Example: Open channel modeling

Arash Massoudieh

October 10, 2021

In this example we will show how to model transient water flow in a prismatic trapezoidal open channel using *OpenHydroQual*. We will consider a channel with a length of $120m$ that will be discretized into six segments. The profile and the cross section of the channel is shown in figure 1. So we will build the channel as a connected array of six blocks each representing a channel segment with a length of $20m$. We will assume the Manning's roughness coefficient of the channel to be 0.011. As for the initial condition, we assume that there is 0.1 m of water in the channel. Initially, we will assume the channel to have a fixed-head and constant upstream and downstream boundary condition. In the next step, we will change the upstream boundary condition to a fixed-head but time-varying boundary condition and in the third stem we will replace the upstream boundary condition to a flow boundary.

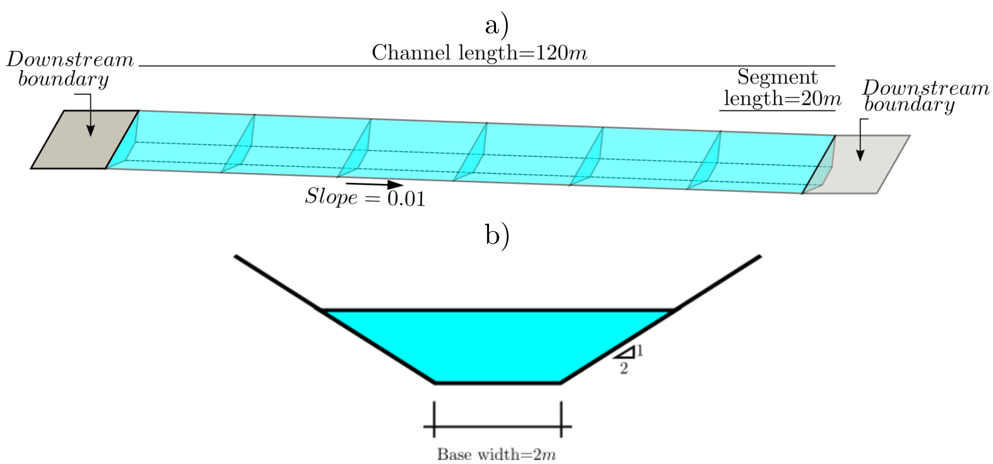


Figure 1: Model representation and cross-section of the channel

Please note that the default "Open Channel" plugin in *OpenHydroQual* uses the Diffusive Wave model with Manning's equation for modeling flow in channels. In other words, the flow velocity is calculated as:

$$V = \frac{S_f^{0.5}}{n} R^{\frac{2}{3}} \tag{1}$$

where n is the Manning's roughness coefficient, S_f is the hydraulic gradient which for two connected blocks is approximated as:

$$S_f = \frac{h_1 - h_2}{L} \tag{2}$$

where $h_1 = z_1 + d_1$ and $h_2 = z_2 + d_2$ are the hydraulic heads in the connected blocks. R is the hydraulic radius which for a trapezoidal cross-section is calculated as:

$$R = \frac{(W + md)d}{W + 2\sqrt{1 + m^2}d} \tag{3}$$

where W is the base width of the channel, d is the water depth, and m is the bank slope. The water depth in each channel segment is related to the water storage in the segment via:

$$d = \frac{-W + \sqrt{W^2 + \frac{4mS}{L}}}{2m} \tag{4}$$

where S is the total storage in the channel segment.

All the governing equations can be found in the file name "open_channel.json" in the "resources" sub-folder of where the *OpenHydroQual* is installed.

Follow the following steps to create your model.

1 Step 1: A simple system consisting of two tanks and a pipe with gravity flow

1. Start *OpenHydroQual*.

2. **Add the Open Channel plugin:** From **File→Preferences→Add Plugin**, add the **Open Channel**  plugin to your model.

3. **Add a single channel segment to your model:** Add a Channel Segment by clicking on the channel segment button  from the components toolbox.

4. **Assign properties to the open channel block:** We will first assign the needed properties to the single open-channel block that was added to the model and then will make copies of it to create the other five segments. We will set the reference elevation to be the bottom of this channel segment and then we will lower the value for the copied channels to enforce the downward slope. Select the open channel block that was just added to your model and then assign the following properties:

    - *Side Slope:* 2

- *Manning's roughness coefficient:* 0.011
- *Base Width:* 2 [m]
- *Water depth:* 0.1 [m]
- *Segment length* 20 [m]
- *Bottom Elevation:* 0 [m]

5. **Make five copies of the channel block:** Right-click on the channel block "Trapezoidal Channel Segment (1)" and click on **Copy** from the drop-down menu. Then click somewhere else on your screen and select **Paste** from the drop-down menu. A new block named "Trapezoidal Channel Segment (2)" will be added to your model. This block will have identical properties to your original channel. Repeat this for four more times to get all your six channel segments. After you are done, your GUI should look like figure 2.

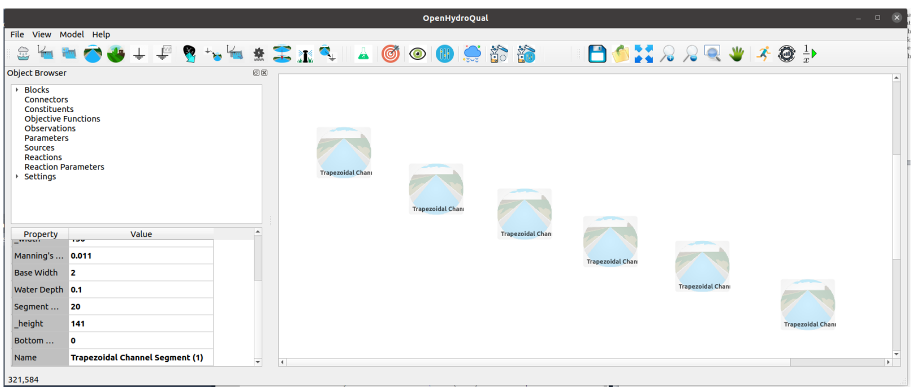


Figure 2: Screenshot of the GUI after creating the six open channel segments

6. **Change the bottom elevations of the channels:** Since the distance between each channel is 20m and the longitudinal slope of the channel is 0.01, each channel segment should have a bottom elevation of 0.2m below to preceding channel block. So select channels 2,3,4,5, and 6 and respectively assign bottom elevations of -0.2, -0.4, -0.6, -0.8, and -1.0 to them.

7. **Connect the channel segments to each other:** Use the "Link from a channel segment to another"  from the component toolbox and connect each channel to the adjacent one. It is a good practice

to start from upstream to downstream. After you are done, your GUI should look like figure 3.

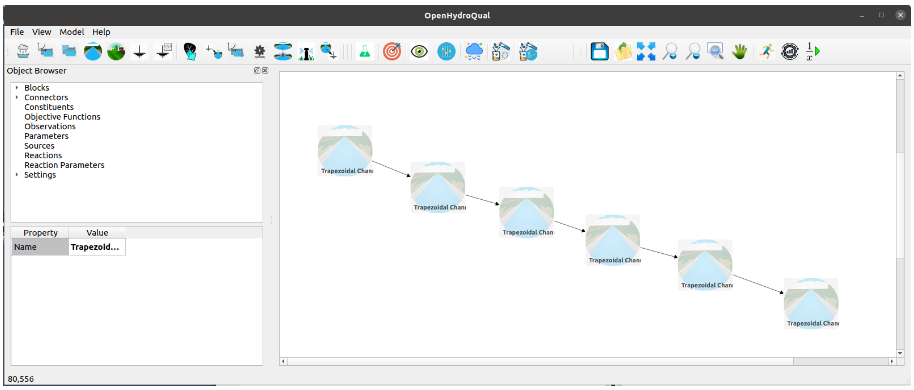


Figure 3: Screenshot of the GUI after connecting the segments

8. **Save your model**

9. **Run the model** . Run the model and then evaluate the results. Look at the flow through the links and also water depth and head in each channel segment. As you can see water flows downstream to the lowest segment and then stops. This is because there is no outlet from the most downstream segment that allows water to flow out of the system. We will fix this problem in the next step.

10. **Add the downstream boundary condition:** We will use a fixed-head block to represent the downstream boundary of the channel system. We will assume the fixed-head value of the downstream boundary to be $-1m$. To add the downstream boundary, add a fixed-head block to the model and drag it to the right side of the most downstream channel segment in your model. Select the newly added *fixed-head* block and assign a value of -1 to the Head property. You can also rename the fixed-head block to "DS_boundary".

11. **Connect the most downstream block to the fixed-head block:** Select the "Link from a channel segment to fixed head boundary" from the components toolbar and then connect the "Trapezoidal Channel Segment (6)" to the fixed-head block. Your screen should look like figure 4.

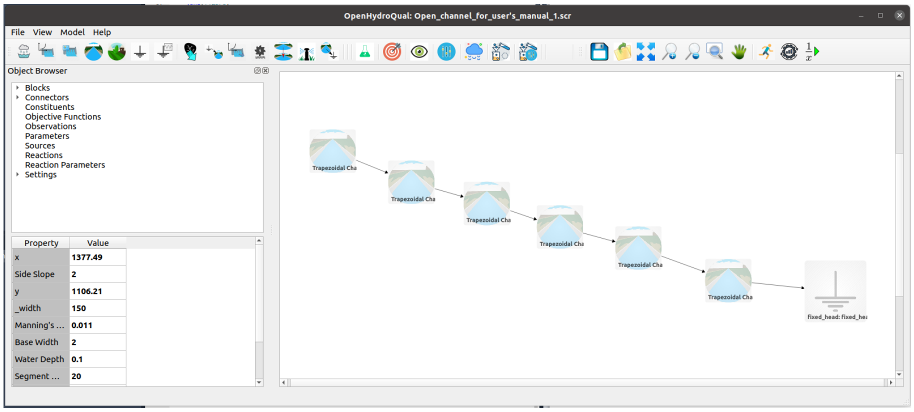
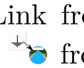


Figure 4: Screenshot of the GUI after adding the downstream boundary

12. **Run the model:** You may run the model now. As you see, with this version of the model, all the water in the channel segments eventually drain to the downstream boundary block. In the next step we will implement an upstream boundary that will allow water to flow into our channel system.

13. **Add the upstream boundary:** Add another fixed-head block  and drag it to the left of your most upstream channel segment block. At this stage we are going to assume the hydraulic head at the upstream boundary is fixed and is equal to $0.2m$. So select the newly added fixed-head block and assign a value of 0.2 to the property "Head". You may also change the name of this fixed-head block to "US_boundary".

14. **Connect the upstream boundary to the upstream channel block:** Use the "Link from a fixed head boundary to a channel segment" component  from the components toolbar and connect the *US_boundary* block to *Trapezoidal Channel Segment (1)*. Your GUI should look like figure 5.

15. **Run the model:** Run the model and look at the flows in the links and water depths and heads in the blocks. You can also check the values of hydraulic radius, wetted perimeters and cross-sectional areas.

16. **Note:** If you run the model over a longer period of time (for example 10 days instead of one day) you will see that the flow will stop at one

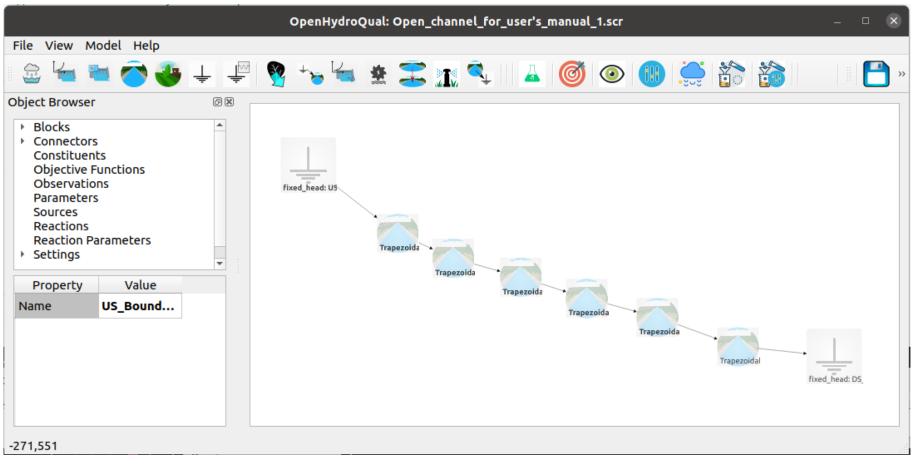


Figure 5: Screenshot of the GUI after adding the upstream boundary

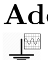
point. The reason for this is that the initial storage of water in the upstream boundary block is limited and it runs out at one point. To avoid this, you can increase the storage in *US_boundary* to a value much larger than what you currently have.

2 Step 2: Making the upstream boundary temporally variable

In this step, we will change the upstream boundary condition to temporally variable head. We will use a half-sinuousoidal curve to represent the temporal variation of the hydraulic head at the upstream boundary.

Follow the following steps:

1. **Delete the upstream boundary block - "US_boundary"**: Right-click on the upstream boundary block and then select "Delete" from the drop-down menu. This will also delete the existing link from the fixed-head block to the upstream channel segment.

2. **Add a time variable fixed-head boundary:** Add a *time_variable_fixed_head*  boundary from the components toolbar and drag it to the left of the upstream side of the channel. **Add the data representing time-varying head:** Download the file called "half_sin_wave.csv" from `https://openhydroqual.com/wp-content/Data/half_sin_wave.csv` and save it somewhere on your hard-drive. Open the file using a text

editor (like Notepad) and see the files structure. The file contains two columns separated by commas. The first column contains time and the second column is the values of time series at those times. The intervals between time points do not need to be equal but one should make sure the values in the time column are increasing. Now select the time variable fixed-head block you just added and then click on the box against the **Head** property in the property window and then choose the file you just downloaded. To make sure the file is loaded properly right-click on the file name in the property box and click on *Plot* from the drop-down menu. A graph showing the temporal change of head should appear like figure 6.

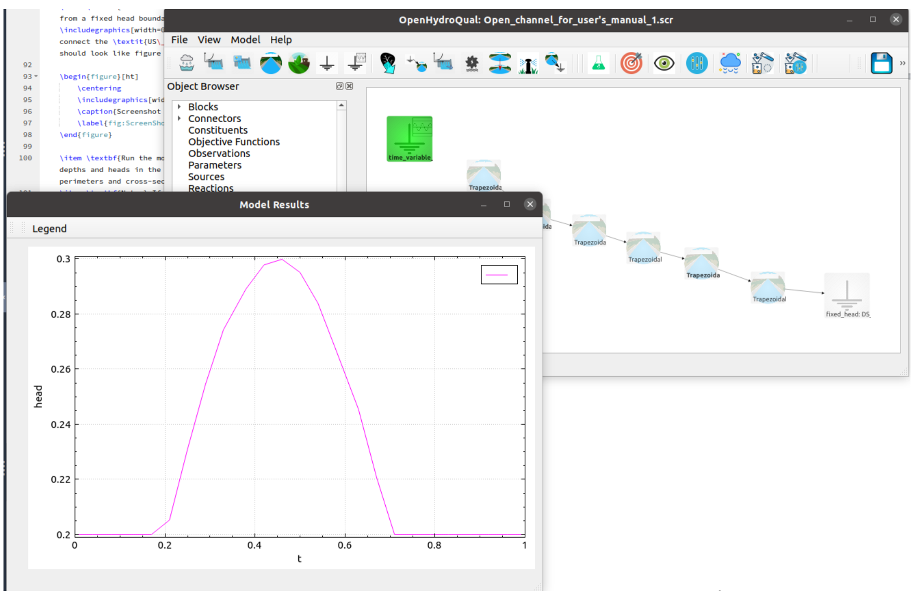


Figure 6: Time series data for head in the upstream boundary

3. **Connect the upstream boundary to the upstream channel block:** Use the "Link from a fixed head boundary to a channel segment" component  from the components toolbar and connect the *US_boundary* block to *Trapezoidal Channel Segment (1)*.

4. **Run the model:** Run the model and look at the flows in the links and water depths and heads in the blocks. You can also check the values of hydraulic radius, wetted perimeters and cross-sectional areas and see how each changes with time.

5. **Change the time resolution of the model:** As you can see the graphs appear a bit rough and inaccurate. This is due to the fact that our computational time-step is a bit too large. To reduce the time-step at which the results are stored from Object Browser window select **Settings→Solver Settings** and the change the values of *Initial time-step* to 0.0001. Now run the model again and check the results.

6. **Note:** Please note that the total channel length is too small compared to the wave speed and that is why the delay in arrival of the wave to downstream is not fully noticeable. Try increasing the length of each channel segment to 200m to see how that affects the wave behaviour. Please note that you copy and paste graphs from one graph window to another one to compare them (figure 7).

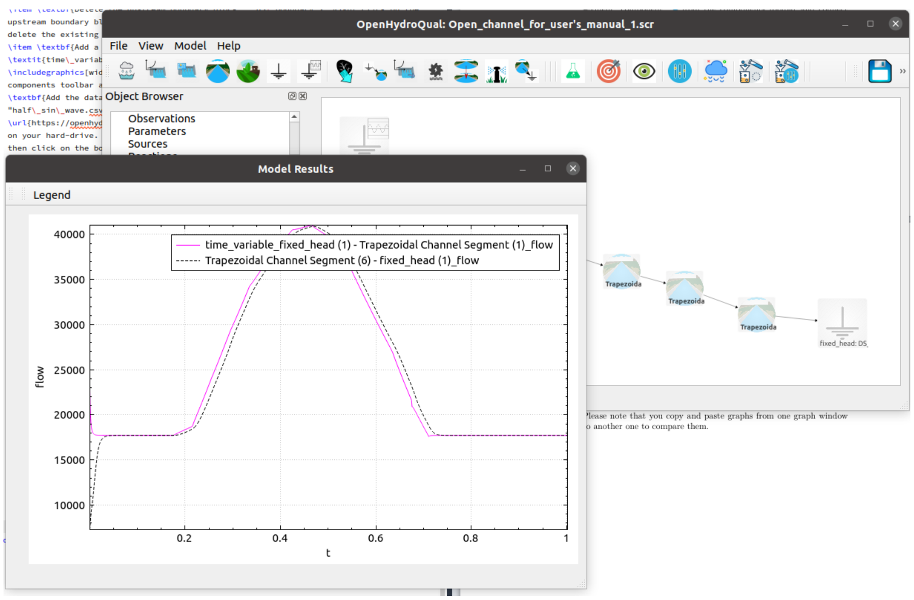


Figure 7: Screenshot of flow rate at upstream and downstream of the channel for the case with a channel length of 1200 m

3 Step 3: Replacing the head boundary condition with a flow boundary condition at the upstream boundary

In natural systems, due to the uncertainty associated with the channel geometry and roughness, using a flow boundary condition results in a more accurate predictions compared to imposing a fixed head boundary. In this step, we will replace the head boundary condition with a flow boundary condition. In fact imposing a flow boundary condition is quite simple and involves introducing an inflow into the upstream block via a time-series data file.

Do the following steps:

1. **Delete the upstream boundary fixed head block:** This will also delete the link connecting the block to the upstream stream channel block.

2. **Add the inflow:** Download the example inflow file from `https://openhydroqual.com/wp-content/Data/half_sin_wave_flow.csv` and save it somewhere on your hard-drive. You can open the file using any text editor (such as Notepad) and look at the contents of the file.

3. **Apply the inflow to the upstream channel segment:** Select the upstream channel segment "Trapezoidal Channel Segment (1)" and from the properties window click on the box in front of inflow time series and then select the file you just downloaded.

4. **Save and then run the model:** Run the model and then inspect the results.