Hydraulics example to Simulink and Simscape model

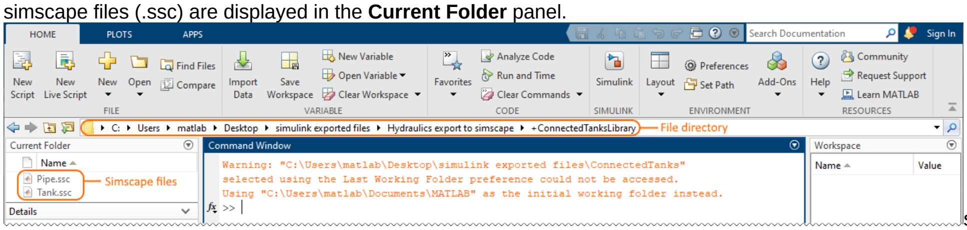
This page describes how to simulate the *Hydraulics* example model in Simulink. Learn more about *Hydraulics* sample model >>

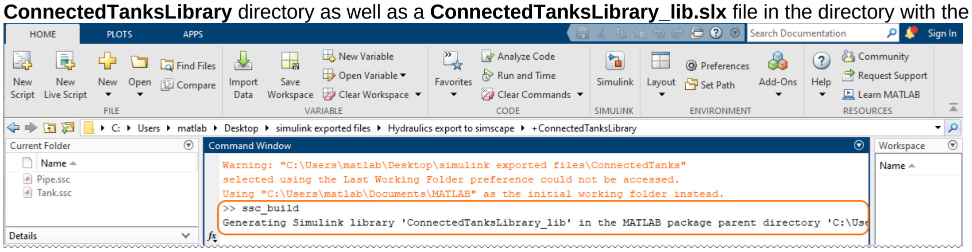
You can find the *Hydraulics* sample model in:

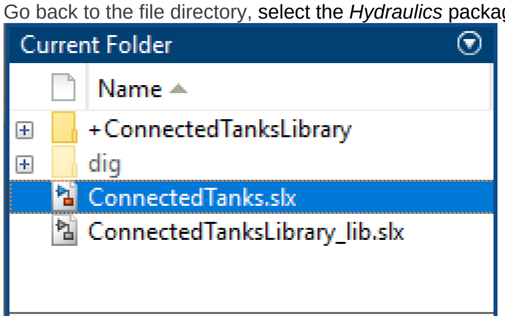
- the modeling tool: Welcome window > Samples > Simulink and Modelica Transformation > Hydraulics .
- the Installation directory: *<modeling tool installation directory>\samples\SysML\Simulink and Modelica Transformation\Hydraulics.*

To simulate the *Hydraulics* model

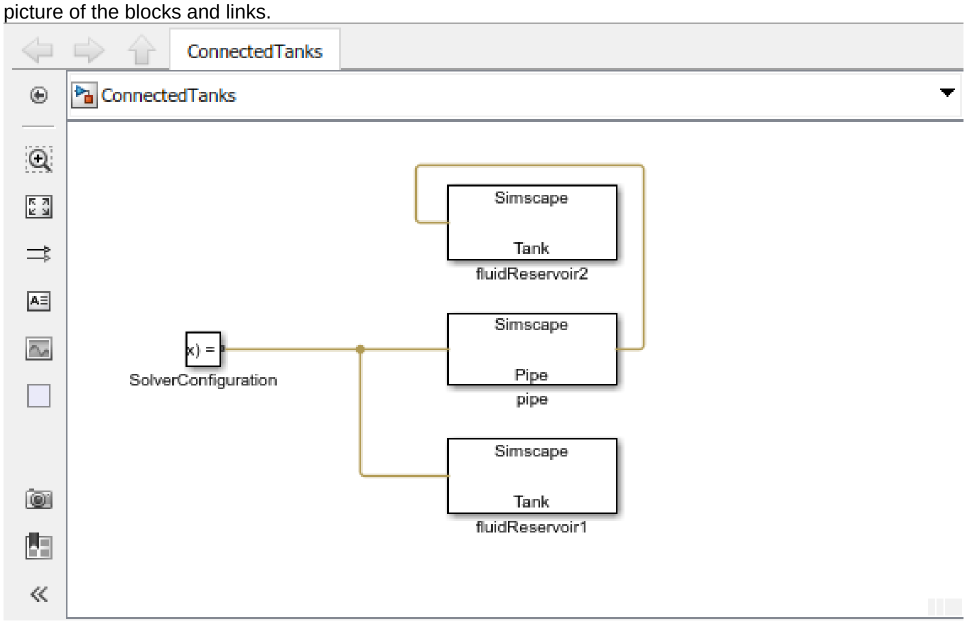
---

1. Export the *ConnectedTanks* Block to Simulink file. How to >>
2. In the **Simulink Export Options** dialog select the following options:
   - **Format**: XML (.slx)
   - **S-Function or Simscape**: Simscape
   - **Simscape port libraries**: Create new port types
   - **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 *Hydraulics* model Simulink files and choose the *+ConnectedTanksLibrary* folder. The simscape files (.ssc) are displayed in the **Current Folder** panel.


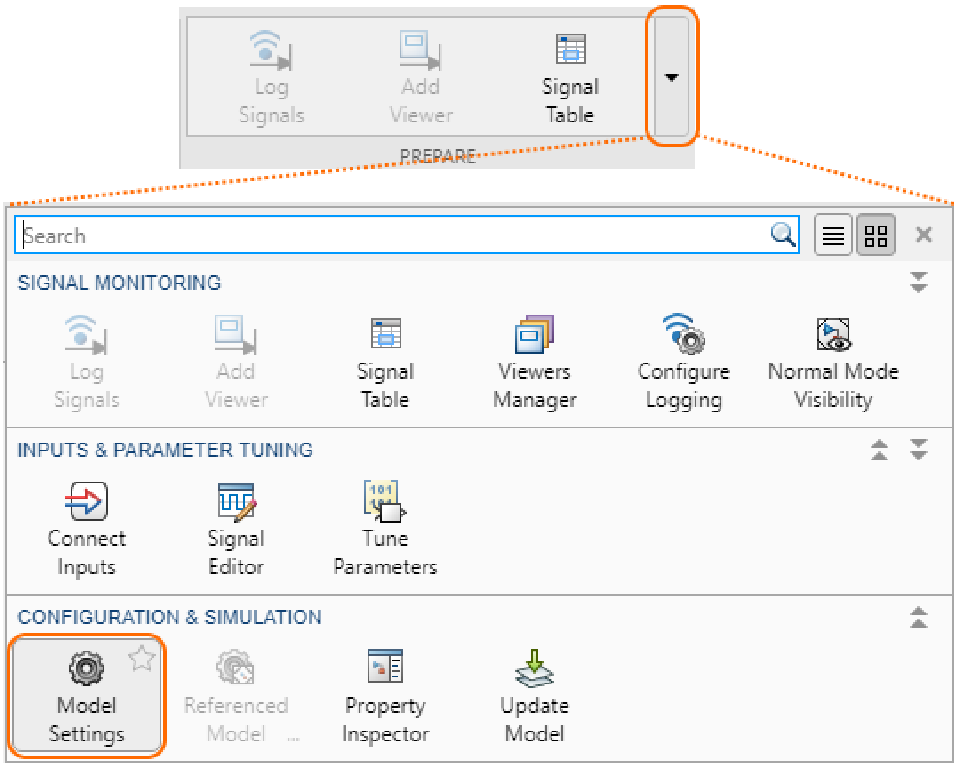
ssc_build
6. In the **Command Window** type **ssc_build** (or **sscbuild** depending on the Matlab version) and press **Enter**. This generates a folder **sscprj** in the **+ ConnectedTanksLibrary** directory as well as a **ConnectedTanksLibrary_lib.slx** file in the directory with the original **ConnectedTanks.slx** file.


7. Go back to the file directory, select the *Hydraulics* package with exported Simulink files and in the **Current Folder** select the *ConnectedTanks.slx*.


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



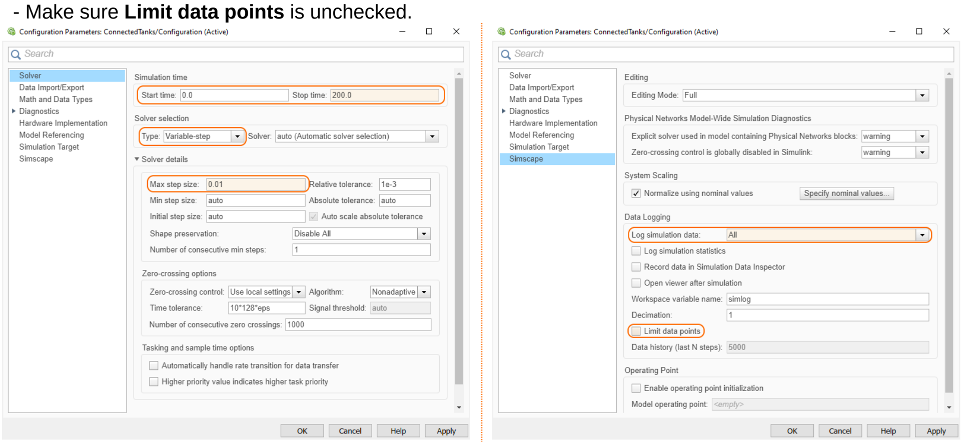
9. On the **SIMULATION** tab, click the arrow on the **PREPARE** area and select the **Model Settings**.



10. In the **Configuration Parameters** dialog, on the left side of the dialog:
a. Select a **Solver** property group and specify the following on the right side of the dialog:
- In the the **Simulation time** group, change **Start time** to **0.0**, **Stop time** to **200** (or any other reasonable number of seconds).
- 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 **0.01**.

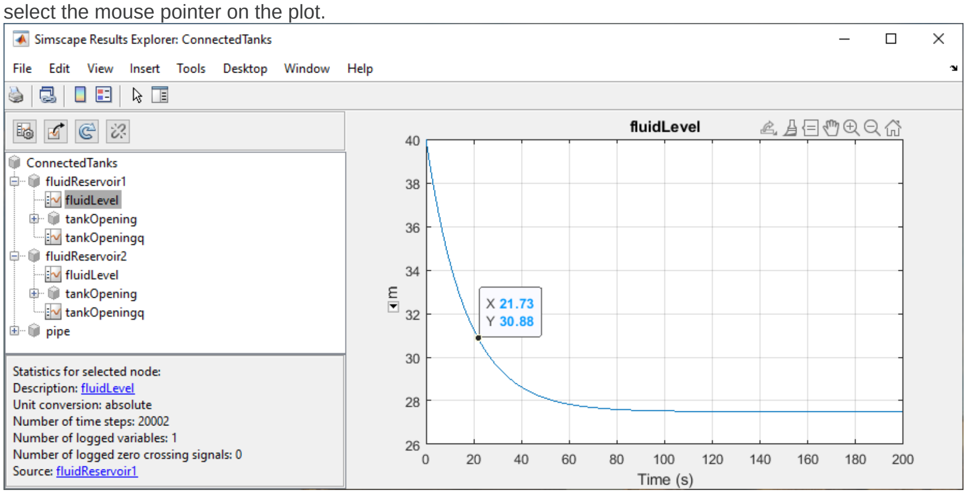
b. Select **Simscape** property group and specify the following on the right side of the dialog:

- In the **Data Logging** group, change **Log simulation data** to **All**.
- Make sure **Limit data points** is unchecked.



11. Press **Apply** > **OK**.
12. In the **ConnectedTanks** Simulink window, click  .
13. On the main toolbar, click 

to open a **Simscape Results Explorer**: **ConnectedTanks** window where you can expand the **ConnectedTanks** blocks to see how their properties simulated through time. For the tanks **fluidReservoir1** and **fluidReservoir2**, the properties **fluidLevel** can be selected to see how the fluid reacted transferred from one tank to the other in the simulation. To see the simulation's specific data points, select the mouse pointer on the plot.



14. To change the default value or initial value parameters of the model, double-click the blocks in the **ConnectedTanks Simulink** model. A **Block Parameters** dialog opens where you can change parameters. Repeat steps from 5 to 12 to run the simulation with (new) block parameters.