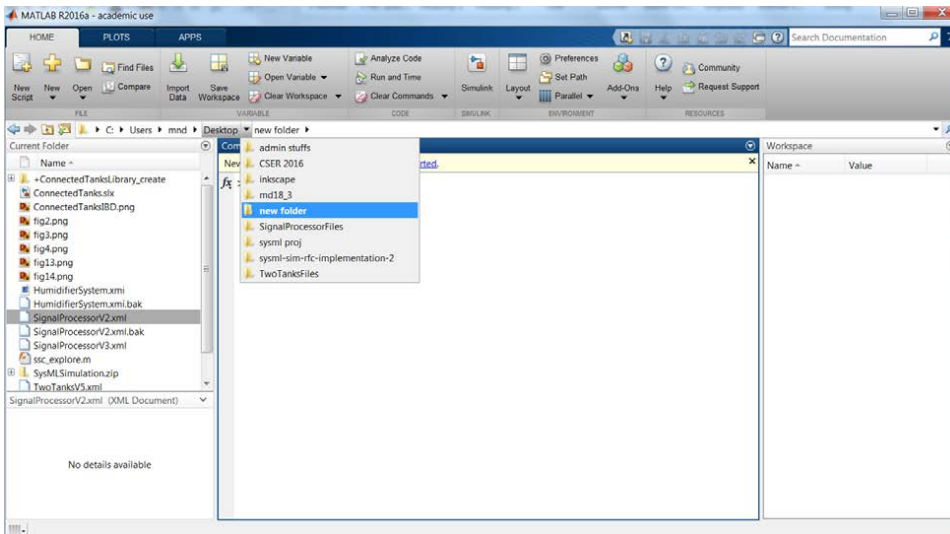


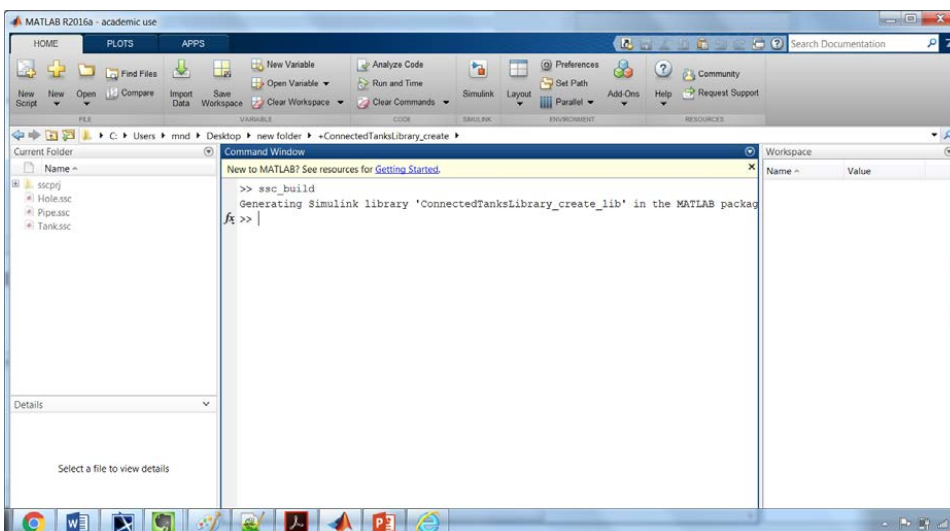
Generating Simulink and Simscape model

To generate a Simulink model directly from the modeling tool

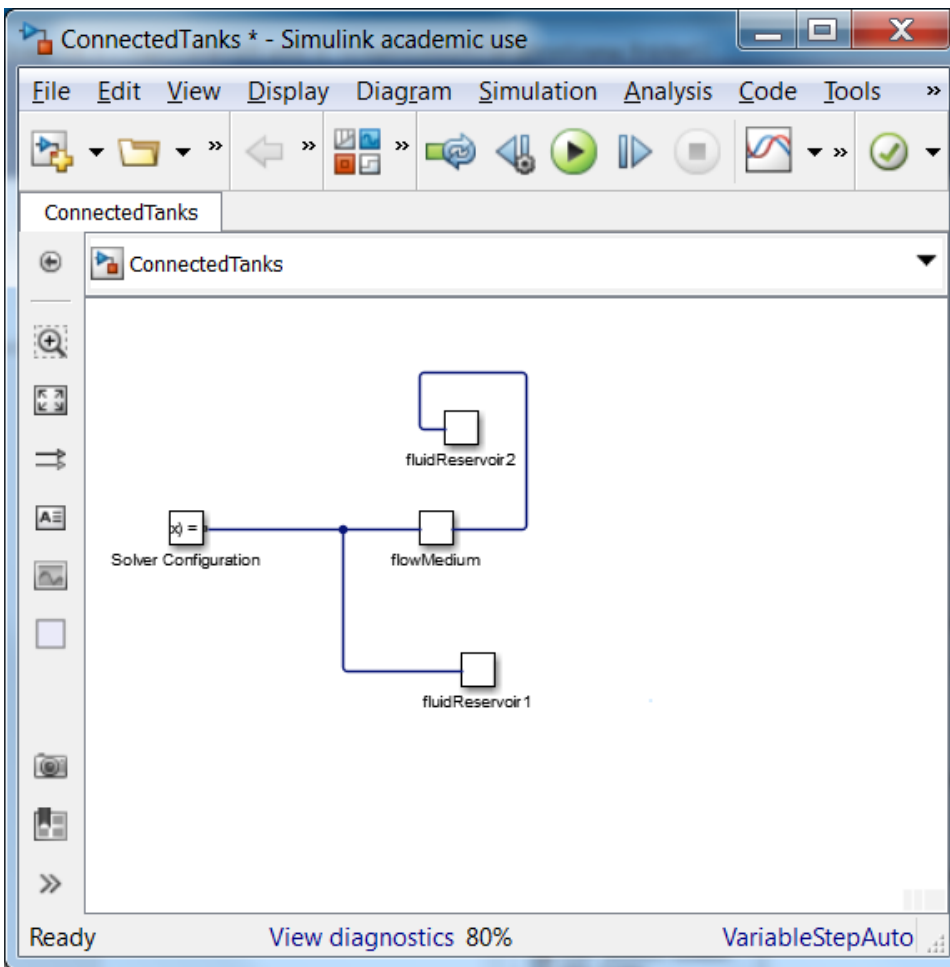
1. Right-click the **Block Data:ConnectedTanks** and select **Tools > Export to Simulink**. Please see [Generating a simulation file](#) for more information.
2. Set the options listed below and click **OK**:
 - **Format:** XML (.sdl)
 - **S-Function or Simscape:** Simscape
 - **Simscape port libraries:** 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.



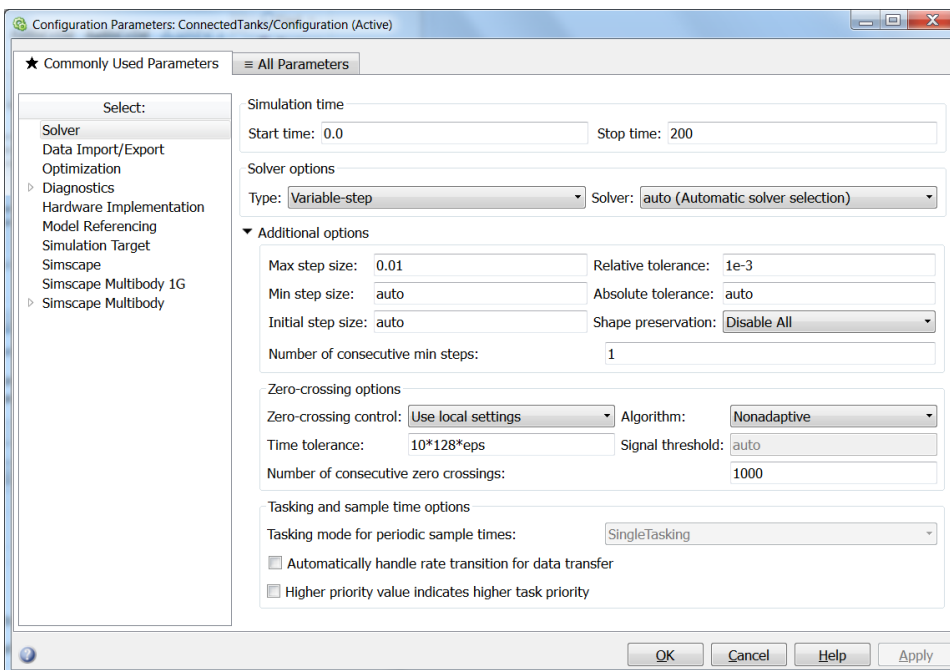
5. In the **Current Folder** panel, double-click on the folder named **+ConnectedTanksLibrary_create** (this sets the current directory to this folder). Type **ssc_build** in the **Command Window** and press Enter. This will generate a folder **sscpj** in the **+ConnectedTanksLibrary_create** directory as well as a **ConnectedTanksLibrary_create_lib.slx** file in the directory with the original **ConnectedTanks.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 set it as the current directory.



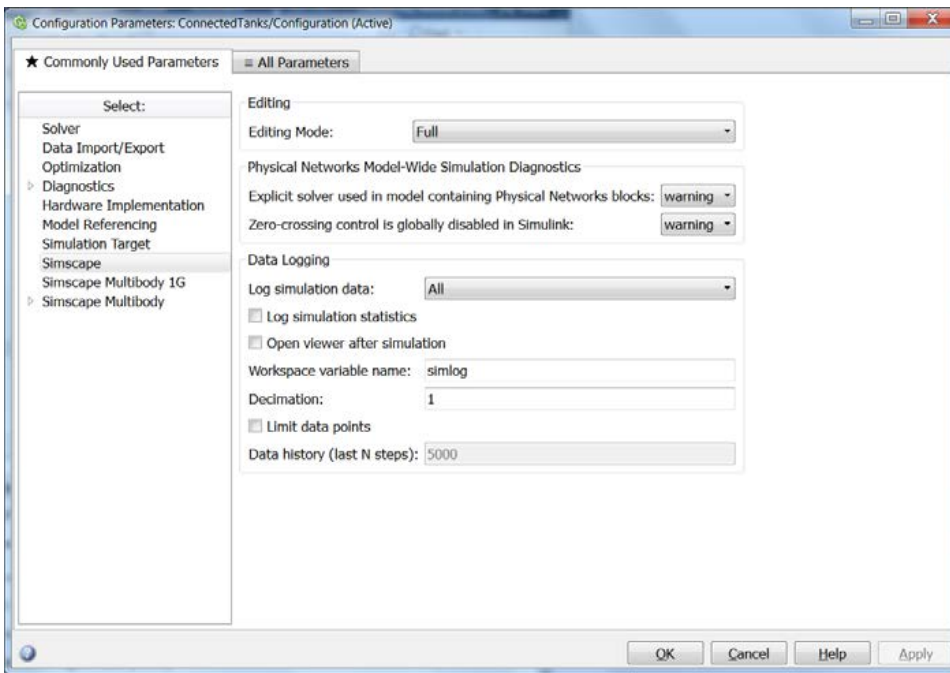
6. In the **Current Folder** panel, double-click on **ConnectedTanks.slx** to open the Simulink/Simscape model. Rearrange the blocks to get a better picture of the blocks and links.



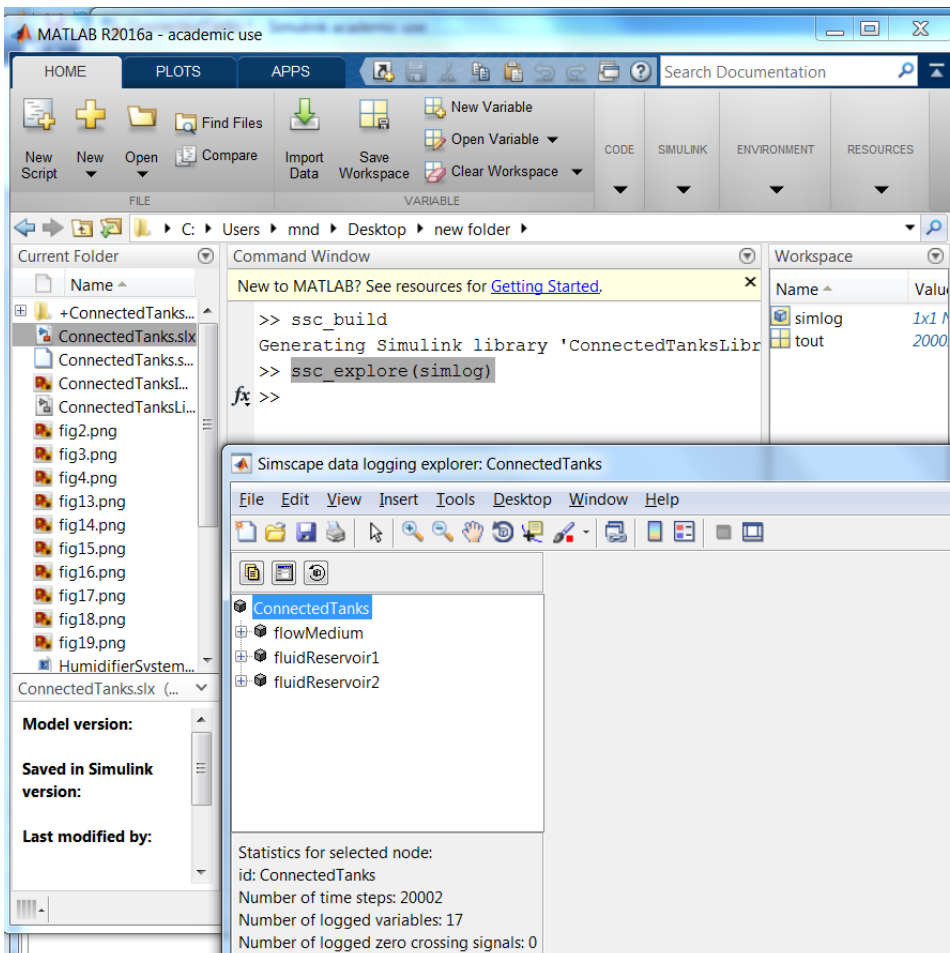
- Go to **Simulation > Model Configuration Settings**. This allows to select the types of solver and the runtime of the simulation. Select a **Start Time** of **0**, and a **Stop Time** of **200** (or any other reasonable number of seconds). Change the **Max step size** to **0.01**. Under **Solver Options**, keep the solver **Type** to **Variable-step** (or any other desirable solver that is suitable). Press **Apply**.



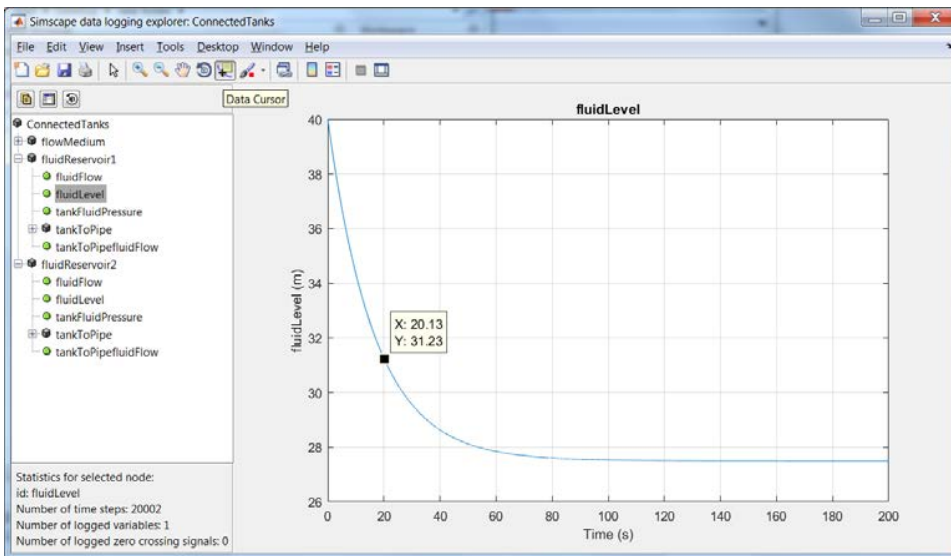
- In Model Configuration Settings, select **Simscape** in the **Commonly Used Parameters** panel. Change **Log simulation data** to **All**. Make sure **Limit data points** is unchecked. Press **Apply**. Press **OK**.



9. In the **ConnectedTanks** Simulink window, go to **Simulation > Run**.
10. Once the model has compiled, go to the Matlab's command window and type/enter **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 **ConnectedTanks.slx** in the **Current Folder** panel). A **Simulink data logging explorer: Connected Tanks** window will open up.



11. In the **Simscape data logging explorer**, the **ConnectedTanks** blocks can be expanded 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, the **Data cursor** icon needs to be selected and points on the plot need to be chosen (by clicking it).



12. To change the default value or initial value parameters of the model, double-click the blocks in the **ConnectedTanks** Simulink model. A window will pop out for block parameters that can be changed. Run through Steps 7 through 10 to run the simulation with (new) block parameters again.

