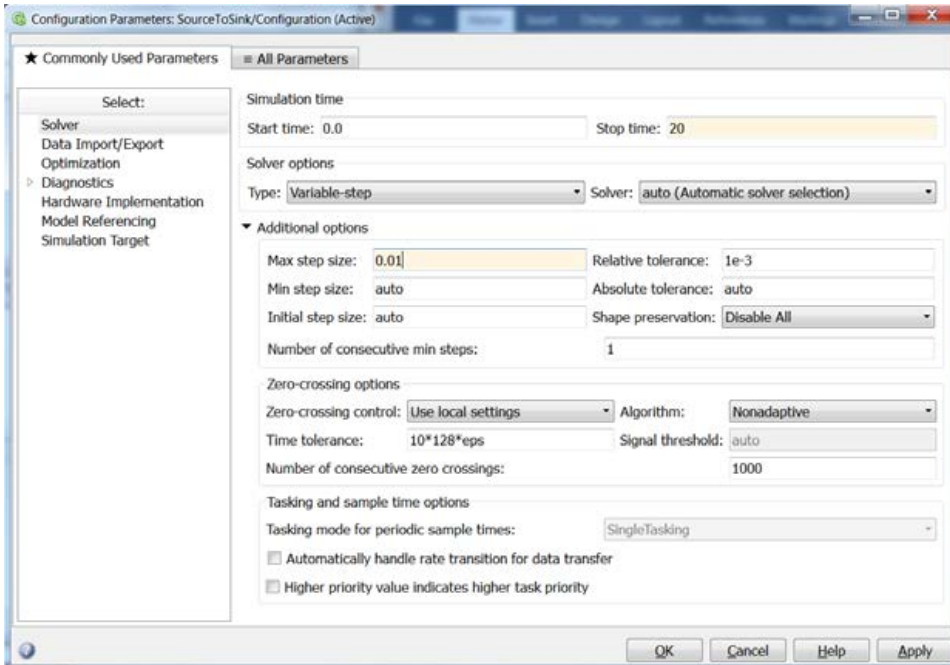


9. Go to **Simulation** > **Run**. Double-click on the **Scope** block to see the simulation results.

Time offset: 0



Time offset: 0