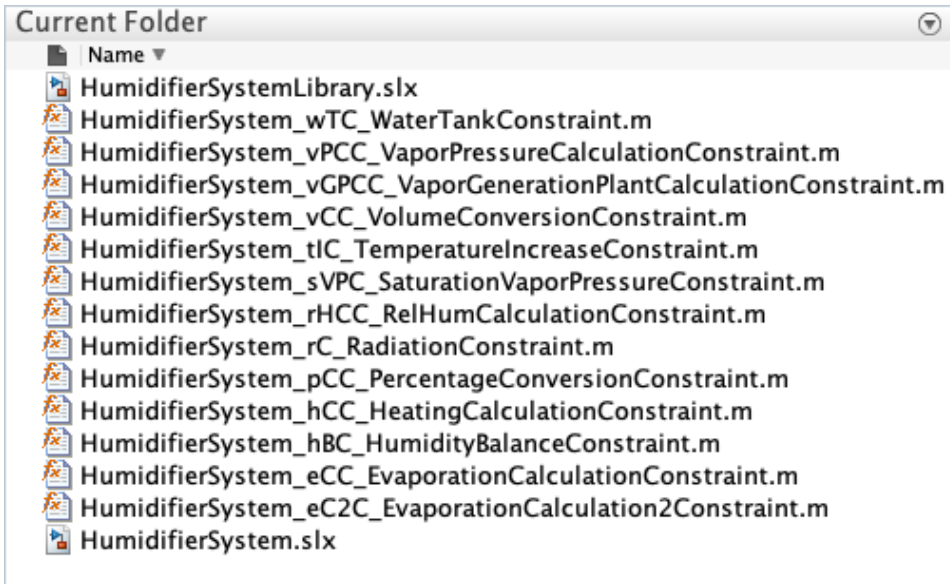


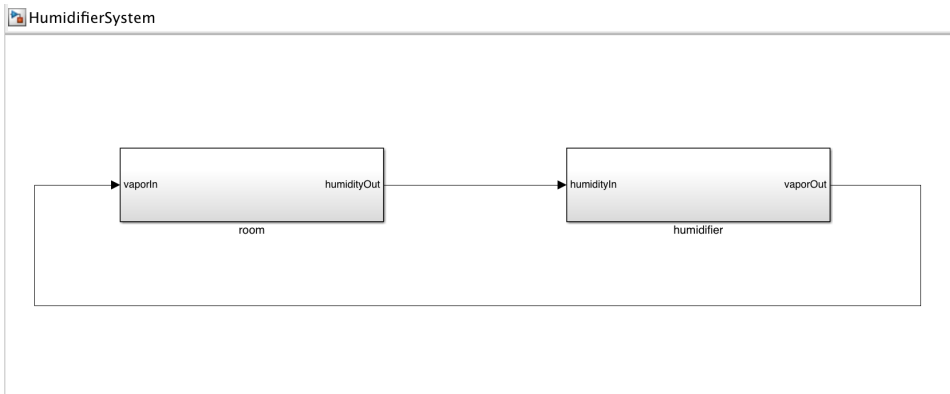
Exporting Humidifier example to Simulink model

To generate a Simulink model directly from the modeling tool

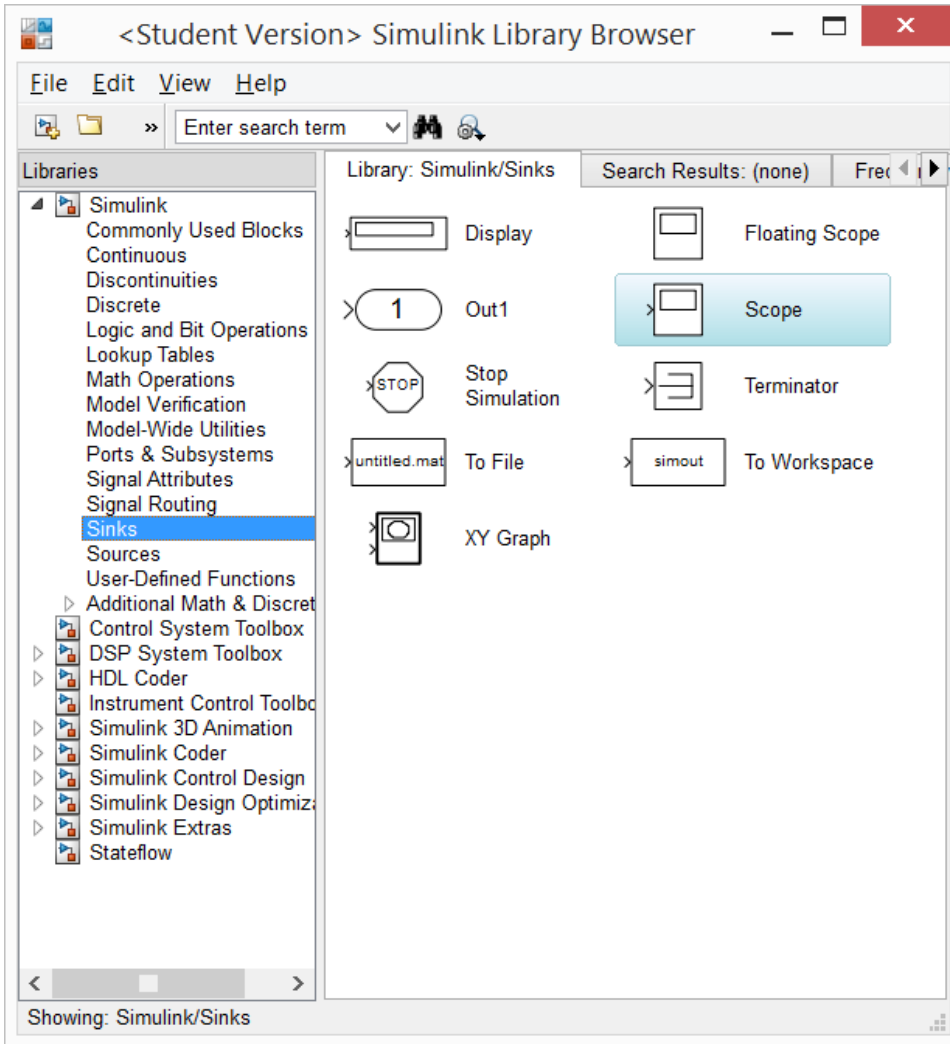
1. Right-click the *Block HumidifierSystem* and select **Tools > Export to Simulink**.
2. Set the options listed below and click **OK**:
 - **Format**: XML (.sdl)
 - **S-Function or Simscape**: S-Function version 2
3. Launch Matlab with Simulink and Stateflow extensions.
4. In Matlab's **Current Directory** 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 generated Simulink (.slx) files.



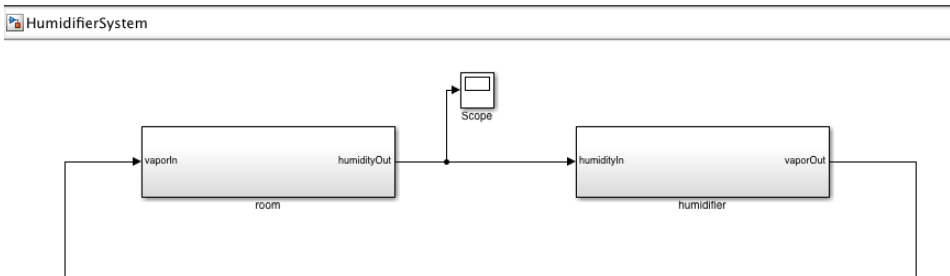
5. In the **Current Folder** panel, double-click the file named **HumidifierSystem.slx**. The model will open up in a Simulink window.



- 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**.



- Pull-and-drag the **Scope** block from the **Sinks** library list into the **HumidifierSystem** Simulink model. Attach a signal line segment from the **Scope** port onto the signal line that connects the *humidity-out* port of the room block to the *humidity-in* port of the humidifier block.



- Go to **Simulation > Model Configuration Settings**. This allows selecting the types of solver and the runtime of the simulation. Select a **Start Time** of **0.0**, and a **Stop Time** of **3600**. Under **Solver Options**, select the solver **Type** to **Variable-step** (or any other desirable solver that is

suitable) and change the **Max step size** to **0.1**. Press **Apply**, then press **OK**.

★ Commonly Used Parameters ≡ All Parameters

Select:

- Solver
- Data Import/Export
- Optimization
- Diagnostics
- Hardware Implementation
- Model Referencing
- Simulation Target
- Code Generation
- Coverage

Simulation time

Start time: 0.0 Stop time: 3600

Solver options

Type: Variable-step Solver: auto (Automatic solver selection)

Additional options

Max step size: 0.1 Relative tolerance: 1e-3

Min step size: auto Absolute tolerance: auto

Initial step size: auto Shape preservation: Disable All

Number of consecutive min steps: 1

Zero-crossing options

Zero-crossing control: Use local settings Algorithm: Nonadaptive

Time tolerance: 10*128*eps Signal threshold: auto

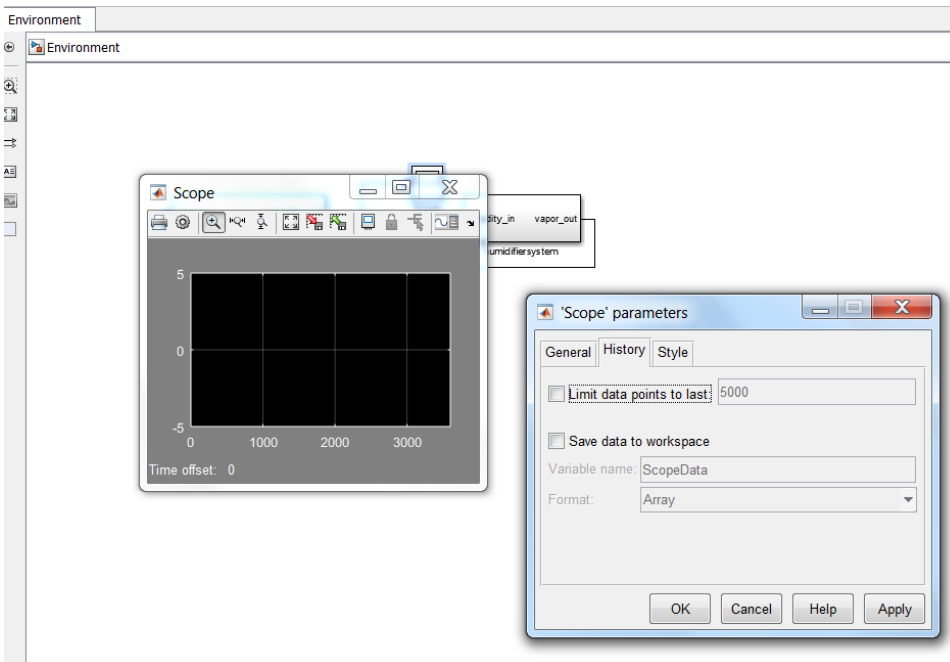
Number of consecutive zero crossings: 1000

Tasking and sample time options

☐ Automatically handle rate transition for data transfer

☐ Higher priority value indicates higher task priority

- Double-click the **Scope** block in the model. An empty (black) plot will show up. Press the **'Scope' parameters** button (the gear icon above the empty plot). A **'Scope' parameters** dialogue box will pop up. Go to the **History** tab and uncheck the option box **Limit data points to last**. Press **Apply**, and then **OK**.



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

