

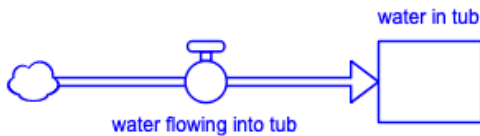



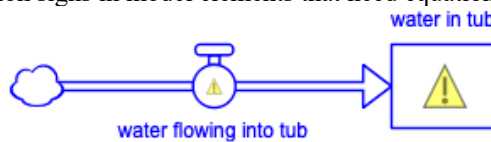
Lesson 1-5-B: Practice building a bathtub model


You can practice building a model in STELLA using a familiar dynamic system: a bathtub filling with water.

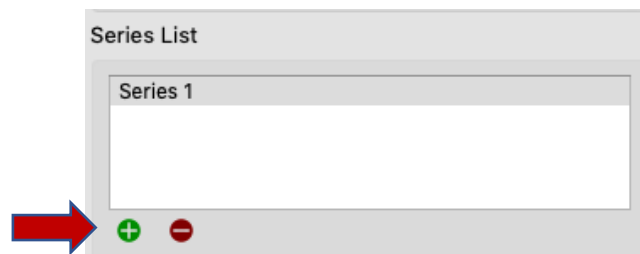
- In STELLA, go to **File > New Model**. You'll see a panel on the right that shows **Model Settings**. Change the **Time Units** to **Minutes** and leave the other settings at their default values.
- Select the **stock**  tool at the top left of the toolbar in STELLA and place the stock in your model by moving your cursor to the white space in your model workbench. Click your mouse to place the stock in the model. Enter the stock name **water in the tub**.
- Select the **flow**  tool, which is next to the **stock** tool in the top toolbar. Place it in your model workspace and then drag and drop the flow into your stock, **water in the tub**. STELLA shows you if the flow is successfully connected to the stock by highlighting the stock. Enter the flow name **water flowing into tub**.



- In order to simulate the model, you next need to enter equations. Switch to **Model View: x^2** . Click on the globe icon  to toggle from **Map View** to **Model View**.
- You will now see yellow caution signs in model elements that need equations:



- Enter an initial value and units for the stock, **water in tub**, by clicking on the stock. The equation and **Units** panel will be on the right side of your model map. In the equation box, click on "Enter initial value here" and type **0** to represent that the tub is empty at the beginning of the simulation. In the **Units** dialog box, type **Gallons**.
- Enter a flow rate by clicking on **water flowing into tub** and typing **10**. The **Units** box should show **Gallons/Minutes**. Water is now set to flow into your tub at a rate of 10 gallons/minute.
- Simulate the model by selecting **Model > Run**.
 - o What happens? Why?
- Add a graph to the model by clicking on the graph icon  in the toolbar and then clicking on the location in your workspace where you would like the graph to be. The panel to the right of your model shows graph settings. Click on the green + sign under the **Series List** and select **water in tub**. The graph will now show the level of water in the tub over time. Simulate the model using **Model > Run**. How much water is in the tub at the end of the run?



(continued on next page)

- What if you have a tub with a drain that doesn't fully close? You could represent that situation by adding an outflow to your model. Select the flow tool from the toolbar, click on **water in tub** and drag and drop the flow from the stock to the right. Enter the outflow name, **water draining**. In **Model View**, set the outflow rate to 2 gallons/minute by typing **2** in the equation box for **water draining**. Simulate the model using **Model > Run**. How is this model run different from the prior one? What is the level of water in the tub at the end of the run now?

