

1. Windows-based E1DISP Model

Figure 1 shows the windows-based E1DISP model that solves the advection-dispersion equation (1) using the finite segment method.

$$\frac{ds}{dt} = -u \frac{\partial s}{\partial x} + E \frac{\partial^2 s}{\partial x^2} - k s + W \quad (1)$$

The FORTRAN coding of the model is given in Koh (2004). The step-by-step instructions on how to run this model are presented below.

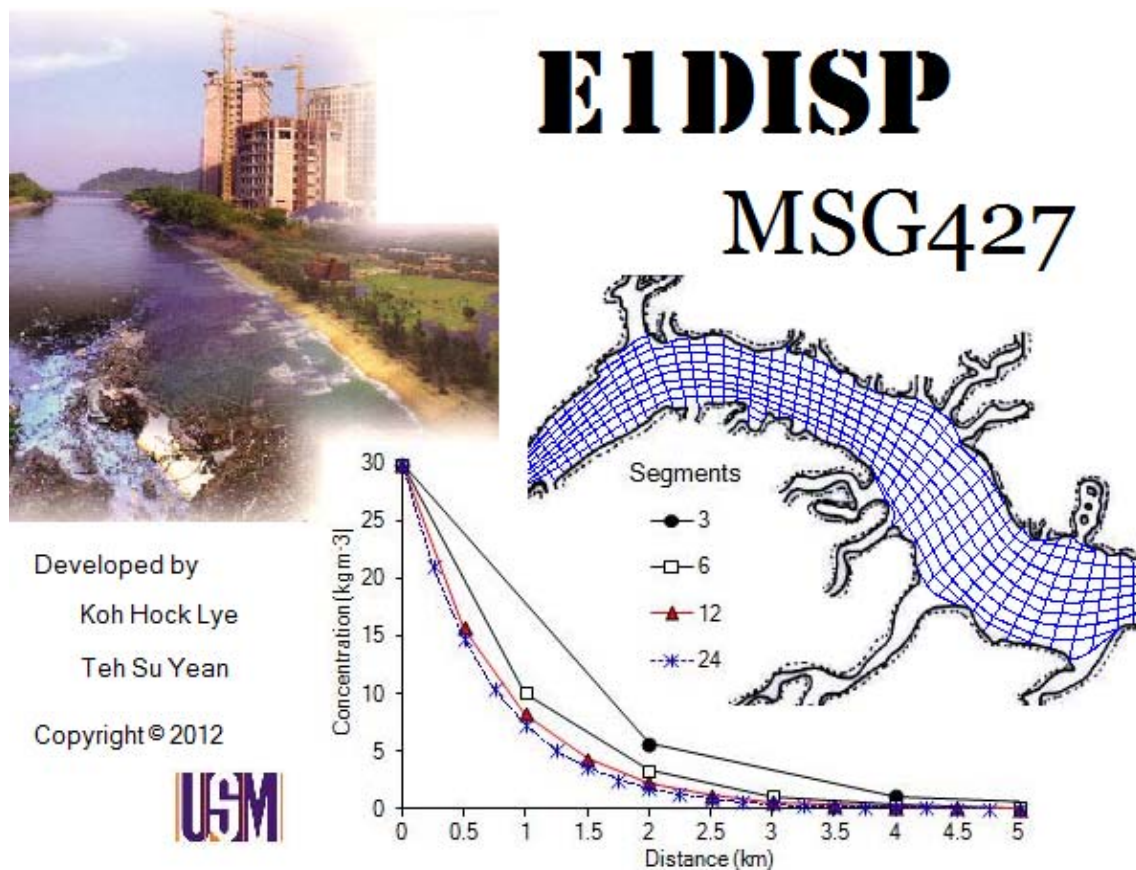


Figure 1: Windows-based E1DISP model

2. An Illustrative Example (Example 5.1, Koh 2004)

The river segmentation as shown in Figure 2 is used as an illustrative example. Figure 2 will be displayed when the link [Figure](#) in E1DISP input window (see Figure 3) is clicked.

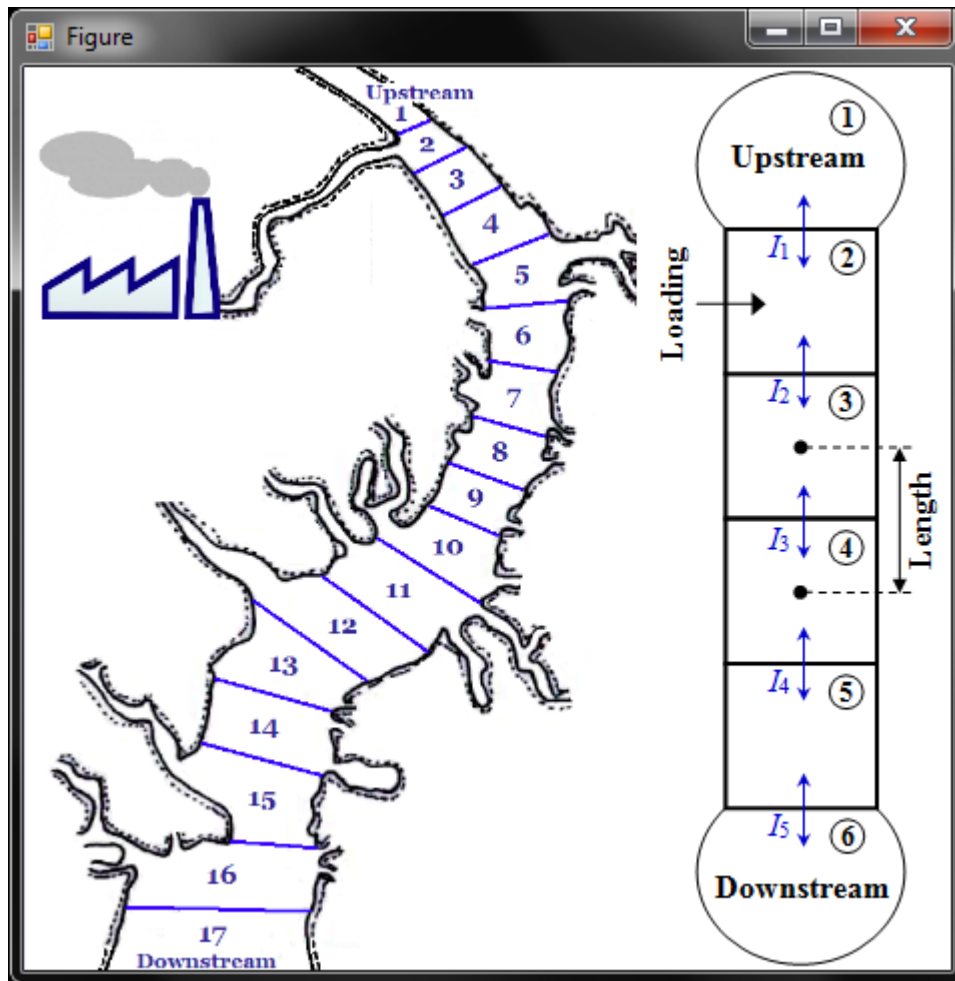


Figure 2: An illustrative river example

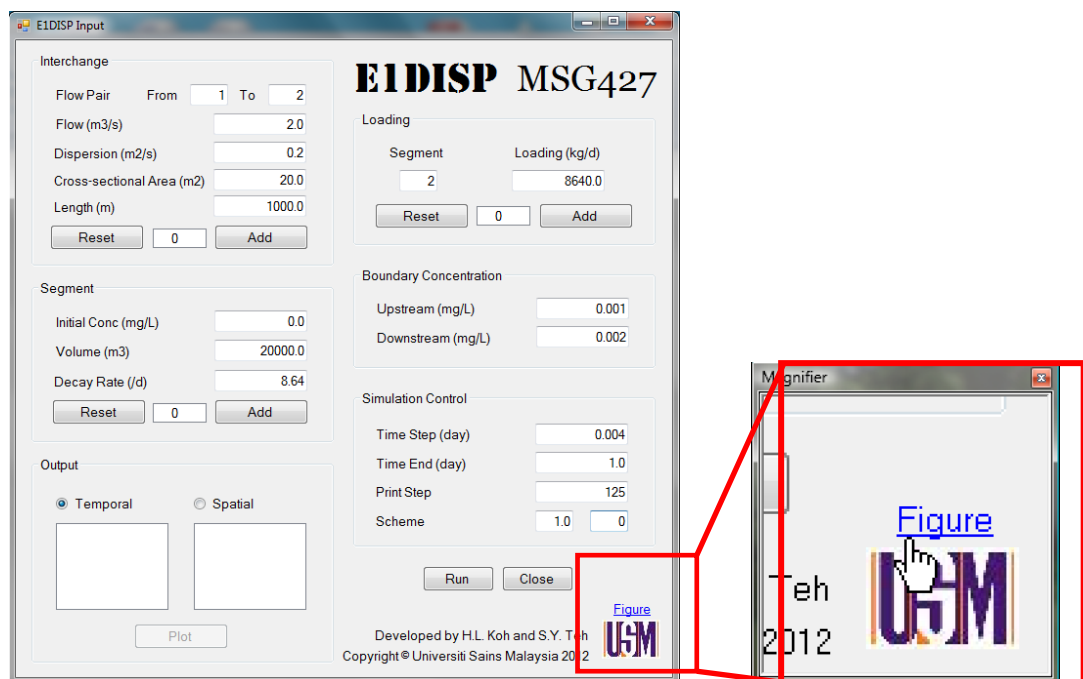


Figure 3: The link Figure in E1DISP input window

2.1 *Description of River*

For convenience of initialization, we treat this river as a uniform river with constant cross-sectional area and constant flow. In reality, the cross-sectional area and flow will increase in the downstream direction. There is a small factory and a small farm releasing a chemical known as BOD into this river at an upstream location. We want to simulate how this BOD will change in its concentration as it moves downstream. We divide this river into several segments. In Figure 2 (left), we have 17 segments with segment 1 as the upstream segment and segment 17 as the downstream segment. Segment 2 is located at the upstream and segment 16 is located at the downstream end. The length of each and every segment is 1000.0 m with a constant cross-sectional area of 20.0 m². The volume of water contained in each segment is 20000.0 m³. We assume that the flow in this river is 2.0 m³/s through the entire stretch, which implies a river velocity of 0.1 m/s. We further assume that the BOD from the factory/farm is discharged into segment 2 at the loading rate of 8640.0 kg/d (= 0.1 kg/s). The decay rate for BOD is 8.64 d⁻¹ (= 0.0001 s⁻¹). We need to impose the upstream and downstream boundary conditions. Again, for convenience, we assume the upstream boundary is 0.001 mg/L and downstream boundary is 0.002 mg/L. The dispersion coefficient is 0.2 m²/s. We have chosen the numbers so that no same number will appear twice in the input screen.

2.2 *How to Start Simulation*

As a start, we suggest that you consider a simulation with only 4 segments (2-5) and two boundary segments (1 and 6) as shown in Figure 2 (right). There will be five interfaces ($I_1 - I_5$) which we call interchange or exchange pairs. The first interchange is the interface between the upstream segment 1 and segment 2. In the same way, interchange 5 is the interface between segment 4 and the downstream segment 6. When you double click on E1DISP.exe, the E1DISP input window (Figure 4) will be displayed with default numbers that reflect the river that we have chosen to simulate. There are FIVE groups of input data: (a) **Interchange** (Figure 5), (b) **Segment** (Figure 6), (c) **Loading** (Figure 7), (d) **Boundary Concentration** (Figure 8), (e) **Simulation Control** (Figure 9) and one group of output data: (f) **Output** (Figure 10). The default values and their respective units will be displayed on the input window, prompting you to either use them as is OR modify them as you wish. Note that the button <Plot> in the Output group will remain deactivated until the model is run successfully.

E1DISP Input

E1DISP MSG427

Interchange

Flow Pair From To

Flow (m3/s)

Dispersion (m2/s)

Cross-sectional Area (m2)

Length (m)

Segment

Initial Conc (mg/L)

Volume (m3)

Decay Rate (/d)

Output

☒ Temporal ☐ Spatial

Loading

Segment Loading (kg/d)

Boundary Concentration

Upstream (mg/L)

Downstream (mg/L)

Simulation Control

Time Step (day)

Time End (day)

Print Step

Scheme

Developed by H.L. Koh and S.Y. Teh
Copyright © Universiti Sains Malaysia 2012

Figure USM

Figure 4: Windows-based E1DISP input window

Interchange

Flow Pair From To

Flow (m3/s)

Dispersion (m2/s)

Cross-sectional Area (m2)

Length (m)

Segment

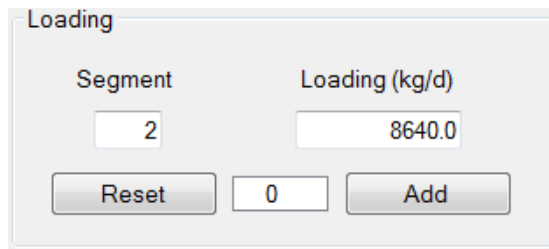
Initial Conc (mg/L)

Volume (m3)

Decay Rate (/d)

Figure 6: Segment input data

Figure 5: Interchange input data

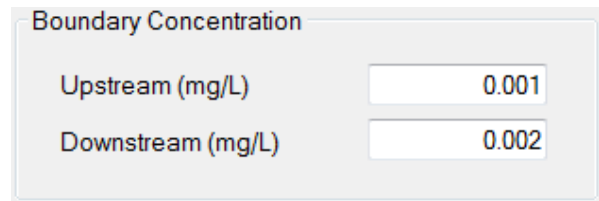


Loading

Segment	Loading (kg/d)
2	8640.0

Reset 0 Add

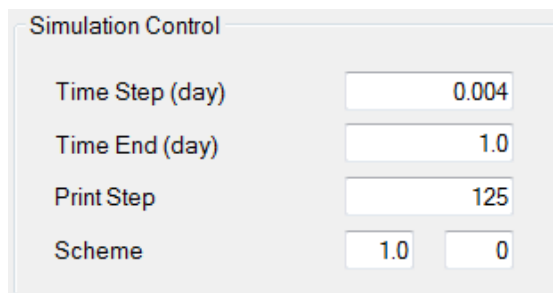
Figure 7: Loading input data



Boundary Concentration

Upstream (mg/L)	0.001
Downstream (mg/L)	0.002

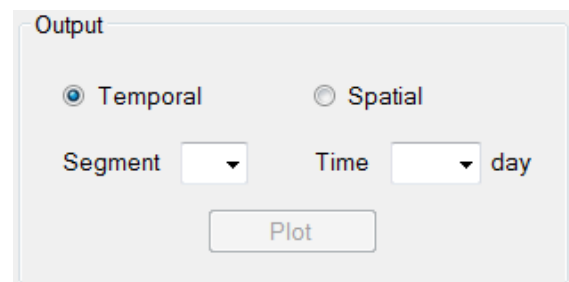
Figure 8: Boundary input data



Simulation Control

Time Step (day)	0.004
Time End (day)	1.0
Print Step	125
Scheme	1.0 0

Figure 9: Simulation control input



Output

☒ Temporal ☐ Spatial

Segment Time day

Plot

Figure 10: Output Graph Control

Interchange

We have decided to simulate with 5 interchanges, with each and every interchange taking the same values as appeared on the screen. To input the first interchange between the upstream segment 1 and segment 2, you click the <Add> button once. Then the displayed values will be assumed for the FIRST interchange in the internal input file, and the number [1] will appear between <Reset> and <Add>. To input the second interchange between the segment 2 and segment 3, change the numbering in Flow Pair (From 2 To 3) and then click <Add> a second time with the SAME values displayed to input SECOND interchange values into the input file. Then [2] will appear between <Reset> and <Add>. Repeat this process a total of 5 times in order to input the pre-set values for the 5 interchanges. This completes the input of the interchange (total of 5).

Segment

Click <Add> just once to input the preset values of segments into the input files for segments. Then [1] will appear between <Reset> and <Add>. This means that segment 2 has been assigned the values as indicated on the segment screen. Then you click

<Add> second time to input values for segment 3. You continue to click <Add> for the third and fourth times to complete input for segments (total of 4, segments 2 – 5).

Boundary

You accept the preset values of upstream and downstream boundary conditions namely 0.001 mg/L and 0.002 mg/L respectively. You may change the preset values as you wish.

Loading

Unless specified here in this segment, all segments have zero loadings. For this example, the BOD loading is discharge into segment 2, i.e. the first segment immediately after the upstream segment. Click <Add> to add the loading of 8640.0 kg/d to segment 2. You may add loading to other segments as you wish.

Simulation Control

The Time Step Δt , Time End and Print Step for the numerical calculation should be input in this section. The scheme values control the numerical scheme in E1DISP and should be left at the default setting of 1.0 and 0. The time step is chosen depending on the characteristics of the problem. A larger time step can be used when the solution varies slowly while smaller time steps are used where the solution varies rapidly. The value for Time End determines when the numerical simulation will end. Print Step controls the interval of output being displayed. For this example, Time Step = 0.004 day, Time End = 1.0 day and Print Step = 500. This means the numerical calculation is performed for $\frac{\text{Time End}}{\text{Time Step}} = \frac{1.0}{0.004} = 250$ iterations. The output will be printed for every 125 (Print Step) iterations, i.e. when $t = 0, 0.5$ and 1.0 day.

You are Ready to RUN.

Click <Run> once to execute the model simulation (Figure 11). If all are set up properly then the simulation results as shown in Figure 12 will appear in a pop up window. The output is also saved in a text file named “answer.txt” (Figure 13) to enable plotting by other software such as Excel. If the model is executed successfully, the <Plot> button in the Output group will be activated. You may select to plot temporal (concentration vs. time) or spatial (concentration vs. segment) plots. For example, selecting Temporal plot and Segment 2 in the drop down box (Figure 14) will display the concentration time

series for segment 2 when the button <Plot> is clicked. If Spatial plot and Time = 0.5 day are selected (Figure 15), the spatial concentration for 0.5 day will be displayed when the button <Plot> is clicked. Good Luck.

Figure 11: Setup model ready to run

Time	Segment 1	Segment 2	Segment 3	Segment 4	Segment 5	Segment 6
0.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
0.500	0.0010	24.9590	12.4666	6.1952	3.0386	0.0020
1.000	0.0010	24.9635	12.4880	6.2471	3.1235	0.0020

Figure 12: Simulated concentrations for each segment at selected times

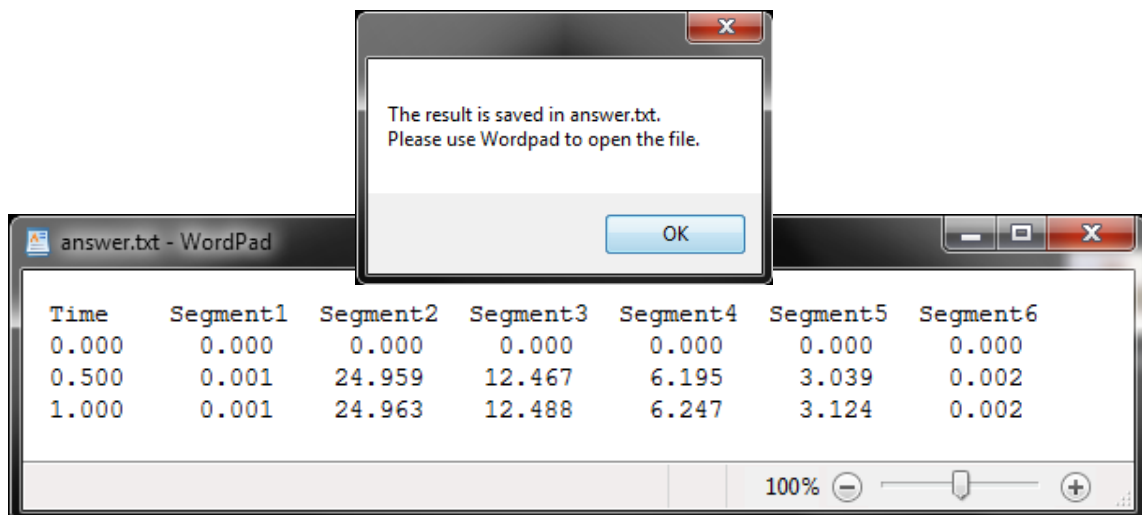


Figure 13: Simulation results saved in “answer.txt”

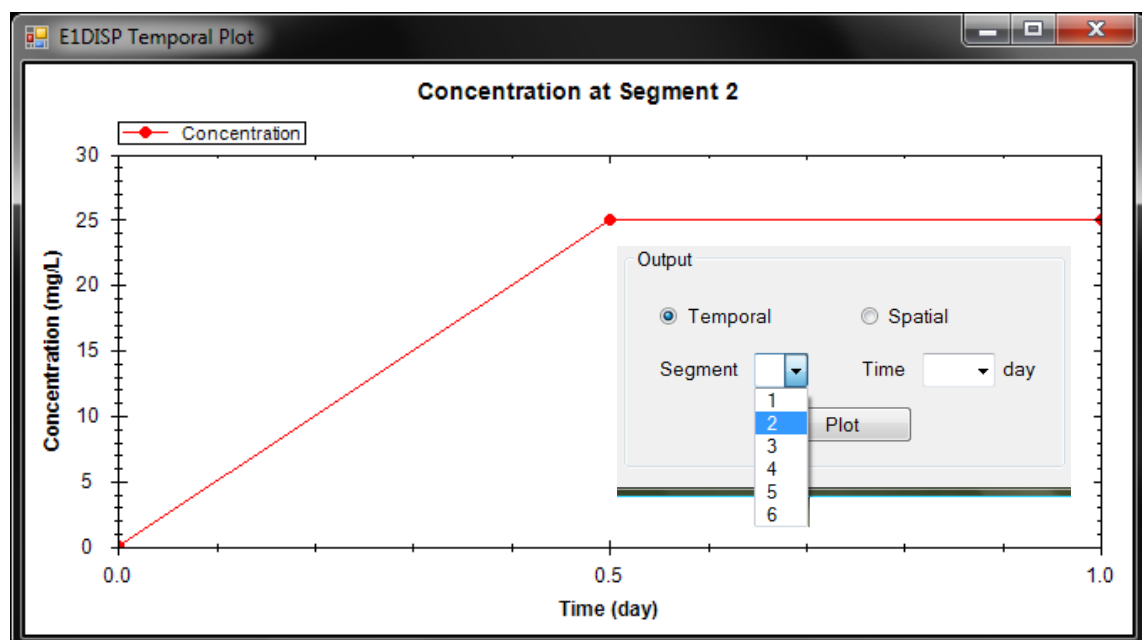


Figure 14: Graph of simulated concentration vs. time for Segment 2

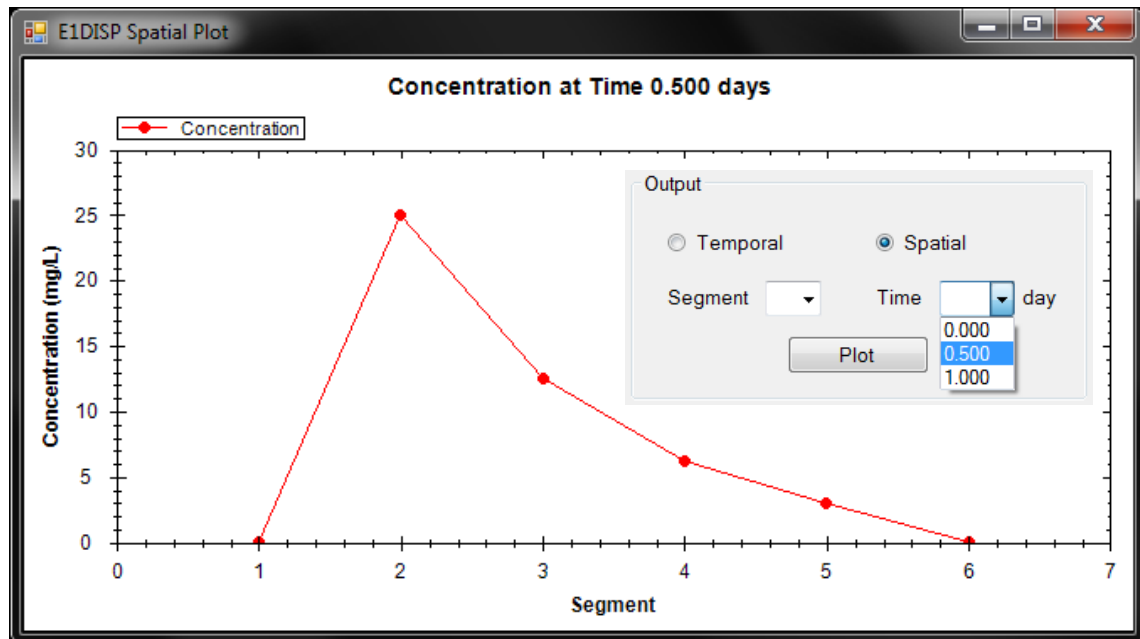


Figure 14: Graph of simulated concentration vs. segment at time $t = 0.5$ day

3. Check the E1DISP model's result with the steady-state solution.
4. Now set up the input data for the river with 15 segments as shown in Figure 2 (left).
5. Consider a completely mixed lake with volume 86400 m^3 . If the initial toxic chemical in the lake is 10 mg/L , the first order decay constant is 0.432 per day, the mean hydraulic detention time of the lake is 10 days and the inflow concentration is 50 mg/L , plot the concentration versus time (in days) of the toxic chemical in the lake outflow. For this purpose, use the E1DISP model. How does your answer compare with analytical solution and steady state solution?