

Humidifier example to Simulink model

This page describes how to simulate the *Humidifier* example model in Simulink. [Learn more about Humidifier sample model >>](#)

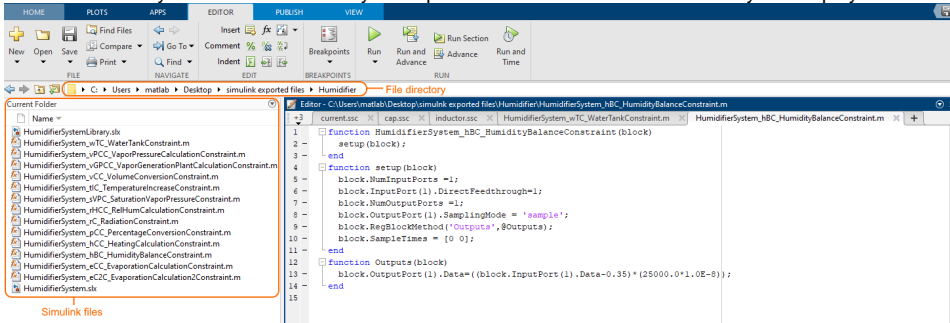


You can find the *Humidifier* sample model in:

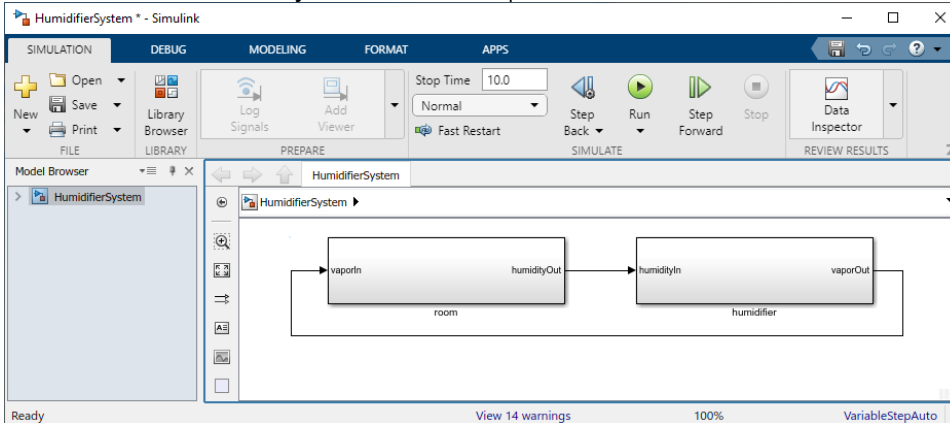
- the modeling tool: Welcome window > Samples > Simulink and Modelica Transformation > Humidifier.
- the Installation directory: `<modeling tool installation directory>\samples\SysML\Simulink and Modelica Transformation\Humidifier`.

To simulate the *Humidifier System* model

1. Export the *HumidifierSystem* Block to Simulink file. [How to >>](#)
2. In the **Simulink Export Options** dialog select the following options:
 - **Format:** XML (.slx)
 - **S-Function or Simscape:** S-Function version 2
 - **Composite Signals:** Bus Creators/Selectors
3. Make sure the MATLAB tool is installed.
4. Double-click the MATLAB icon to start it.
5. In the file directory select the location of your exported Humidifier Simulink files. They are displayed in the **Current Folder** panel.

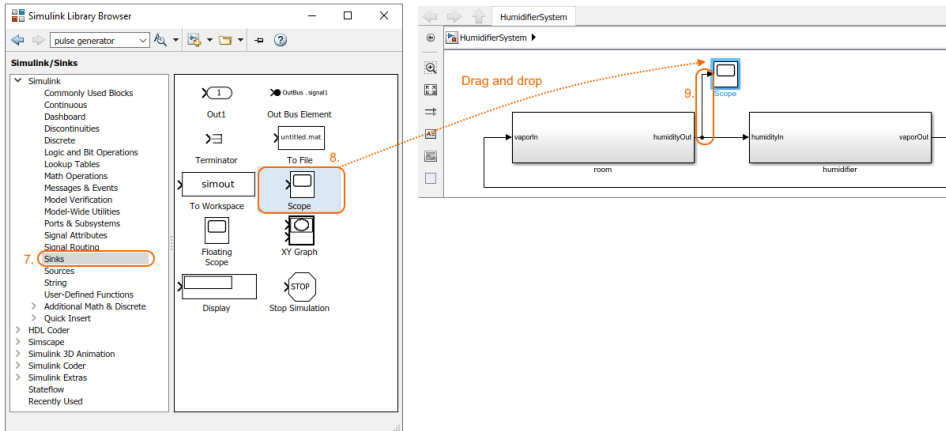


6. Double-click the file **HumidifierSystem.slx**. The model open in the **Simulink** window.

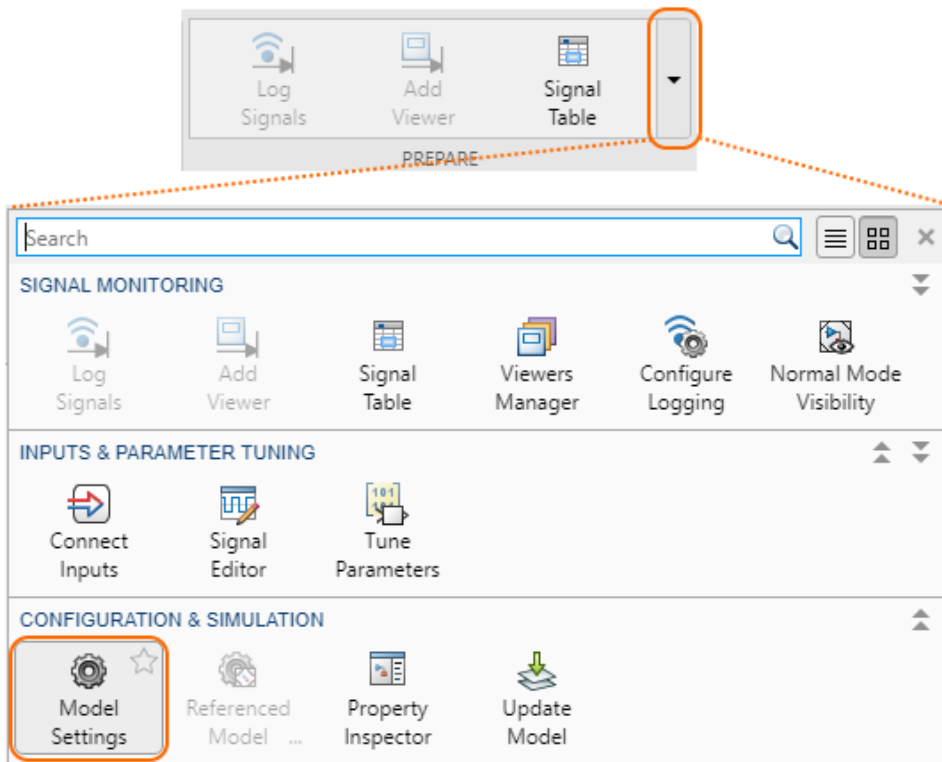


7. On the **SIMULATION** tab, click **Library Browser**.
8. In the **Simulink Library Browser** dialog, select expand the **Simulink** group and select **Sinks**. See the figure below.
9. Drag and drop the Scope block onto the **HumidifierSystem** Simulink model. See the figure below.

- Select signal line that connects the *humidity-out* port of the room block to the *humidity-in* port of the humidifier block and click on the Scope block. See the figure below.



- On the **SIMULATION** tab, click the arrow on the **PREPARE** area and select the **Model Settings**.



- In the **Configuration Parameters** dialog, on the left side of the dialog select a **Solver** and specify the following on the right side of the dialog:
 - In the **Simulation time** group change **Start time** to **0.0**, **Stop time** to **3600**.

- In the **Solver selection** group select **Type Variable-step** or any other desirable solver that is suitable.
- Click the arrow of the **Solver details** group and change the **Max step size** to **0.1**.

Configuration Parameters: HumidifierSystem/Configuration (Active)

Search

Solver

- Data Import/Export
- Math and Data Types
- Diagnostics
- Hardware Implementation
- Model Referencing
- Simulation Target

Simulation time

Start time: 0.0 Stop time: 3600.0

Solver selection

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

▼ Solver details

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

Min step size: auto Absolute tolerance: auto

Initial step size: auto ☒ Auto scale absolute tolerance

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

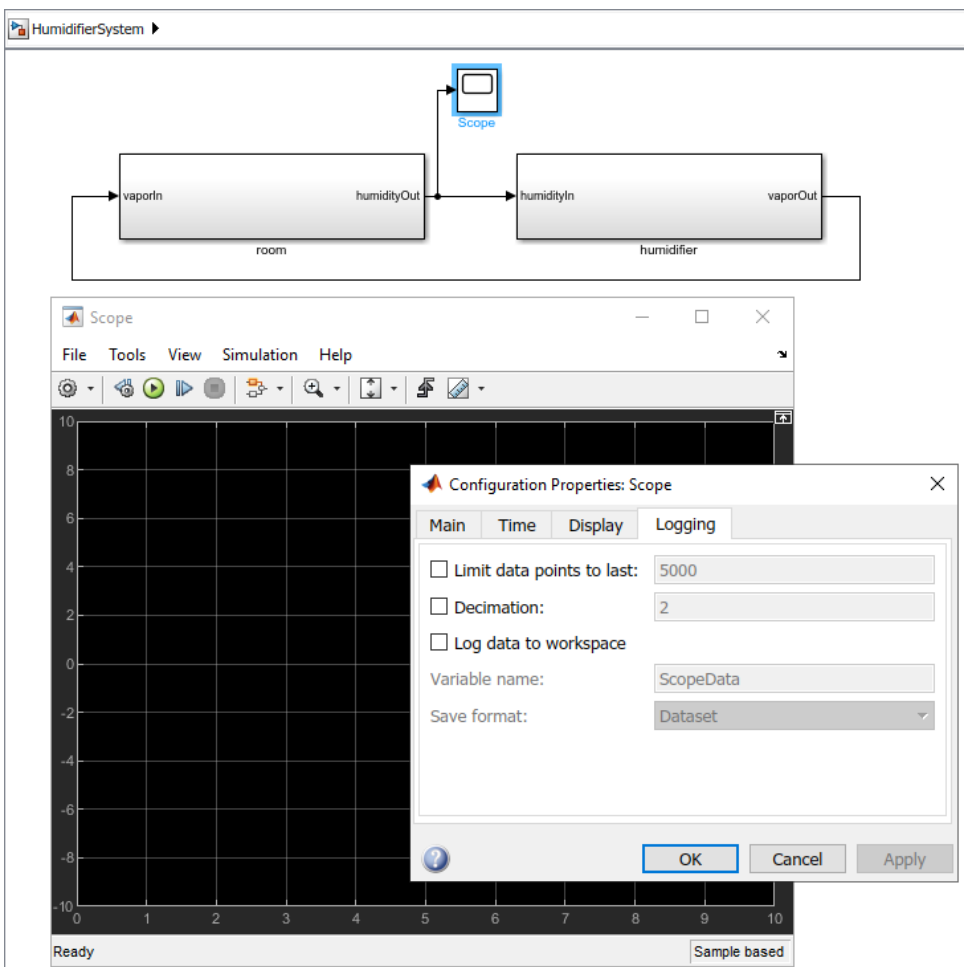
☐ Higher priority value indicates higher task priority

OK Cancel Help Apply


13. Press **Apply** > **OK**.

14. Double-click the **Scope** block in the model. An empty black plot opens.

15. Click button. In the **Configuration Properties: Scope** dialog, select the **Logging** tab and clear the **Limit data points to last** box.



16. Press **Apply** > **OK**.

17. In the Scope plot toolbar, click  and wait for the simulation results in the plot.

