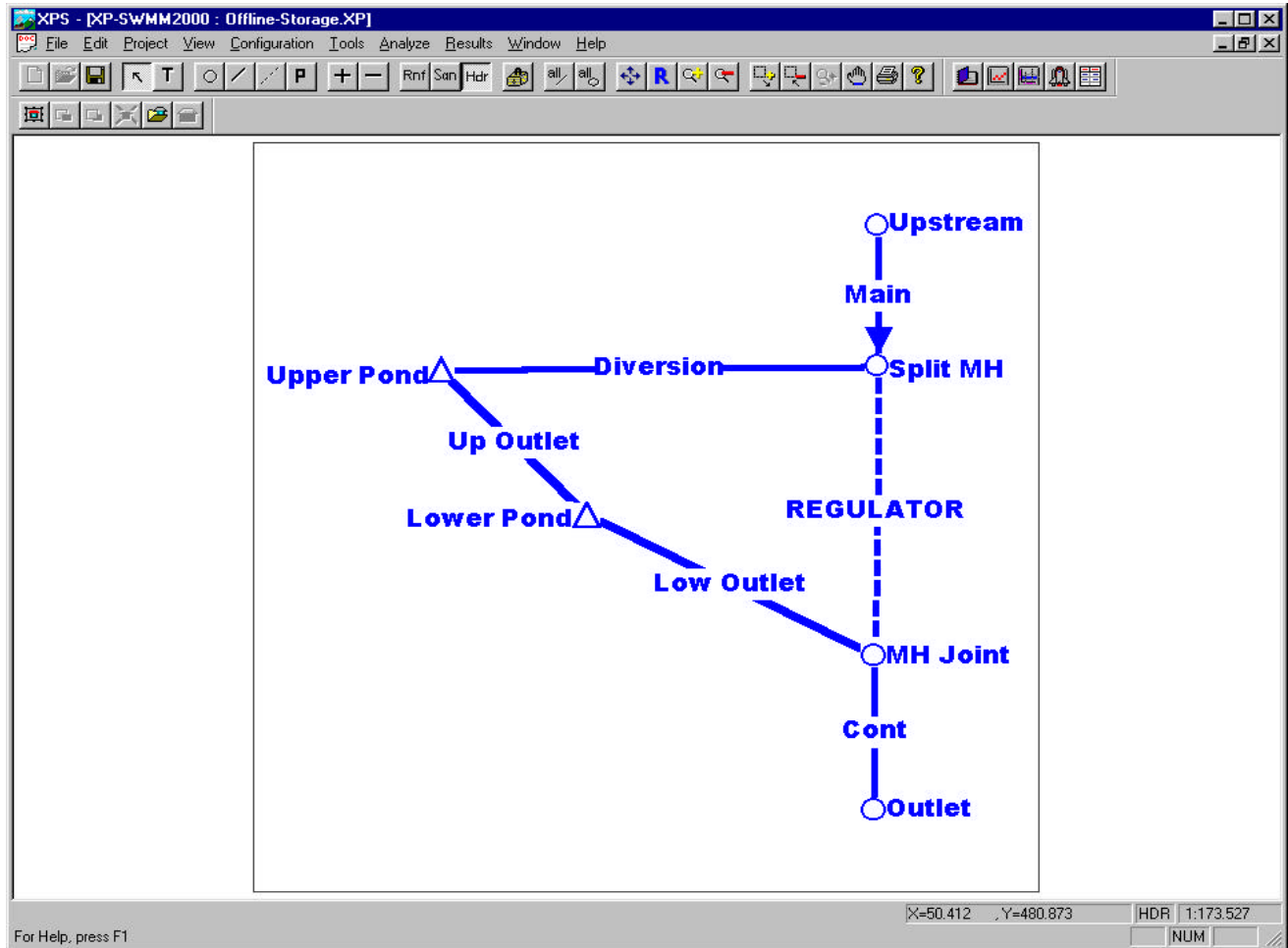


Module 55: Offline Storage with Real Time Control Regulator

Synopsis

In this module we will look at a Runoff – Hydraulics (EXTRAN) surface pond system with a regulator that will control the flow in a conduit based on the depth of water at node “upper pond”. This model depicts a method of controlling water levels in sensitive offline storage facilities such as wetlands.

Open up the file **Offline-Storage.XP** in your work directory. There are two subcatchments in the Runoff layer at nodes “Upstream” and “Upper Pond”. The Runoff and Hydraulics layers are connected together with an interface file.



The screenshot shows the "Runoff Node: Node Upper Pond" dialog box. It contains a table for sub-catchments and several checkboxes.

Sub-Catchments	1	2	3	4	5
Area	30				
Imp. (%)	10.				
vWidth	1000.				
Slope	.01				

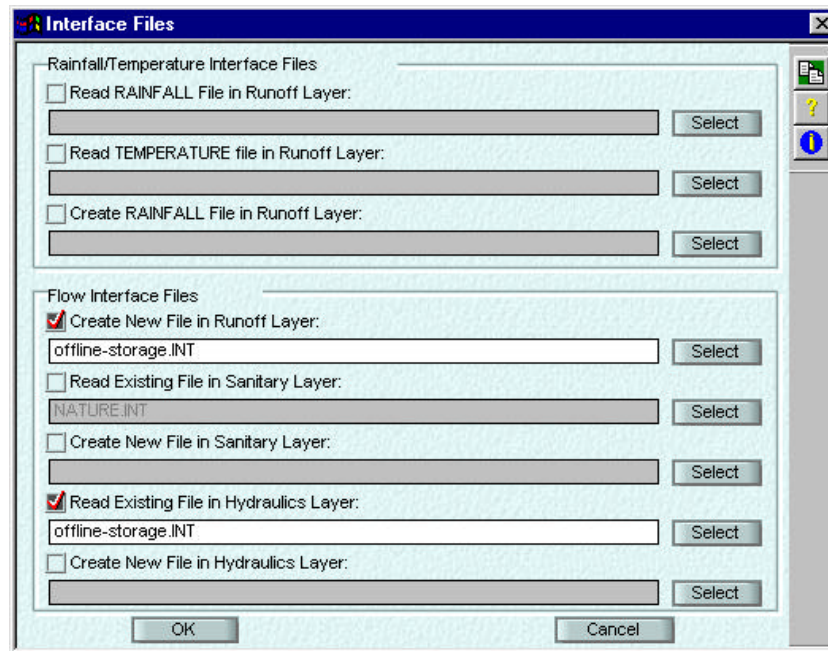
Below the table, there are two checked checkboxes:

- ☒ Print Flows and Concentrations
- ☒ Save Results for Review

At the bottom, there are three buttons: "OK", "Cancel", and "Gauged Data".

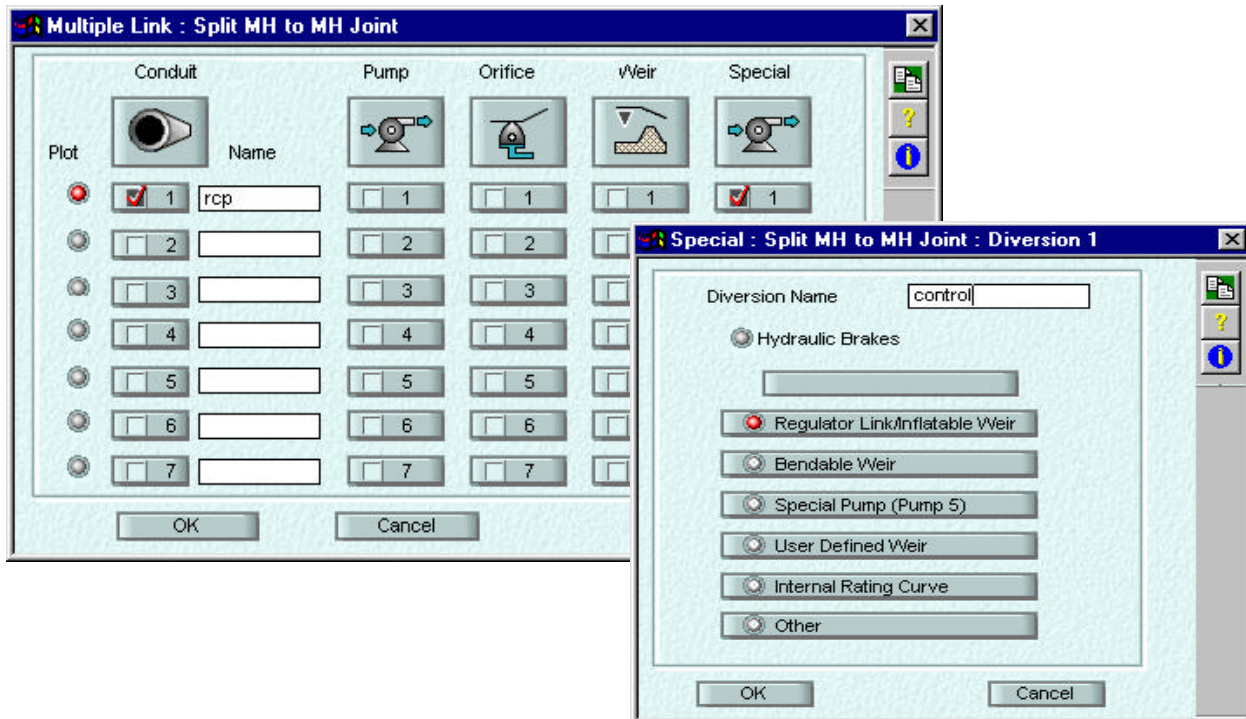
The two subcatchments have four physical parameters, use the EPA Runoff Hydrology, and are connected to two global databases for rainfall and infiltration. This data can be viewed by double-clicking on the nodes “Upstream” and “Upper Pond” and on the selected subcatchment buttons.

The interface file from the Runoff layer is named as the “Create New File in Runoff Layer” section of the Interface File dialog. This dialog is accessed under the Configuration menu in the interface. The same named interface file should be entered in the “Read Existing File in Hydraulics Layer”. Two locations will be saved to the Runoff interface file; these are nodes “Upper Pond” and “Upstream”.



The two flow time series will be read by Hydraulics and routed through the open and closed conduit network shown in the network. Click on the “Hyd” icon to move to the Hydraulics layer of XP-SWMM2000. The low flow will be diverted down the conduit diversion to the Upper pond and Lower pond storage nodes. The regulator will control the flow down the regulator link dependent on the depth of water in node.

A regulator is defined in a multi-link under the Special list of conduits. The regulator in a row modifies the behavior of a conduit in that row. Double-click on the Special conduit in the first row to look at the regulator.



The regulator data is set up in the following way: A table of depth versus flow multiplier is entered in the data fields. A multiplier of zero means that no flow will go through conduit “rcp”. In the above table until the depth of water in node “Upper Pond” exceeds 8 feet there will be no flow in conduit “rcp”.

Depth	Multiplier	Depth	Multiplier
0.	0.		
8.	0.		
8.01	1.		
12.	1.		

Node Name: Upper Pond

Buttons: OK, Cancel, Sort, Goto, Graph, Row 1

Also note that “Upper Pond” is initialized at a depth of 7 feet. This leaves a maximum 1 foot of bounce (difference between control and the maximum water surface elevation) before the regulator will open.

Spill Crest: 753.

Inlet Capacity: ☐

Initial Depth: 7.

Inflow Data:

Constant Inflow: Inflow: 0.0

Pollutant Loads:

Time Series Inflow:

☐ User Inflow

☐ Gauged Inflow

☐ Dry Weather

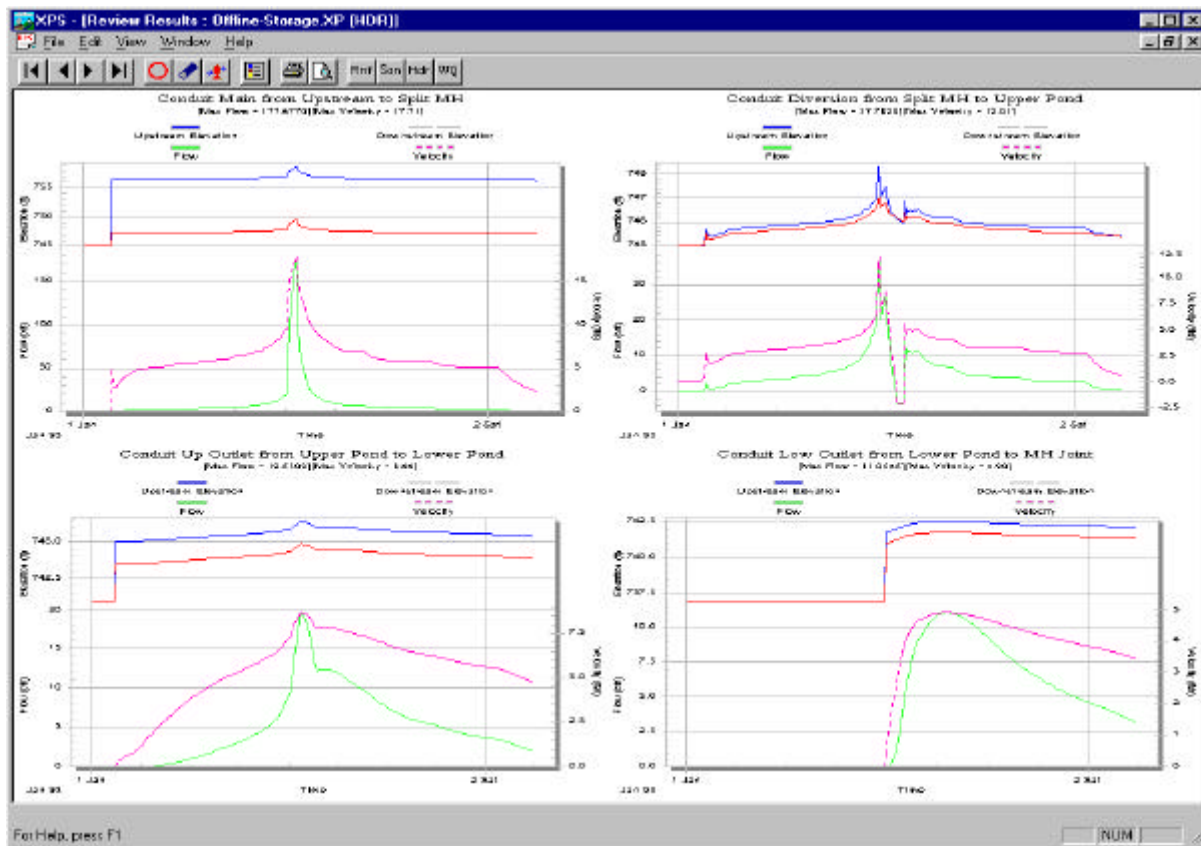
Ponding: ☒ None ☐ Allowed ☐ Sealed

☒ Storage ☐ Outfall ☐ Detail Printout ☐ Save Overflow Results

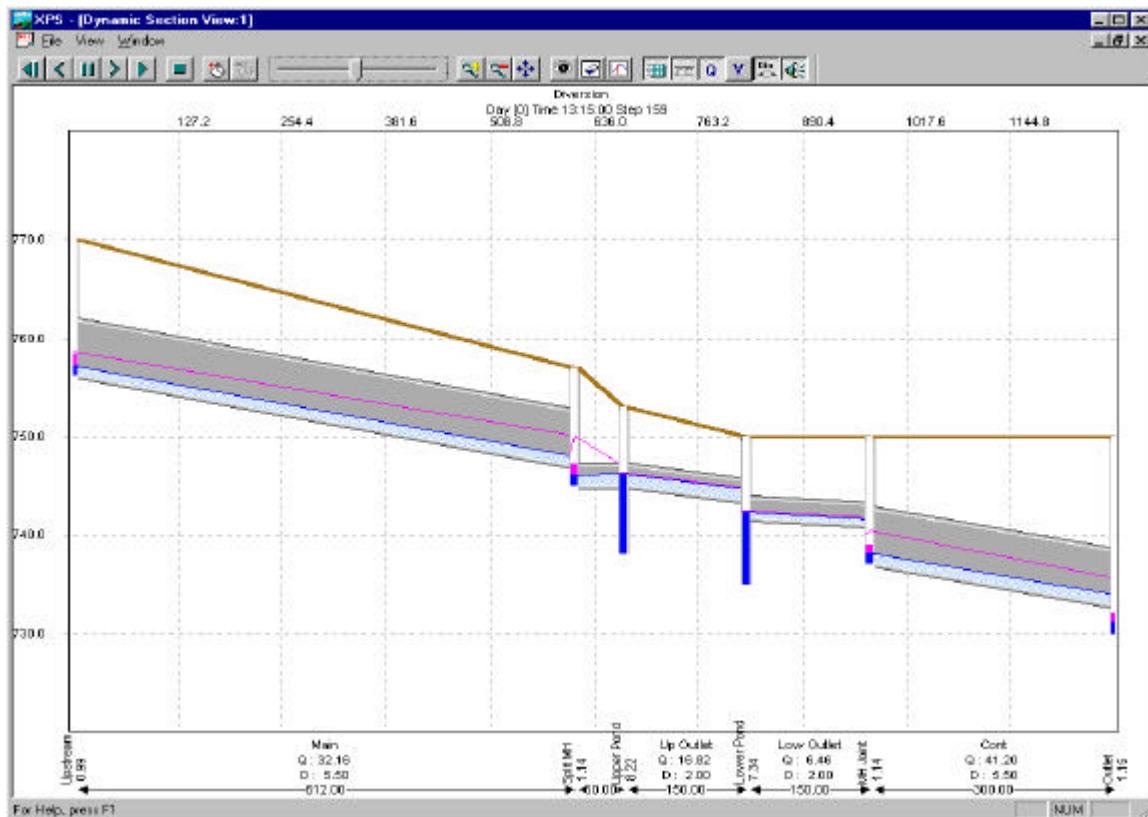
☐ Plot Water Levels

Buttons: OK, Cancel, Gauged Data

Solve the model and look at the results using Review Results. You can see the influence of the regulator and the delayed hydrograph through the two ponds. Notice the time that the water surface elevation in “Upper Pond” is above 746 ft. This is the same time frame that there is flow in “rcp” as we would expect from our *rule* in the regulator.



Highlight the “Upstream” through “Outlet” set of conduits. To accomplish this you could highlight the whole model and then using the shift key and the left mouse button deselect link “regulator”. The long section plot can only show straight runs of links. Next use the long section icon and look at the long section plot. You can see the dead storage in the two ponds. Experiment and try to zoom in on the center section of the network.



An important table in the output file is Table E5a, which lists the internal time step for each conduit during the simulation. The conduit in the above table with the most control over the internal time step is link “Up Outlet”.

```

*=====
      Table E5a - Conduit Explicit Condition Summary
      Courant   =      Conduit Length
      Time step = -----
                  Velocity + sqrt(g*depth)

      Conduit Implicit Condition Summary
      Courant   =      Conduit Length
      Time step = -----
                  Velocity
*=====
      The 3rd column is the Explicit time step times the
      minimum courant time step factor

      Minimum Conduit Time Step in seconds in the 4th column
      in the list. Maximum possible is 10 * maximum time step

      The 5th column is the maximum change at any time step
      during the simulation. The 6th column is the wobble
      value which is an indicator of the flow stability.

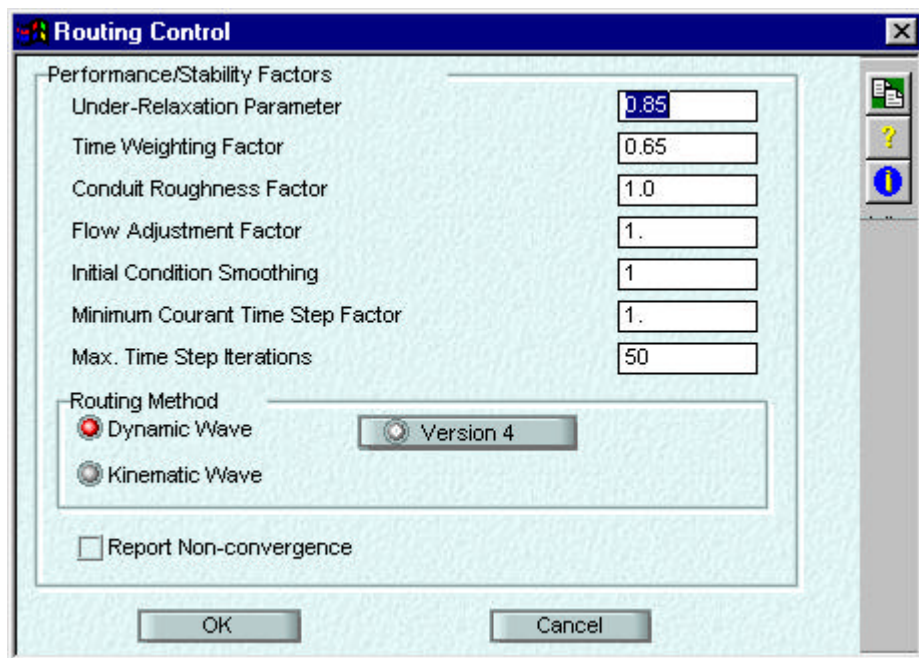
      You should use this section to find those conduits that
      are slowing your model down. Use modify conduits to
      alter the length of the slow conduits to make your
      simulation faster, or change the conduit name to
      "CHME?????" where ????? are any characters, this will
      lengthen the conduit based on the model time step,
      not the value listed in modify conduits.
*=====

```

Conduit	Time(exp)	Expl*Cmin	Time(imp)	Time(min)	Max Qchange	Wobble	Type of Soln
Main	23.07	23.07	34.56	110.0	-4.965	0.803	Normal Soln
Diversion	5.53	5.53	9.39	645.0	-1.775	154.067	Normal Soln
Up Outlet	10.52	10.52	18.53	865.0	0.066	1.529	Normal Soln
Low Outlet	32.91	32.91	66.49	0.0	0.013	0.508	Normal Soln
Cont	11.98	11.98	18.16	0.0	35.895	1.494	Normal Soln
rcp	16.99	16.99	28.53	0.0	-9.397	1.504	Normal Soln
control	3000.00	3000.00	3000.00	0.0	0.000	0.000	Normal Soln

The conduit with the smallest time step limitation was..Up Outlet
The conduit with the largest wobble was.....Diversion
The conduit with the largest flow change in any
consecutive time step.....Cont

The internal time step is controlled by the minimum courant time step factor. This model will run faster if the courant time step factor is 2 or 3. The selection of the time step factor should be based in the overall continuity error and the model efficiency. A low efficiency and low continuity error means that the model will run faster if the courant time step factor is increased.



The image shows a software dialog box titled "Routing Control". It contains several input fields for performance and stability factors, a section for selecting a routing method, and a checkbox for reporting non-convergence. The "Under-Relaxation Parameter" is set to 0.85, "Time Weighting Factor" to 0.65, "Conduit Roughness Factor" to 1.0, "Flow Adjustment Factor" to 1., "Initial Condition Smoothing" to 1, "Minimum Courant Time Step Factor" to 1., and "Max. Time Step Iterations" to 50. Under "Routing Method", "Dynamic Wave" is selected with a radio button, and "Version 4" is chosen from a dropdown menu. "Kinematic Wave" is also an option but is not selected. The "Report Non-convergence" checkbox is unchecked. At the bottom are "OK" and "Cancel" buttons. On the right side of the dialog, there are icons for help (question mark) and information (i).

Performance/Stability Factors	
Under-Relaxation Parameter	0.85
Time Weighting Factor	0.65
Conduit Roughness Factor	1.0
Flow Adjustment Factor	1.
Initial Condition Smoothing	1
Minimum Courant Time Step Factor	1.
Max. Time Step Iterations	50

Routing Method

- ☒ Dynamic Wave
 - Version 4
- ☐ Kinematic Wave

☐ Report Non-convergence

OK Cancel