

WASP7 Stream Transport - Model Theory and User's Guide

Supplement to Water Quality Analysis Simulation Program (WASP) User Documentation

Robert B. Ambrose, Jr., P.E.
U.S. EPA, Office of Research and Development
National Exposure Research Laboratory
Ecosystems Research Division
Athens, Georgia

Tim A. Wool
U.S. EPA, Region 4
Water Management Division
Atlanta, Georgia

U.S. Environmental Protection Agency
Office of Research and Development
Washington, DC 20460

NOTICE

The U.S. Environmental Protection Agency (EPA) through its Office of Research and Development (ORD) funded and managed the research described herein. It has been subjected to the Agency's peer and administrative review and has been approved for publication as an EPA document. Mention of trade names or commercial products does not constitute endorsement or recommendation for use.

Abstract

The standard WASP7 stream transport model calculates water flow through a branching stream network that may include both free-flowing and ponded segments. This supplemental user manual documents the hydraulic algorithms, including the transport and hydrogeometry equations, the model input and output, and a series of model verification tests.

For one-dimensional, branching streams or rivers, flow routing can be calculated for free-flowing stream reaches, for ponded reaches, and for backwater or tidally-influenced reaches. Kinematic wave flow routing is a simple but realistic option to drive advective transport through free-flowing segments. The kinematic wave equation calculates flow wave propagation and resulting variations in flows, volumes, depths, and velocities resulting from variable upstream inflow. This well-known equation is a solution of the one-dimensional continuity equation and a simplified form of the momentum equation that considers the effects of gravity and friction. Advective transport through ponded segments is controlled by a sharp-crested weir equation. This equation calculates outflow based on water elevation above the weir. Ponded reaches can include a flowing surface water segment (epilimnion) as well as stagnant underlying segments (hypolimnion).

For dynamic flow through backwater segments, the momentum equations from DYNHYD provide a simple solution for calculating outflows and resultant changes in velocity, surface elevation, depth and volume. Driven by variable upstream flows and downstream heads, the dynamic flow routine solves one-dimensional equations describing the propagation of a long wave through a shallow water system while conserving both momentum and volume. This approach considers the effects of gravity, friction, and convective inertia, assuming that flow is predominantly one-dimensional, that accelerations normal to the direction of flow are negligible, that channels can be adequately represented by a constant top width with a variable hydraulic depth (i.e., "rectangular"), and that bottom slopes are moderate. This option can be used to represent simple two-dimensional (x-y) water bodies with a branching link-node network.

To run the WASP7 stream transport module, the user must supply segment information and flow information. Required segment information includes lengths, widths, and depths for average flow conditions, as well as bottom slopes and Manning friction coefficients. The hydrogeometric depth exponents may also be specified to control the channel shape. Minimum channel depths for zero-flow conditions may be specified, with a default value of 0.001 m. Bottom slopes less than 10^{-6} signify ponded or backwater segments. For ponded segments, the minimum channel depth is interpreted as the outlet weir height. If weir height is not specified, then the segment is treated as a backwater using the dynamic flow option. Required flow information includes the flow pathways for the main channel and each of the tributaries, as well as the inflow time functions for the simulation period.

Transport variables are provided as output for each of the WASP7 kinetic modules. Standard output includes segment outflows, depths, velocities, and widths.

Table of Contents

1- Introduction	1
2- Background	1
2.1 WASP Transport Fields	1
2.2 WASP Surface Water Flow	2
2.2.1 WASP Surface Water Descriptive Flow Options	2
2.2.2 WASP Kinematic Wave Stream Flow Option.....	3
3- Development of Equations.....	4
3.1 Hydrogeometry	4
3.2 Governing Flow Equations	7
3.2.1 Kinematic Wave Flow	7
3.2.2 Ponded Weir Flow	9
3.2.3 Dynamic Flow.....	9
3.3 Implementation of Equations.....	10
3.3.1 Kinematic Wave Flow	11
3.3.2 Ponded Weir Flow	12
3.3.3 Dynamic Flow.....	14
4- Stream Transport Model Inputs.....	16
4.1 Data set screen	16
4.1.1 Restart Options.....	16
4.1.2 Date and Times	17
4.1.3 Non-Point Source File.....	18
4.1.4 Hydrodynamics	18
4.1.5 Solution Technique	19
4.1.6 Time Step Definition.....	19
4.2 Segments screen.....	20
4.2.1 Description.....	20
4.2.2 Volume.....	20
4.2.3 Velocity (multiplier and exponent).....	21
4.2.4 Depth (multiplier and exponent).....	21

4.2.5	Segment Type	22
4.2.6	Bottom Segment.....	22
4.2.7	Length	22
4.2.8	Width.....	22
4.2.9	Minimum Depth.....	22
4.2.10	Slope	22
4.2.11	Segment Roughness	23
4.3	Flows screen.....	23
4.3.1	Flow Field	24
4.3.2	Flow Function	25
4.3.3	Flow Path Function	26
4.3.4	Flow Time Function.....	27
4.4	Exchanges screen	28
4.4.1	Exchange Field.....	28
4.4.2	Exchange Function.....	29
4.4.3	Exchange Path Function	29
4.4.4	Exchange Time Function	30
5- Stream Transport Model Outputs.....		30
6- References		31
7- Appendix 1: Derivation of Equations.....		33
7.1	Hydraulic Exponents for Kinematic Wave Flow	33
8- Appendix 2: Model Verification Tests		33
8.1	Kinematic Wave Tests	34
8.1.1	Stream Transport Test 1	34
8.1.2	Stream Transport Test 2.....	34
8.1.3	Stream Transport Test 3.....	34
8.1.4	Stream Transport Test 4.....	34
8.1.5	Stream Transport Test 5.....	35
8.2	Weir Overflow Verification Tests	35
8.2.1	Weir Overflow Test 1 – Steady Flow	35
8.2.2	Weir Overflow Test 2 – Variable Flow	35

8.2.3	Weir Overflow Test 3 – Long Term Dynamics	35
8.3	Dynamic Flow Verification Tests	35
8.3.1	Dynamic Flow Test 1 - Steady Backwater	35
8.3.2	Dynamic Flow Test 2 – Steady Flow, Elevation	36
8.3.3	Dynamic Flow Test 3 – Variable Stream Flow	36
8.3.4	Dynamic Flow Test 4 – EFDC Stream	36
8.3.5	Dynamic Flow Test 5 – Tidal Stream	36
8.3.6	Dynamic Flow Test 6 – 2-D Tidal Stream.....	36

List of Figures

Figure 1 - Model network with advective transport pathways	1
Figure 2 - WASP Descriptive Flow Options	3
Figure 3 - Channel hydraulic cross-sections	6
Figure 4 - Definition sketch for ponded flow	9
Figure 5 - WASP Main Screen Toolbar, data input buttons	16
Figure 6 Dataset Parameterization	17
Figure 7 - WASP Segment Definition Screen	21
Figure 8 - Flows screen	23
Figure 9 - Example WASP flow input	24
Figure 10 - Exchanges Screen	28

List of Tables

Table 1 - Hydraulic Exponents for Figure 3	6
Table 2 - Comparison of Empirical Hydraulic Exponents	6
Table 3 - Transport output variables	31

1 - Introduction

The Water Quality Analysis Simulation Program, WASP7 (Di Toro et al. 1983, Ambrose et al. 1988, Wool et al. 2001, Wool et al. 2006) is a general dynamic mass balance framework for modeling contaminant fate and transport in surface waters. Based on the flexible compartment modeling approach, WASP can be applied in one, two, or three dimensions with advective and dispersive transport between discrete physical compartments, or “segments.” WASP provides a selection of modules to allow the simulation of conventional water quality variables as well as toxicants.

The WASP kinetic models are based on a set of transport and transformation equations. Advective transport is driven by water flow through a specified computational network (e.g. Figure 1). Inflows bring boundary concentrations into the network, and internal flows advect most constituents along specified flow paths through the network and out the downstream boundaries.

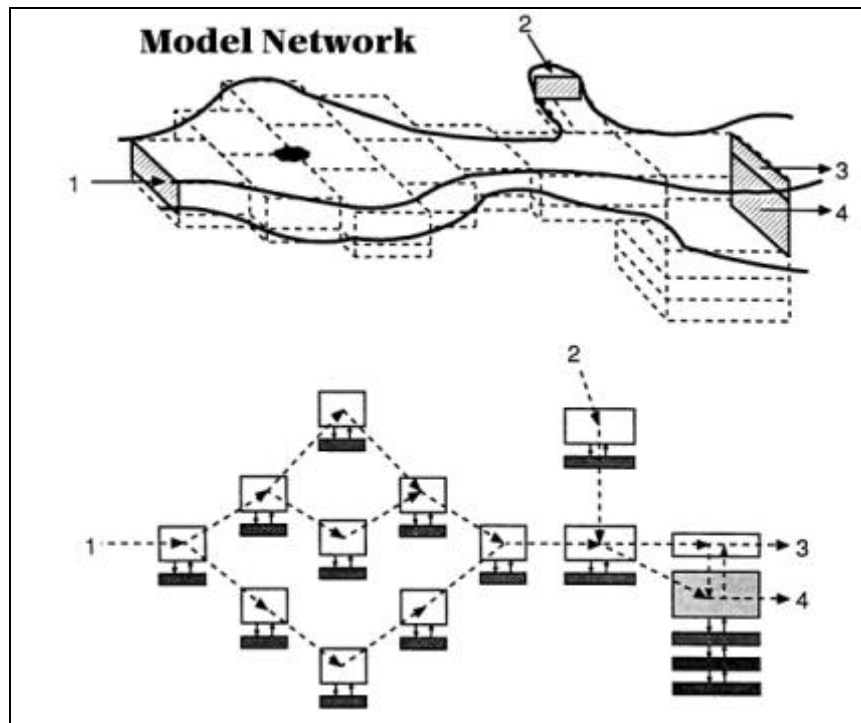


Figure 1 - Model network with advective transport pathways

2 - Background

2.1 WASP Transport Fields

Advective transport in WASP is divided into six distinct types, or "fields." The first transport field is advective flow in the water column. Advective flow carries water quality constituents "downstream" with the water and accounts for instream dilution. The second transport field specifies the movement of pore water in the sediment bed.

Dissolved water quality constituents are carried through the bed by pore water flow. The third, fourth, and fifth transport fields specify the transport of particulate pollutants by the settling, resuspension, and burial of solids. Water quality constituents sorbed onto solid particles are transported between the water column and the sediment bed. The sixth transport field represents evaporation or precipitation from or to surface water segments.

2.2 WASP Surface Water Flow

Advective water column flows directly control the transport of dissolved and particulate pollutants in many water bodies. In addition, changes in velocity and depth resulting from variable flows can affect such kinetic processes as reaeration, volatilization, and photolysis. In WASP, water column flow is input via transport field 1. Circulation patterns may be described (flow options 1 and 2) or simulated by a hydrodynamic model, such as DYNHYD, EPDRIV1, or EFDC (flow option 3). Beginning with version 6.2, WASP is capable of internally calculating flow through a one-dimensional, branching network using the kinematic wave formulation (flow option 4).

2.2.1 WASP Surface Water Descriptive Flow Options

Two descriptive flow options are available in WASP – Net Flows (flow option 1) and Gross Flows (flow option 2). For these descriptive flow options, WASP tracks each separate inflow specified by the user from its point of origin through the model network. For each inflow, the user must supply both a continuity (i.e., unit flow response) function and a time function. The time function describes the inflow as it varies in time. The continuity function describes the unit flow response throughout the network. The actual flow between segments that results from each inflow is the product of the time function and the continuity function.

If several inflow functions are specified, then the total flow between segments is the sum of the individual flow functions. If the total flow into a segment is not equal to the total flow out of a segment, then that segment volume is adjusted by WASP to maintain continuity. Changes in inflows are instantly propagated throughout the model network. While the steady-state effect of several tributaries can be described with these flow options, unsteady flow transients are not calculated.

These descriptive flow options are illustrated in Figure 2. In the Net Flow option, WASP sums all the flows at a segment interface to determine the direction of net flow, and then moves mass in that **ONE** direction. In the Gross Flow option, WASP moves mass with each separate flow at a segment interface. If opposite flows are specified at an interface, WASP will move mass in **BOTH** directions. This option allows the user to describe large dispersive circulation patterns. Note that if all flows are directed downstream, then the mass transport in the Gross Flow option is equivalent to the mass transport in the Net Flow option.

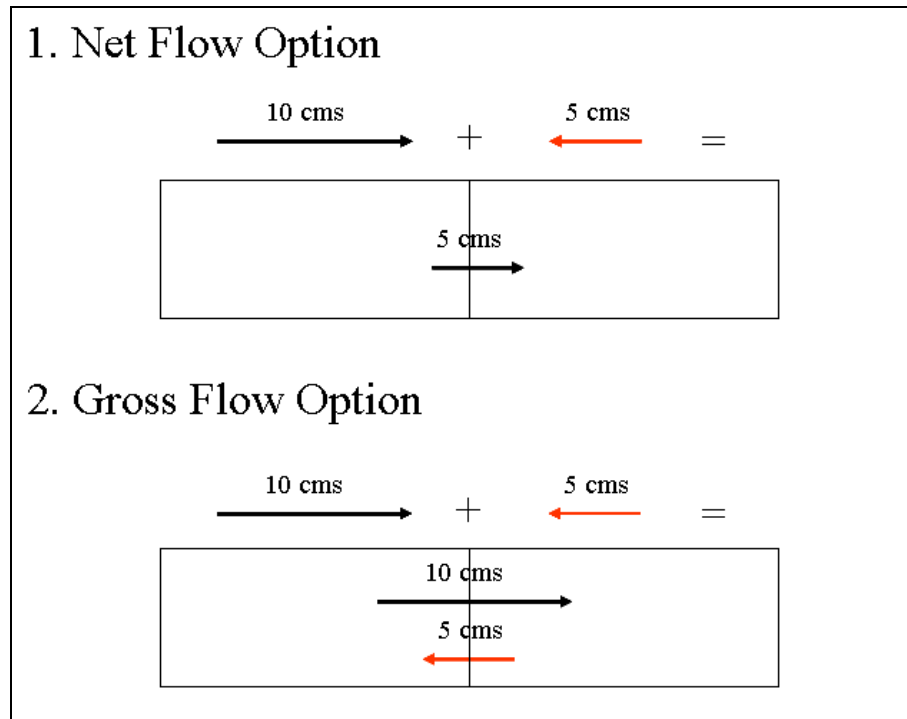


Figure 2 - WASP Descriptive Flow Options

2.2.2 WASP Kinematic Wave Stream Flow Option

The kinematic wave stream flow option was implemented to provide a more realistic simulation of flow dynamics in branching, one-dimensional networks. Kinematic flow is controlled by bottom slope and bottom roughness. The kinematic wave formulation can be used for most stream and small river systems. WASP simulates downstream flow through the network in response to time-variable inflows and withdrawals.

As in the descriptive flow options, the user must supply both a continuity function and a time function for each inflow (or withdrawal). Flow paths may diverge (branch) and then re-join. For surface water segments, the user must specify bottom slopes and roughness factors, as well as widths and depths for average flow conditions and a depth hydraulic exponent for non-rectangular channels. The model uses the inflow and flow path functions, along with specified channel geometry and hydraulic coefficients to calculate time-variable water movement (flows and velocities) and channel hydrogeometry (top widths, cross-sectional average depths, and volumes).

Beginning with version 7.3, the stream network can include ponded flow segments along with kinematic flow segments. Flow through ponded segments is controlled by a downstream low-head dam, weir, or natural sill. For these segments, the user must specify bottom slope of 0 (or less than 10^{-6}) and the downstream weir height. Ponded segments may include stagnant underlying water layers.

Beginning with version 8.0, the stream network can include dynamic flow (or “backwater”) segments along with kinematic flow and ponded flow segments. Dynamic flow is controlled by gradients in surface elevation and velocity, as well as bottom

roughness. For these segments, the user must set bottom slope to 0 and specify bottom elevation in reference to a downstream control point.

3 - Development of Equations

3.1 Hydrogeometry

A good description of segment hydrogeometry as a function of flow can be important in properly using WASP to simulate streams and rivers. For the hydrodynamic linkage flow option, velocities and depths computed by the hydrodynamic model are used in WASP. For the internal flow options (Net Flow, Gross Flow, Kinematic Wave), a set of user-specified hydraulic discharge coefficients defines the relationship between velocity, depth, and stream flow in surface water segments. This method follows the implementation in QUAL2E (Brown and Barnwell, 1987). For the descriptive flow options (Net Flow, Gross Flow), segment velocities and depths do not influence the transport scheme; they are only used in calculations of reaeration and volatilization rates. For the Kinematic Wave flow option, segment velocities, widths, and depths are integral to the transport calculations.

Discharge coefficients giving depth and velocity from stream flow are based on empirical observations of the stream flow relationship with velocity and depth (Leopold and Maddox, 1953). The equations relate velocity, channel width, and depth to stream flow through power functions:

Equation 1

$$v = vmult \cdot Q^{vexp}$$

Equation 2

$$R = dmult \cdot Q^{dexp}$$

Equation 3

$$B = bmult \cdot Q^{bexp}$$

where v is velocity [m/sec], R is hydraulic radius, or cross-sectional average depth [m], B is top width [m], $vmult$, $dmult$, and $bmult$ are empirical coefficients, and $vexp$, $dexp$, and $bexp$ are empirical exponents. Cross-sectional area, A is the product of top width and average depth, and from continuity, flow is given by:

Equation 4

$$\begin{aligned} Q &= v \cdot A = v \cdot R \cdot B = (vmult Q^{vexp}) \cdot (dmult Q^{dexp}) \cdot (bmult Q^{bexp}) \\ &= (vmult \cdot dmult \cdot bmult) Q^{vexp + dexp + bexp} \end{aligned}$$

From inspection, the following hydraulic relationships hold:

Equation 5

$$vmult \cdot dmult \cdot bmult = 1$$

Equation 6

$$v_{\text{exp}} + dxp + b_{\text{exp}} = 1$$

The Net Flow and Gross Flow options in WASP require specification of the hydraulic relationships for velocity and depth; the width coefficients are calculated internally from Equation 5 and Equation 6. The Kinematic Wave Flow option requires specification of the hydraulic depth exponent d_{xp} , along with depth D_m and width B_m under average flow conditions. Manning's equation (rearranged) is used to calculate velocity v_m under average flow conditions, and then average flow Q_m from depth, width, and velocity:

Equation 7

$$v_m = \frac{D_m^{2/3} \cdot S_f^{1/2}}{n}$$

Equation 8

$$Q_m = v_m \cdot D_m \cdot B_m$$

A consistent set of hydraulic exponents are set (see Appendix, Section 7.1):

Equation 9

$$v_{\text{exp}} = \left(\frac{2}{3}\right) \cdot dxp$$

Equation 10

$$b_{\text{exp}} = 1 - dxp - v_{\text{exp}}$$

Finally, a consistent set of hydraulic multipliers are then derived from mean flow width, the hydraulic geometry equations, and Manning's equation:

Equation 11

$$b_{\text{mult}} = B_m \cdot Q_m^{-b_{\text{exp}}}$$

Equation 12

$$v_{\text{mult}} = 1 / \alpha$$

Equation 13

$$d_{\text{mult}} = \alpha / b_{\text{mult}}$$

Equation 14

$$\alpha = \left(\frac{n \cdot b_{\text{mult}}^{2/3}}{S_f^{1/2}} \right)^{3/5}$$

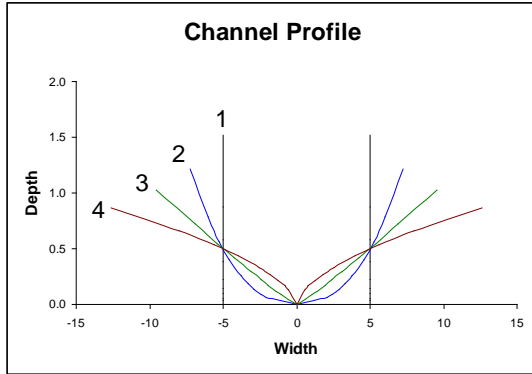


Figure 3 - Channel hydraulic cross-sections

Channel	Velocity	Depth	Width
1. Rectangular	0.40	0.60	0.00
2. U-Shape	0.32	0.48	0.20
3. V-Shape	0.26	0.39	0.35
4. Shallow	0.20	0.30	0.50

Table 1 - Hydraulic Exponents for Figure 3

Channel cross-sections for representative hydraulic geometry coefficients in Table 1 are illustrated in Figure 3. Under mean flow, these channels are 10 m wide and 0.5 m deep. Leopold et al. (1964) have noted that stream channels in humid regions tend towards a rectangular cross-section because cohesive soils promote steep side slopes whereas noncohesive soils encourage shallow sloped, almost undefined banks.

Table 2 - Comparison of Empirical Hydraulic Exponents

Channel	Velocity	Depth	Width
Rectangular	0.40	0.60	0.00
Average of 158 U.S. Gaging Stations	0.43	0.45	0.12
Average of 10 Gaging Stations on Rhine River	0.43	0.41	0.13
Ephemeral Streams in Semiarid U.S.	0.34	0.36	0.29

Table 2 compares hydraulic exponents for a rectangular channel with data reported by Leopold et al. (1964). Note that the average velocity exponent is relatively constant for all channel cross sections. The major variation occurs as a decrease in the depth exponent and concomitant increase in the width exponent as channel cross-sections change from the steep side slopes characteristic of cohesive soils to the shallow slopes of arid regions with noncohesive soils.

For site-specific river or stream simulations, hydraulic coefficients and exponents must be estimated. Brown and Barnwell (1987) recommended estimating the exponents (b and d) and then calibrating the coefficients (a and c) to observed velocity and depth. The exponents may be chosen based on observations of channel shape noted in a reconnaissance survey. If cross sections are largely rectangular with vertical banks, the first set of exponents shown should be useful. If channels have steep banks typical of areas with cohesive soils, then the second set of exponents is appropriate. If the stream is

in an arid region with typically noncohesive soils and shallow sloping banks, then the last set of exponents is recommended.

The key property of the channel that should be noted in a reconnaissance survey is the condition of the bank slopes or the extent to which width would increase with increasing stream flow. Clearly the bank slopes and material in contact with the stream flow at the flow rate(s) of interest are the main characteristics to note in a reconnaissance. This gives general guidance but it should be noted that values are derived for bankful flows. Even in streams with vertical banks, the low flows may be in contact with a sand bed having shallow sloped, almost nonexistent banks that are more representative of ephemeral streams in semi-arid areas.

3.2 Governing Flow Equations

The WASP stream flow model consists of a set of one-dimensional equations solving water flow and water volume in a branching stream or shallow river network. This network can include free-flowing stream reaches (kinematic wave flow), ponded reaches (weir overflow), and backwater or tidally influenced reaches (dynamic flow). The equation of motion, based on the conservation of momentum, predicts water velocities and flows. The equation of continuity, based on the conservation of volume, predicts water heights (heads) and volumes.

The one-dimensional continuity equation is given by:

Equation 15

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} = 0$$

where Q is volumetric flow, [m³/sec] and A is cross-sectional area [m²]. For rectangular channels, where width is constant, Equation 15 becomes:

Equation 16

$$\frac{\partial Q}{\partial x} + B \frac{\partial H}{\partial t} = 0$$

where B is channel width [m] and H is water surface elevation [m]. As presently implemented in WASP, kinematic flow reaches have shapes described by the hydrogeometric relationships described in Section 3.1, while ponded reaches and dynamic flow reaches have rectangular channel shapes.

The equations of motion implemented in the three reach types are described in the following sections.

3.2.1 Kinematic Wave Flow

For one-dimensional, free-flowing stream reaches, kinematic wave flow routing is a simple but realistic option to drive advective transport. The kinematic wave equation calculates flow wave propagation and resulting variations in flows, velocities, widths, and depths throughout a stream network. This well-known equation is a solution of the one-

dimensional continuity equation and a simplified form of the momentum equation that considers the effects of gravity and friction:

Equation 17

$$g (S_0 - S_f) = 0$$

where g is acceleration of gravity [m/sec²], S_0 is the bottom slope, and S_f is the friction slope. Manning's equation expresses the friction force as a function of water velocity and hydraulic radius:

Equation 18

$$S_f = \frac{n^2 v^2}{R^{4/3}}$$

where n is the Manning friction factor, v is water velocity [m/sec], and R is hydraulic radius [m], which is equivalent to the cross-sectional average depth, D . From the simplified momentum equation, S_0 can be equated to S_f . Hydraulic radius can be expressed as cross-sectional area divided by width, B [m]. Substituting these into the Manning's equation and rearranging terms gives flow as a function of bottom slope, cross-sectional area, and width:

Equation 19

$$Q = \frac{1}{n} \frac{A^{5/3}}{B^{2/3}} S_0^{1/2}$$

Substituting this expression into the continuity equation and differentiating A with respect to time gives the kinematic wave differential equation:

Equation 20

$$\frac{\partial Q}{\partial x} + \alpha \beta Q^{\beta-1} \frac{\partial Q}{\partial t} = 0$$

where, for rectangular channels:

Equation 21

$$\beta = 3/5, \quad \alpha = \left(\frac{n B^{2/3}}{S_f^{1/2}} \right)^{3/5}$$

For channels where width varies with flow, α and β are functions of hydraulic coefficients:

Equation 22

$$\beta = 0.6 + 0.4 \cdot dxp, \quad \alpha = \left(\frac{n \cdot bmult^{2/3}}{S_f^{1/2}} \right)^{3/5}$$

where the hydraulic coefficients dxp and $bmult$ are defined in Section 3.1.

3.2.2 Pondered Weir Flow

For flow through pondered segments controlled by a downstream low-head dam or natural sill (Figure 4), the sharp-crested weir overflow equation is a simple solution for calculating outflows and resultant changes in depth and volume. Weir height H_w [m] and width B_w [m] are specified by the user, and hydraulic head H_h is the difference between pondered depth H and H_w .

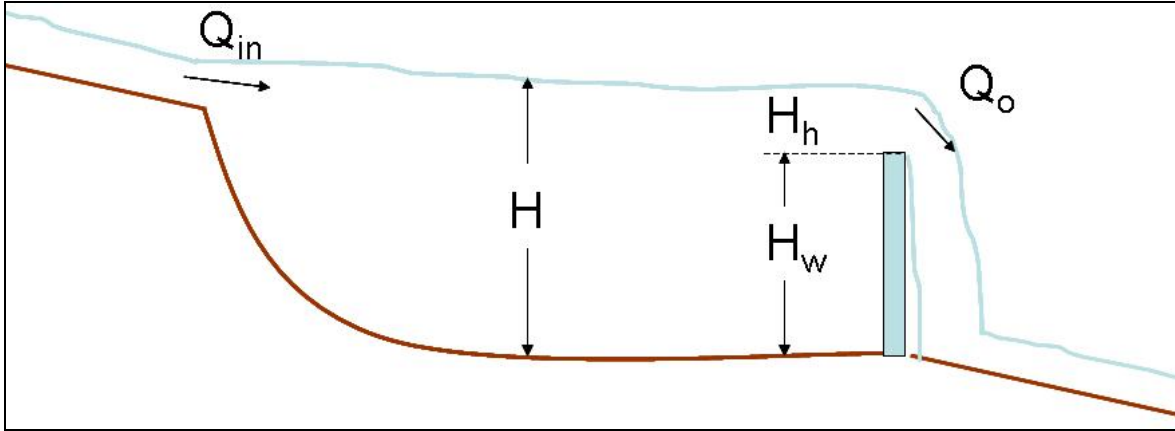


Figure 4 - Definition sketch for pondered flow

For a sharp-crested weir where $H_h/H_w < 0.4$, velocity and flow are related to head by (Finnemore and Franzini 2002):

Equation 23

$$\begin{aligned} v_o &= 1.83 H_h^{1/2} \\ Q_o &= v_o A_h = 1.83 B_w H_h^{3/2} \\ &= 0 \quad \text{when } H \leq H_w \end{aligned}$$

where v_o is the velocity of flow over the weir [m/sec], and A_h is the cross-sectional area of flow over the weir [m²], given by the product of H_h and B_h .

3.2.3 Dynamic Flow

For dynamic flow through backwater segments, the momentum equations from DYNHYD provide a simple solution for calculating outflows and resultant changes in velocity, surface elevation, depth and volume. Bottom elevation H_b [m], width B [m], initial depth D [m], and initial velocity v [m/sec] are specified by the user. Surface elevation Y is the sum of D and H_b . Driven by variable upstream flows and downstream heads, simulations typically proceed at 1- to 5-minute intervals.

The dynamic flow routine solves one-dimensional equations describing the propagation of a long wave through a shallow water system while conserving both momentum and volume. The equation of motion calculates water velocities and flows. The equation of continuity calculates surface elevations, along with associated depths and volumes. This

approach assumes that flow is predominantly one-dimensional, that accelerations normal to the direction of flow are negligible, that channels can be adequately represented by a constant top width with a variable hydraulic depth (i.e., "rectangular"), that the wave length is significantly greater than the depth, and that bottom slopes are moderate. Reaches with steep bottom slopes are best solved with the kinematic wave equations.

Equation 16 gives the equation of continuity implemented for dynamic flow reaches. The equation of motion calculates local acceleration, the velocity rate of change with respect to time [m/sec²]:

Equation 24

$$\frac{dv}{dt} = -v \frac{dv}{dx} + a_g + a_f$$

where v is water velocity [m/sec], a_g is gravitational acceleration along the axis of the channel [m/sec²], and a_f is frictional acceleration [m/sec²]. The first term, convective inertia or Bernoulli acceleration, represents the rate of momentum change by mass transfer, [m/sec²]. The second term, gravitational acceleration, is driven by the slope of the water surface:

Equation 25

$$a_g = -g \frac{\partial Y}{\partial x}$$

where Y is surface elevation [m], and g is the acceleration of gravity [9.81 m/sec²]. The third term, frictional acceleration, can be expressed using the Manning equation for steady uniform flow:

Equation 26

$$a_f = -g \frac{n^2 v^2}{R^{4/3}}$$

where n is the Manning friction factor, v is water velocity [m/sec], and R is hydraulic radius [m], which is equivalent to the cross-sectional average depth, D .

Local wind acceleration is not included in this implemented of the dynamic flow equations.

3.3 Implementation of Equations

WASP7 solves the kinematic flow, ponded flow, and dynamic flow equations for appropriate surface water segments in a stream network using finite-difference formulations for flow and for continuity.

For each segment, a maximum numerical time step DT_{max} is calculated from the segment length and characteristic velocity, as described in the sections below. The overall time step is the product of the minimum DT_{max} in the network and a user-specified fraction, DTF (default = 0.9) that is set to ensure stability:

Equation 27

$$DT = DTF \cdot \min(DT_{\max})$$

This time step DT is divided into two half time steps. For kinematic wave reaches and ponded weir reaches, flows are calculated sequentially for each half time step and then averaged for subsequent use by the water quality module. Final velocities, depths, volumes, and surface elevations at the end of the full time step are passed along to the water quality module.

For dynamic flow reaches, the two half time steps are used in a predictor-corrector scheme as described in Section 3.3.3 below. Flows, velocities, volumes, depths, and surface elevations are updated following each of the numerical steps. Final flows, velocities, volumes, depths, and surface elevations at the end of the full time step are passed along to the water quality module.

3.3.1 Kinematic Wave Flow

To solve for flow in kinematic wave reaches, Equation 20 is expressed in finite difference form:

Equation 28

$$\frac{Q_{Dt} - Q_{D0}}{DT} = \frac{Q_{Dt}^{1-\beta}}{\alpha \beta L} (Q_U - Q_{D0})$$

where Q_U is the upstream inflow, Q_{D0} is the outflow from the preceding time step, Q_{Dt} is the outflow for this time step, and DT is the time step [days]. Equation 28 is solved using a Newton-Raphson approach:

Equation 29

$$f(Q_{Dt}) = Q_{Dt} + c_1 Q_{Dt}^{1-\beta} + c_2 = 0, \quad c_1 = -\frac{DT}{\alpha \beta L} (Q_U - Q_{D0}), \quad c_2 = -Q_{D0}$$

Equation 30

$$f'(Q_{Dt}) = 1 + c_1 (1 - \beta) Q_{Dt}^{-\beta}$$

Given an initial estimate of Q_{Dt} , an updated estimate, Q_{Dt2} , is calculated by:

Equation 31

$$Q_{Dt2} = Q_{Dt} - \frac{f(Q_{Dt})}{f'(Q_{Dt})}, \quad err = \left| \frac{Q_{Dt2} - Q_{Dt}}{Q_{D0}} \right|$$

Equation 29, Equation 30, and Equation 31 are solved in an iterative loop where Q_{Dt} is set equal to Q_{Dt2} until err is less than 10^{-5} . Given the new set of flows for the water column network $Q_{Dt,ij}$, volumes for all water segments “ i ” are updated using the continuity equation:

Equation 32

$$DV_i = \sum_j Q_{Dt,ij} DT, \quad V_{t,i} = V_{0,i} + DV_i$$

Segment widths are updated with the new flows using Equation 3. Associated cross-sectional areas and depths are then calculated:

Equation 33

$$DA_i = \frac{DV_i}{L_i}, \quad A_{t,i} = A_{0,i} + DA_i$$

Equation 34

$$D_{t,i} = \frac{A_{t,i}}{B_i}$$

To prevent slow numerical drift in calculated volumes during lengthy simulations, small adjustments are made to flows based on differences between hydraulic radius calculated from Equation 2 and cross-sectional average depth calculated through continuity. Applying Equation 19:

Equation 35

$$Q_{err,i} = Q_{Dt,i} - Q_{R,i} = \frac{B_i}{n_i} S_{0,i}^{1/2} (D_i^{5/3} - R_i^{5/3})$$

Equation 36

$$Q_{Dt,i} = Q_{Dt,i} + Q_{err,i}$$

For each kinematic flow segment, a maximum stable numerical time step DT_{max} [days] is calculated from the segment length L [m] and celerity c [m/s]:

Equation 37

$$c = v / \beta$$

Equation 38

$$DT_{max} = \frac{0.5}{86400} \cdot \left(\frac{L}{c} \right)$$

where 0.5 is a safety factor.

3.3.2 Pondered Weir Flow

For ponded reaches “ i ”, weir overflow (Equation 23) must be solved along with continuity such that:

Equation 39

$$Q_{Dt,i} = 1.83 B_{w,i} H_{ht,i}^{3/2}$$

and

Equation 40

$$H_{ht,i} = H_{h0,i} + \frac{Q_{Ut,i} - Q_{Dt,i}}{B_i L_i} DT$$

where $Q_{Ut,i}$ and $Q_{Dt,i}$ are the upstream inflow and outflow for this time step [m^3/sec], $H_{ht,i}$ is the head for this time step [m], $H_{h0,i}$ is the head from the previous time step [m], and DT is the time step [$days$]. These equations are solved using a Newton-Raphson approach where:

Equation 41

$$f(Q_{Dt}) = Q_{Dt} - 1.83 B H_{ht}^{3/2} = 0$$

Equation 42

$$\begin{aligned} f'(Q_{Dt}) &= 1 - 1.83 B \left(\frac{3}{2} H_{ht,i}^{1/2} \frac{DT}{B_i L_i} \right) \\ &= 1 + \frac{1.5 \cdot 1.83 \cdot DT}{L_i} H_{ht,i}^{1/2} \end{aligned}$$

Given an initial estimate of $Q_{Dt,i}$, a consistent value for H_{ht} is calculated using Equation 40. An updated estimate Q_{Dt2} , is then calculated by:

Equation 43

$$Q_{Dt2} = Q_{Dt} - \frac{f(Q_{Dt})}{f'(Q_{Dt})}, \quad err = \left| \frac{Q_{Dt2} - Q_{Dt}}{Q_{Do}} \right|$$

Equation 41, Equation 42, Equation 43, and Equation 40 are solved in an iterative loop where Q_{Dt} and H_{ht} are set equal to Q_{Dt2} and H_{ht2} until err is less than 10^{-5} .

WASP7 solves this weir overflow equation for each ponded segment in a stream network. In each of the numerical steps, calculated values of dQ/dt and Q_o are used to update volumes, depths, and H_h , which are used in the next numerical step to calculate Q_o .

For each weir overflow segment, a maximum stable numerical time step DT_{max} [$days$] is calculated from the segment length and overflow velocity v_o :

Equation 44

$$DT_{max} = \frac{0.5}{86400} \cdot \left(\frac{L}{1.5 v_o} \right)$$

where 0.5 and 1.5 are safety factors.

3.3.3 Dynamic Flow

Equation 16 and Equation 24 form the basis of the hydrodynamic model DYNHYD5, which is implemented within this version of WASP. These equations are integrated numerically on a flexible, computationally efficient "link-node" network (Feigner and Harris, 1970), which solves the equations of motion and continuity at alternating grid points. At each time step, the equation of motion is solved at the links (or "channels"), giving velocities for mass transport calculations, and the equation of continuity is solved at the nodes (or "junctions"), giving heads for pollutant concentration calculations. Link-node networks can treat fairly complex branching flow patterns and irregular shorelines with acceptable accuracy for many studies. They cannot handle stratified water bodies, small streams, or rivers with a large bottom slope. Link-node networks can be set up for wide, shallow water bodies if primary flow directions are well defined.

In WASP, nodes correspond to segments, and links correspond to segment interfaces. For every dynamic flow segment in a WASP network, a distinct channel number "*ich*" is defined for each of its downstream segments. Channel *ich* is defined by upstream segment *j* and downstream segment *i*. Positive flow in channel *ich* is outflow from segment *j* and inflow to segment *i*. Negative flow in channel *ich* is outflow from segment *i* to segment *j*.

In finite difference form, Equation 24 is given by:

Equation 45

$$\frac{v_{t,ich} - v_{ich}}{DT} = v_{ich} \frac{\Delta v_{ich}}{\Delta x_{ich}} - g \frac{\Delta H_{ich}}{\Delta x_{ich}} - \frac{g n_{ich}^2}{R_{ich}^{4/3}} v_{ich} |v_{ich}|$$

where $v_{t,ich}$ is the velocity for this time step [m/sec], v_{ich} is the velocity from the preceding time step [m/sec], Δx_{ich} is channel *ich* length [m], $\Delta v_{ich} / \Delta x_{ich}$ is the velocity gradient in channel *ich* with respect to distance [sec^{-1}], $\Delta H_{ich} / \Delta x_{ich}$ is the water surface gradient in channel *ich* with respect to distance [m/m], and DT is the time step [days]. All values on the right hand side of equation 20 are referenced to the previous time step.

The water surface gradient can be computed from the junction heads at either end of the channel:

Equation 46

$$\frac{\Delta H_{ich}}{\Delta x_{ich}} = \frac{H_j - H_i}{(L_j + L_i)/2}$$

where H_j and L_j are the water surface elevation and length of the upstream segment[m], and H_i and L_i are the water surface elevation and length of the downstream segment[m].

The velocity gradient cannot be computed directly from upstream and downstream channel velocities because of possible branching in the network. If branching does occur, there would be several upstream and downstream channels, and any computed velocity gradient would be ambiguous. An expression for the velocity gradient within a channel can be derived by applying the continuity equation to the channel and substituting $v \times A$ for Q :

Equation 47

$$\frac{\partial A}{\partial t} = -\frac{\partial Q}{\partial x} = -v \frac{\partial A}{\partial x} - A \frac{\partial v}{\partial x}$$

Rearranging terms gives the channel velocity gradient:

Equation 48

$$\frac{\partial v}{\partial x} = -\frac{1}{A} \frac{\partial A}{\partial t} - \frac{v}{A} \frac{\partial A}{\partial x}$$

Writing this in finite difference form and substituting $B \times R$ for A and $B \times \Delta H$ for MA gives the velocity gradient term:

Equation 49

$$\frac{\Delta v_{ich}}{\Delta x_{ich}} = -\frac{1}{R_{ich}} \frac{\Delta H_{ich}}{\Delta t} - \frac{v_{ich}}{R_{ich}} \frac{\Delta H_{ich}}{\Delta x_{ich}}$$

The term $\Delta H_{ich}/\Delta t$ is computed as the average water surface elevation change between the segments at each end of channel ich during time step t . Substituting Equation 49 into Equation 45 and rearranging gives the explicit finite difference equation of motion applied to each channel:

Equation 50

$$v_{t,ich} = v_{ich} + DT \left[\frac{v_{ich}}{R_{ich}} \frac{\Delta H_{ich}}{\Delta t} + \left(\frac{v_{ich}^2}{R_{ich}} - g \right) \frac{\Delta H_{ich}}{\Delta x_{ich}} - \frac{g n_{ich}^2}{R_{ich}^{4/3}} v_{ich} |v_{ich}| \right]$$

Writing the equation of continuity in finite difference form and rearranging terms gives:

Equation 51

$$H_{t,j} = H_j - DT \frac{\sum_i Q_{ij}}{B_j L_j}$$

Equation 50 and Equation 51 are solved using a 2-step predictor-corrector routine. Based on initial velocities, surface elevations, and depths from the previous time step, new velocities and flows are solved for the half time step, along with new surface elevations and depths. Using these predicted half-time step values, velocity and flow derivatives are recalculated for the half time step. These corrected derivatives are then used with the initial velocities, depths, and surface elevations to calculate velocities and flows for the full time step. Finally, surface elevations, depths, and volumes are calculated for the full time step.

For each dynamic flow segment, a maximum stable numerical time step DT_{max} [days] is calculated from the segment length L [m] and celerity c [m/s]:

Equation 52

$$c = \sqrt{g D}$$

Equation 53

$$DT_{\max} = \frac{0.5}{86400} \cdot \left(\frac{L}{c} \right)$$

where 0.5 is a safety factor.

4 - Stream Transport Model Inputs

To implement stream flow routing, the user must specify information in the Data set screen, the Segments screen, and the Flows screen, accessed from the gears, the cube, and the faucet on the main WASP toolbar (Figure 5). Each of these is briefly described in the sections below.

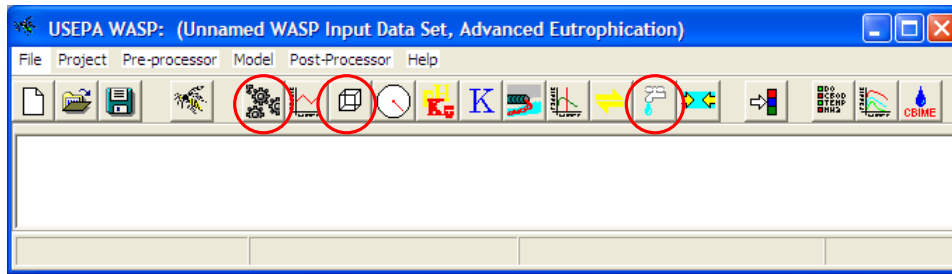


Figure 5 - WASP Main Screen Toolbar, data input buttons

4.1 Data set screen

In the Data set screen (**Error! Reference source not found.**), the user must select the Model Type. In the Time Range section, the user must specify the simulation Start Date, Start Time, End Date and End Time. In the hydrodynamics section, the user must select the flow option. Finally, the Time Step information should be specified. Default values are supplied for Fraction of max time step (0.9), Max time step (1.0 day), and Min time step (0.0001 day). These values should work well for most cases, but if numerical instability is encountered, lowering the Fraction of max time step (to 0.5 or even 0.1) could help. In some cases, the user may want to specify a maximum times step of less than a day. If diurnal output is desired, then a maximum time step of 0.1 or 0.05 days should give the necessary precision.

When creating a new input dataset the input parameterization data entry form is the first one that needs to be completed. This form provides basic information that is needed by the program to parameterize the other data entry forms that follow. This screen informs the program what type of WASP file you are going to be creating.

4.1.1 Restart Options

The methods used by WASP to read and create restart files have changed substantially in this version. In previous versions the user would have selected Create Restart File, for WASP to write the final conditions of the simulation to an output file. This is true for the current version as well. If the user wants to restart a simulation with the final conditions of previous simulation this radio must be set. At the end of the WASP simulation a restart file with the same name as the WIF except with the extension *.RST will be saved.

With the current release of WASP if the user wants to use a restart file they simply click on the Load Restart File button, this will allow the user to browse to whatever restart file they want to use. Once the file is selected and the user clicks on the Okay button, the restart file is opened up and segment volumes and state variable initial conditions are reset to the values in the user selected *.RST file.

Figure 6 Dataset Parameterization

4.1.2 Date and Times

The previous versions of WASP did not require that the model time functions be represented in Gregorian date format. WASP requires all time functions be represented in Gregorian fashion (mm/dd/year hh:mm:ss).

Start Date and Start Time - The start time dialog is used to define the date and time for the start of the simulation or time period being considered in the model input files. This date and time correspond to time zero within the model.

End Date and End Time - The end time dialog is used to define the date and time when the simulation will end.

Skip Ahead to Date and Time - This new addition to the WASP interface allows the user to skip to any portion of the simulation and/or the selected loaded hydrodynamic linkage file. When the user selects a hydrodynamic linkage file the start time and end

time of the file is read and the interface automatically sets the beginning and end time to these values. It is best that the user build all of the time series (environmental, boundary and dispersion) to cover this full range of time. Once the WIF is built the user can set a date and time to skip the simulation from the start time set in the hydrodynamic file. This is handy for using the whole hydrodynamic linkage file for calibration and verification, and then using a small portion of the hydrodynamic linkage file for scenario analysis. It could be the critical time period that will be used for the waste load allocation or TMDL. The start time of the simulation should still be set the beginning time in the hydrodynamic linkage file. The user can change the end time of the simulation by changing the last date/time pair in the Time Step screen.

4.1.3 Non-Point Source File

The non-point source file is an external file that contains a time-series of loads (kg/day) for a given segment and system. This file is typically created either by the user manually or using other software like the Stormwater Management Model (SWMM) in conjunction with the Linked Watershed/Waterbody Model. This file can be used to provide loading information to WASP on virtually any time scale, from time step to time step, to year average loads.

4.1.4 Hydrodynamics

There are currently three surface flow options available for WASP. The first two options pertain to how WASP will calculate the exchange of mass between adjoining segments with flow in both directions across a segment interface. The three flow options available for surface water flow are:

1. Gross Flows -- WASP will calculate net transport across a segment interface that has opposing flow. WASP will net the flows and move the mass from the segment that has the higher flow leaving. If the opposed flows are equal no mass is moved.
2. Net Flows -- Pertains to mass and water being moved without regard to net flow.
3. Kinematic Wave -- For one-dimensional, branching streams or rivers, kinematic wave flow routing is a simple but realistic option to drive advective transport. The kinematic wave equation calculates flow wave propagation and resulting variations in flows, volumes, depths, and velocities throughout a stream network.
4. Hydrodynamic Linkage -- Realistic simulations of unsteady transport in rivers, reservoirs, and estuaries can be accomplished by linking WASP7 to a compatible hydrodynamic simulation. This linkage is accomplished through an external "hyd" file chosen by the user at simulation time. The hydrodynamic file contains segment volumes at the beginning of each time step, and average segment interfacial flows during each time step. WASP7 uses the interfacial flows to calculate mass transport, and the volumes to calculate constituent concentrations. Segment depths and velocities may also

be contained in the hydrodynamic file for use in calculating reaeration and volatilization rates. Before using hydrodynamic linkage files with WASP, a compatible hydrodynamic model must be set up for the water body and run successfully, creating a hydrodynamic linkage file with the extension of *.hyd. This is an important step in the development of the WASP input file because the hydrodynamic linkage file contains all necessary network and flow information. When Hydrodynamic Linkage is selected in the Data Set Parameters screen, the user cannot provide any additional surface flow information. When you are ready to begin the development of a WASP input deck, simply open the hydrodynamic linkage file from within the data preprocessor. The hydrodynamic linkage dialog box allows the user to browse and select a hydrodynamic linkage file. The data preprocessor will open the hydrodynamic interface file and extract the number of segments, the starting and ending time. The data processor will also determine the set of boundary segments (segments that receive flow from outside the model network) and set the boundary concentrations to 1.0 mg/L. Once a hydrodynamic linkage file is selected in the data preprocessor, WASP has enough information to execute a simple test run with no loads or kinetics enabled. This step is recommended to test the network and transport integrity. If the simulation is run for a sufficient duration, concentrations should approach 1.0 mg/L throughout the network. If you are getting a number other than 1 mg/L, you may have to use a different time step in the hydrodynamic model. This is especially true if the concentrations are oscillating between large and small numbers, a clear indication of numerical instability. WASP has the ability to get hydrodynamic information from a host of hydrodynamic models. If a hydrodynamic model does not support the WASP linkage it is relative straightforward to create a hydrodynamic linkage file (see Appendix X for file format). The hydrodynamic models that currently support the WASP7.x file format are: EFDC (three dimensions), DYNHYD (one dimension branching), RIVMOD (one dimension no branching), CE-QUAL-RIV1 (one dimension branching), SWMM/Transport (one dimension branching), SWMM/Extran (one dimension branching)

4.1.5 Solution Technique

The user now has the ability to select the model solution technique to be used by the water quality module during the simulation. Currently there are 3 solution techniques that can be selected: 1) Euler – which is the traditional solution technique that has been in WASP since its inception, 2) COSMIC Flux Limiting – this solution technique is typically used when WASP is linked to multi-dimensional hydrodynamic models like EFDC, 3) Runge-Kutta 4 step solution technique used for diurnal simulations.

4.1.6 Time Step Definition

Starting with WASP Version 7.3 the user no longer has control over the computational time step. Time step optimization routines have been refined to the point where the model can determine what the most appropriate time step should be used next. This

assures the most efficient run time as well as minimizing numerical dispersion caused by too small of a time step. While the user can not set the time step directly, they do have some control over what would be an acceptable time step.

Fraction of Maximum Time Step - This dialog box specifies what fraction of the model calculated time step will be used for the next time step. Its primary purpose is aid the user in keeping the model stable. The default is 0.9 (or 90%) of the optimal time step.

Maximum Time Step - This specifies the maximum time step that will be used. If the time step optimizer calculates a time step larger than this value, this value will be used. This could be important in constraining the time step for diurnal or daily calculations.

Minimum Time Step - This specifies the minimum time step that will be used. The default minimum time step is defined in the model as 0.0001 days. Use this dialog to raise the minimum time step.

4.2 Segments screen

In the Segments screen, the user must enter a row for each segment in the model network (e.g., by pressing “Insert,” by pressing the down arrow from the bottom row or by copying from spreadsheet). For each segment, the user must enter a minimum amount of information, which depends on the selected flow option. For the Net Flows and Gross Flows options, the user must enter Depth (i.e. “Depth Multiplier”), Segment Type, Bottom Segment, and either Volume or Length and Width. If an input segment volume is 0, then WASP will calculate an initial volume as the product of Length, Width, and Depth. For the 1-D Network Kinematic Wave option, the user must enter Depth, Segment Type, Length, Width, Slope, and Bottom Roughness. Some default values are provided, including Description (“Wasp Segment”), Segment Type (“Surface”), Bottom Segment (“none”), Depth Exponent (0.6), Minimum Depth (0.001 m), and Bottom Roughness (0.05).

4.2.1 Description

Descriptive segment names may be entered, but are not required. If the user wants to paste segment descriptions/names that include spaces, the segment description must be placed in quotes (i.e. “I 20 Bridge”).

4.2.2 Volume

Segment volumes [m^3] should be specified when using the Net Flow or Gross Flow options. If a segment volume is not entered (or is 0), then WASP will calculate that volume from specified segment length, width, and depth.

For the 1-D Network Kinematic Wave option, input volumes are only used for benthic segments. WASP calculates initial water column segment volumes from length, width, and depth under initial flow conditions.

When using the hydrodynamic linkage flow option, initial water column volumes are read from the external hydrodynamic file. Only benthic segment volumes must be entered.

Segment	Description	Volume	Velocity Multiplier	Velocity Exponent	Depth Multiplier	Depth Exponent	Segment Type	Bottom Segment	Length	Width	Minimum Depth	Slope	Bottom Roughness
1	Upstream with Dam	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.5000	0.0000	0.0400
2	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
3	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
4	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
5	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
6	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
7	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
8	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
9	Wasp Segment	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400
10	Downstream	0	0.0000	0.0000	0.2890	0.4500	Surface	None	1000.0000	2.0000	0.0000	0.0010	0.0400

Figure 7 - WASP Segment Definition Screen

4.2.3 Velocity (multiplier and exponent)

The velocity hydraulic multipliers $[(\text{m}/\text{sec}) / (\text{m}^3/\text{sec})]$ and exponents should be specified only when using the Net Flow or Gross Flow options. For the Kinematic Wave option, the velocity multipliers and exponents are internally calculated from the input depth multipliers and exponents, and the input width. Any input velocity multiplier or coefficient will be ignored when using this option.

4.2.4 Depth (multiplier and exponent)

The depth hydraulic multipliers $[\text{m} / (\text{m}^3/\text{sec})]$ and exponents should be specified when using the Net Flow, Gross Flow, or Kinematic Wave options. Depth multipliers are required for all segments. For benthic segments, the depth multipliers are interpreted as segment depths [m].

For the Net Flow and Gross Flow options, the depth multipliers and exponents are used along with initial segment flows to calculate initial segment depths. If a depth multiplier is left at 0, it is reset internally to 1.0 and a message is issued to the screen. If a depth exponent is left at 0, then the depth multiplier is equal to the initial segment depth [m]. During simulations using these descriptive flow options, changing flows do not directly change segment depths, even if the hydraulic exponent is nonzero. Depths are recalculated along with volumes based on flow continuity. If a segment outflow continuity multiplier is equal to the inflow continuity multiplier, then changing flows will not alter that segment's volume or depth.

For free-flowing segments in the 1-D Network Kinematic Wave option, the depth multiplier is taken to be the cross-sectional average segment depth under average flow conditions [m]. The depth exponent is a value generally between 0.3 and 0.6. If a segment depth exponent is left at 0, a rectangular cross-section is assumed and the exponent is reset internally to 0.6. The average depths and depth exponents are used along with segment widths, slopes, and roughness factors to calculate consistent hydraulic coefficients, which are then used to calculate segment depths under initial flow conditions. During simulations, changing flows directly change hydraulic depths based on the hydraulic coefficients. Total segment depth is equal to the hydraulic depth plus the user-input zero-flow minimum depth.

For the ponded segments in the 1-D Network Kinematic Wave option, the depth multiplier is taken to be the initial hydraulic depth. During simulations, depths are

recalculated along with volumes based on flow continuity. Total segment depth is equal to the calculated depth plus the minimum depth, which is set internally to 0.001 m.

4.2.5 Segment Type

Segment type is entered using a pick list. Four segment types are available: “Surface,” “Subsurface,” “Surface Benthic,” and “Subsurface Benthic.” The default segment type is “Surface,” which represents upper water column segments in contact with the atmosphere. “Subsurface” represents underlying water column segments. “Surface Benthic” represents the upper benthic sediment segments in contact with the water column. “Subsurface Benthic” represents underlying benthic segments.

4.2.6 Bottom Segment

Bottom segment is the segment immediately underneath the current segment. The bottom segment is entered by typing in the segment number or by using a pick list. If no segments are underneath the current segment, then the bottom segment is designated “none.”

4.2.7 Length

Segment length [m] is the bottom length along the center of the flow line from the upstream end to the downstream end of the segment.

4.2.8 Width

Segment width [m] is the top width averaged along the length of the segment. If no input volume is specified, then width is used along with length and depth (multiplier) to calculate an initial volume.

For the 1-D Network Kinematic Wave option, width should be specified for average flow conditions. Average widths are used along with average depths, depth exponents, slopes, and roughness coefficients to back-calculate a consistent set of hydraulic coefficients.

4.2.9 Minimum Depth

Minimum depth [m] is the average segment depth under zero-flow conditions, used only in the 1-D Network Kinematic Wave option. If this cell is left blank, a default minimum depth of 0.001 m is assigned internally. Total depth is hydraulic depth plus minimum depth.

At present, the minimum depth field is also used to input bottom elevations for the dynamic flow option. Bottom elevations are negative values representing the vertical distance from the segment bottom (cross-sectional average) to the bottom of the downstream control segment, which is either a weir or a boundary.

4.2.10 Slope

Segment slope [m/m] is the elevation drop divided by length averaged over the segment length. This is usually calculated as the upstream elevation minus the downstream elevation divided by segment length. Slope is used only in the 1-D Network Kinematic

Wave option. Segments with slope greater than 10^{-6} are considered free-flowing, and flow is computed using the kinematic wave equation. Segments with slope less than or equal to 10^{-6} are considered ponded, and flow is computed using the weir overflow equation.

4.2.11 Segment Roughness

Segment roughness is the Manning's roughness coefficient n . Roughness is used in the 1-D Network Kinematic Wave option for kinematic wave flow and dynamic flow segments. Roughness coefficients should usually be between 0.01 and 0.15. If a coefficient of 0 is input for a free-flowing segment, WASP will reset the coefficient to 0.05 and issue a message to the screen.

4.3 Flows screen

The Flows screen is used to define advective transport, including surface water and pore water flow, as well as solids settling and resuspension, precipitation and evaporation. The Flows screen is also used to define downstream boundary elevations and two-dimensional channel networks for the Dynamic Flow option.

The flow input screen is a complex screen that contains four tables (Figure 8). The upper left quadrant is used to select the transport field, such as "Surface Water" flow. For each transport field selected, the upper right quadrant is used to define a set of transport functions, including upstream and tributary inflows. For each transport function, the bottom two quadrants are used to define the flow path and the flow time function.

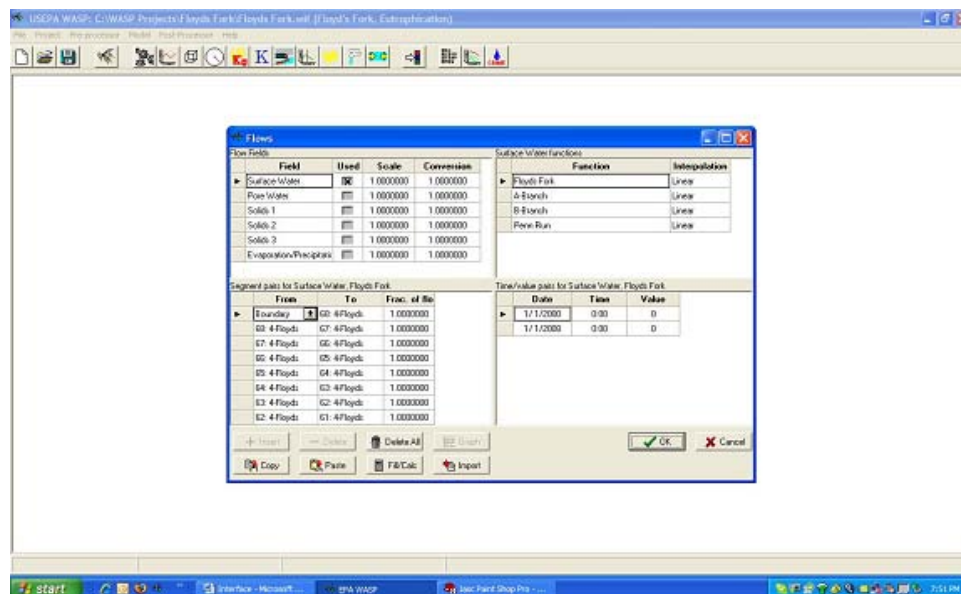


Figure 8 - Flows screen

For surface water and pore water transport, the upstream inflow, each tributary inflow, pore water inflow, and any flow withdrawals must be described by continuity path functions and inflow time functions. An example is shown in Figure 9.

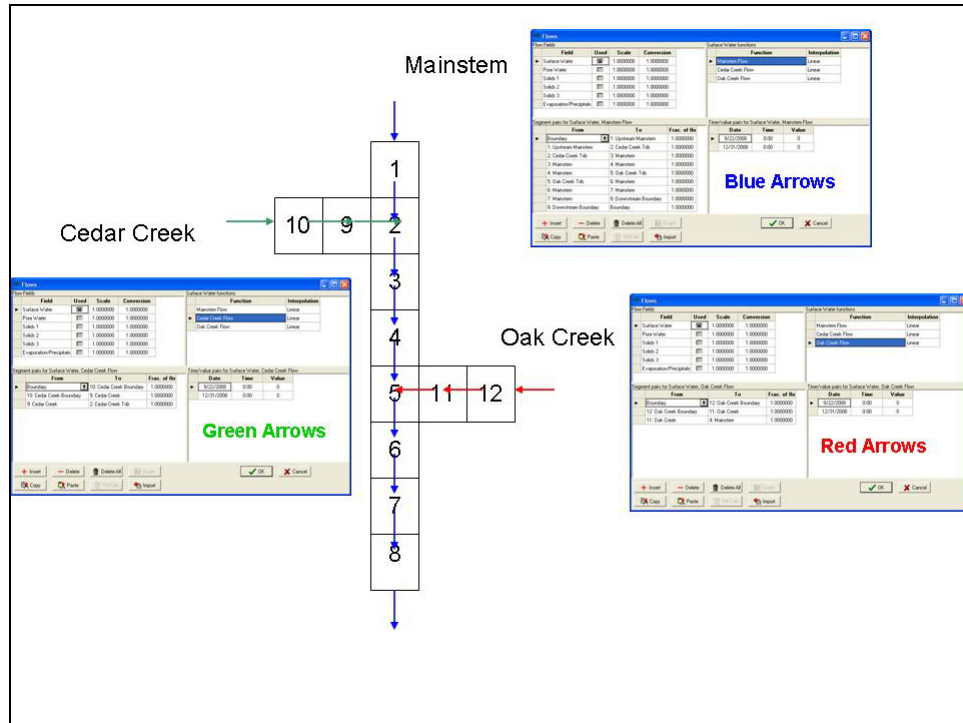


Figure 9 - Example WASP flow input

4.3.1 Flow Field

The transport field must be selected. Six transport fields are available:

1. Surface Water – This transport field is used to describe surface water flows. These flows transport both the particulate and dissolved fractions of a constituent. If the user has selected the hydrodynamic linkage option they will not be able to enter information here.
2. Pore Water – This transport field is used to describe pore water flows. These flows transport only the dissolved fraction of a constituent.
3. Solids 1 – This transport field is used to describe solids type 1 settling and resuspension. These flows transport only the particulate fraction of a constituent that is mapped to solid type 1 in the Systems screen.
4. Solids 2 – This transport field is used to describe solids type 2 settling and resuspension. These flows transport only the particulate fraction of a constituent that is mapped to solid type 2 in the Systems screen.

5. Solids 3 – This transport field is used to describe solids type 3 settling and resuspension. These flows transport only the particulate fraction of a constituent that is mapped to solid type 3 in the Systems screen.
6. Evaporation/Precipitation – This transport field subtracts/adds water from the model network. No constituent mass is added, removed, or transported.

Scale Factor – The scale factor for a transport field multiplies all flows associated with that field by the input value. This is generally used to scale flows in sensitivity tests. The default value is 1.0.

Conversion Factor – The conversion factor for a transport field multiplies all flows associated with that field by the input value. This is generally used to adjust input flow units to the internal units of m^3/sec . If flows are specified in ft^3/sec , the conversion factor should be 0.02832. The default value is 1.0.

4.3.2 Flow Function

The user can define several flow functions for the selected transport field. Each flow function must have its own flow path function (lower left table) and flow time function (lower right table). Normally, a Flow Function defines a discrete inflow, such as upstream flow, tributary flow, or pore water flow. Special flow functions are also used in conjunction with the Dynamic Flow option to define downstream boundary elevations or two-dimensional (x-y) channel networks.

To insert a flow function, first highlight the Surface Water flow field in the upper left table, then move the cursor to the upper right quadrant and click on the insert button. The resulting flow function cell, labeled “Flow Function,” can be edited to provide a descriptive name.

To insert additional flow functions, either click on “insert” or highlight the last flow function and press the down arrow. To delete a flow function, select the function by highlighting the row and click on the delete button. Deleting a flow function will delete the corresponding flow path function (lower left table) and flow time function (lower right table).

Function Name – When a flow function is inserted, it is given the default name “Flow Function.” The Function cell can be edited to provide a descriptive name, such as “upstream inflow,” “tributary inflow,” “downstream elevation,” or “channel network.”

Interpolation Option – The default interpolation option for the flow time function associated with a flow function is “Linear.” This can be changed to “Step” to provide for a step function. To change options, click in the Interpolation cell, press the down arrow, and select the interpolation option for this flow function.

Once a flow function is selected and named, the user must define the associated flow path function and flow time function. Be sure that the correct flow field and flow function are highlighted before entering these next screens.

4.3.3 Flow Path Function

The flow path function traces this flow from its point of entry into the model network to its point of exit either from the model network or to an alternate pathway associated with another flow function. The flow path consists of a set of rows, corresponding to segment interfaces. Each row will have a set of segment pairs and a fraction of flow multiplier.

Segment Pairs - The segment pairs consist of a “From” segment and a “To” segment, and define the direction of flow across this segment interface. Either the “From” or the “To” segment can be defined as “Boundary.” Normally the first row will define the inflow from “Boundary” to the upstream segment and the last row will define the outflow from the downstream segment to “Boundary.” If this flow path is a tributary, then the last row will define the outflow from the downstream tributary segment to a segment in another tributary or the main stem of the river.

Positive values of flow transport water and constituent mass in the defined direction from the first segment to the second segment. Negative flows transport water and constituent mass from the second segment to the first segment. For example, if “From” is segment 1 and “To” is segment 2, then negative values of flow in the time function will cause transport from 2 to 1. Note: While the kinematic wave option checks to make sure that all flow paths are ultimately connected to outflows, neither the preprocessor nor the model can assure that the segments are connected properly. Connectivity is the responsibility of the user.

Fraction of Flow - The fraction of flow column defines what fraction of the total flow in this pathway moves between these segment pairs. For surface water flow, the fraction of flow is normally 1.0. This allows the user to split flows from one segment into two or more downstream directions. This can be used to define diverging and converging flows, but must be used carefully. The sum of all fractions entering each segment must normally equal the sum of all fractions leaving. If the sum is greater than 1.0, then that segment’s volume will continually increase. If the sum is less than 1.0, then that segment’s volume will continually decrease; if the volume reaches 0, the simulation will end badly.

Note for downstream boundary elevation – If a downstream boundary elevation function is being defined for a dynamic flow network, then the flow path should consist of a single segment pair from “Boundary” to the downstream segment number. The fraction of flow should be set to 0.

Note for channel network – If a two-dimensional channel network is being defined for the Dynamic Flow option, then a channel network must be defined by one or more special Flow Functions. In a channel network flow function, each segment pair defines a unique flow channel. The fraction of flow multipliers must be set to 0. Channel lengths and cross-sectional areas for each segment pair are read from the Exchanges screen (Section 4.4). Channel hydraulic radius is calculated internally as the average of the upstream and the downstream segment depths. Channel width is calculated internally as the cross-sectional area divided by hydraulic radius. The channel network must not include any boundaries. Channels connect two segments within the model network. WASP distinguishes channel networks from traditional flow paths by the absence of

boundaries and flow path multipliers of 0. The flow time function associated with a channel network is not used, and flow values can be left at 0.

4.3.4 Flow Time Function

The flow time function is a table consisting of dates, times, and inflow values [m^3/sec]. Each row in the table represents a single point in time. During a simulation, inflows are interpolated between these points based on the flow function interpolation option selected (see Section 4.3.2). At least two rows must be entered in the flow time function to allow for interpolation.

Date – As with other WASP time functions, the date must be entered as mm/dd/year (e.g., 01/01/2004). The first date in the time function should correspond with the Start Date specified in the Data Set Screen (Section 4.1.2). The last date in the time function normally will correspond with the End Date.

Time – The time must be entered as hh:mm (e.g., 14:30).

Value – The inflow for this date and time is specified in units of [m^3/sec]. Different units can be used if a conversion factor is provided with the Flow Field (Section 4.3.1). Note that if a downstream boundary elevation function is being defined, then the value entered will be surface elevation [m].

The time function table allows the user to enter time variable flow information. For constant flows, two rows should be specified with the simulation start and end dates, and the constant flow value. The user can enter the information by hand, paste in from a spreadsheet, or query in from database/spreadsheets.

4.4 Exchanges screen

The Exchanges screen is used to define dispersive transport, including surface water and pore water mixing. The Exchanges screen is also used to define channel lengths and initial cross-sectional areas for channel networks.

The Exchanges input screen is a complex screen that contains four tables (Figure 10). The upper left quadrant is used to select the transport field, such as “Surface Water” flow. For each transport field selected, the upper right quadrant is used to define a set of transport functions, including lateral and longitudinal dispersion functions that provide data for any channel networks defined in the Flow functions (Section 4.3.3). For each exchange function, the bottom two quadrants are used to define the exchange path and the exchange time function.

The screenshot shows the 'Exchanges' window with the following data:

Field	Used	Scale	Conversion
Surface Water	<input checked="" type="checkbox"/>	1.0000000	1.0000000
Pore Water	<input type="checkbox"/>	1.0000000	1.0000000

Function	Interpolation
left side	Linear
right side	Linear
transverse	Linear

Segment one	Segment two	Area	Distance
3: Wasp Seg	6: Wasp Segmen	200.0000000	1000.00000
6: Wasp Segmen	9: Wasp Segmen	200.0000000	1000.00000
9: Wasp Segmen	12: Wasp Segme	200.0000000	1000.00000
12: Wasp Segme	15: Wasp Segme	200.0000000	1000.00000
15: Wasp Segme	18: Wasp Segme	200.0000000	1000.00000
18: Wasp Segme	21: Wasp Segme	200.0000000	1000.00000
21: Wasp Segme	24: Wasp Segme	200.0000000	1000.00000
24: Wasp Segme	27: Wasp Segme	200.0000000	1000.00000
27: Wasp Segme	30: Wasp Segme	200.0000000	1000.00000

Date	Time	Value
1/1/2004	0:00	0
1/10/2004	0:00	0

Buttons at the bottom: + Insert, - Delete, Delete All, Graph, Copy, Paste, Fill/Calc, Import, OK, Cancel.

Figure 10 - Exchanges Screen

4.4.1 Exchange Field

The exchange field must be selected. Two transport fields are available:

1. Surface Water – This transport field is used to describe surface water mixing. These turbulent flows transport both the particulate and dissolved fractions of a constituent.
Pore Water – This transport field is used to describe pore water mixing. These flows transport only the dissolved fraction of a constituent.

Scale Factor – The scale factor for a transport field multiplies all exchanges associated with that field by the input value. This is generally used to scale exchanges in sensitivity tests. The default value is 1.0.

Conversion Factor – The conversion factor for a transport field multiplies all exchanges associated with that field by the input value. This is generally used to adjust input dispersion coefficient units to the internal units of m^2/sec . If dispersion coefficients are specified in cm^3/sec , the conversion factor should be 0.0001. The default value is 1.0.

4.4.2 Exchange Function

The user can define several exchange functions for the selected transport field. Each exchange function must have its own exchange path function (lower left table) and dispersion time function (lower right table). Normally, an Exchange Function defines a type of exchange, such as lateral or longitudinal dispersion or surface water-pore water exchange. Exchange functions are also used in conjunction with the Dynamic Flow option to define channel lengths and initial cross-sectional areas for two-dimensional (x-y) channel networks.

To insert an exchange function, first highlight the Surface Water field in the upper left table, then move the cursor to the upper right quadrant and click on the insert button. The resulting flow function cell, labeled “Exchange Function,” can be edited to provide a descriptive name.

To insert additional exchange functions, either click on “insert” or highlight the last flow function and press the down arrow. To delete an exchange function, select the function by highlighting the row and click on the delete button. Deleting an exchange function will delete the corresponding exchange path function (lower left table) and dispersion time function (lower right table).

Function Name – When an exchange function is inserted, it is given the default name “Exchange Function.” The Function cell can be edited to provide a descriptive name, such as “lateral dispersion,” “longitudinal dispersion,” “pore water exchange,” or “channel network.”

Interpolation Option – The default interpolation option for the exchange time function associated with an exchange function is “Linear.” This can be changed to “Step” to provide for a step function. To change options, click in the Interpolation cell, press the down arrow, and select the interpolation option for this exchange function.

Once an exchange function is selected and named, the user must define the associated exchange path function and exchange time function. Be sure that the correct exchange field and exchange function are highlighted before entering these next screens.

4.4.3 Exchange Path Function

The exchange path function specifies a set of dispersive exchange flows. The function consists of a set of rows, corresponding to segment interfaces (or “channels” in a channel network). Each row will have a set of segment pairs, a cross-sectional area, and a characteristic mixing length.

Segment Pairs – Each discrete exchange pathway is defined by a set of two segments between which exchange flows occur. Either “Segment One” or “Segment Two” can be

defined as “Boundary.” Neither the preprocessor nor the model can assure that the segments are connected properly. Connectivity is the responsibility of the user.

Cross-Sectional Area, m^2 - Cross-sectional areas are specified for each dispersion coefficient, reflecting the area through which mixing occurs. These can be surface areas for vertical exchange, such as in lakes or in the benthos. Areas are not modified during the simulation to reflect flow changes.

Mixing Length, m – Mixing lengths or distance are specified for each dispersion coefficient, reflecting the characteristic length over which mixing occurs. These are typically the lengths between the center points of adjoining segments. A single segment may have three or more mixing lengths for segments adjoining longitudinally, laterally, and vertically. For surficial benthic segments connecting water column segments, the depth of the benthic layer is a more realistic mixing length than half the water depth.

Note for channel network – If a two-dimensional channel network is being defined for the Dynamic Flow option, then cross-sectional areas and mixing lengths entered here will be assigned to corresponding channels defined in the Flow Path screen (Section 4.4.3).

4.4.4 Exchange Time Function

The exchange time function is a table consisting of dates, times, and dispersion coefficient values [m^2/sec]. Each row in the table represents a single point in time. During a simulation, dispersion coefficients are interpolated between these points based on the dispersion function interpolation option selected (see Section 4.4.2). At least two rows must be entered in the flow time function to allow for interpolation.

Date – As with other WASP time functions, the date must be entered as mm/dd/year (e.g., 01/01/2004). The first date in the time function should correspond with the Start Date specified in the Data Set Screen (Section 4.1.2). The last date in the time function normally will correspond with the End Date.

Time – The time must be entered as hh:mm (e.g., 14:30).

Value – The inflow for this date and time is specified in units of [m^2/sec]. Different units can be used if a conversion factor is provided with the Exchange Field (Section 4.4.1). Dispersive mixing coefficients may represent pore water diffusion in benthic segments, vertical diffusion in lakes, and lateral and longitudinal dispersion in large water bodies. Values can range from $10^{-10} m^2/sec$ for molecular diffusion to $5 \times 10^2 m^2/sec$ for longitudinal mixing in some estuaries.

The time function table allows the user to enter time variable exchange information. For constant exchanges, two rows should be specified with the simulation start and end dates, and the constant dispersion coefficient value. The user can enter the information by hand, paste in from a spreadsheet, or query in from database/spreadsheets.

5 - Stream Transport Model Outputs

A set of output display variables is defined for each water quality module. Calculated values of selected display variables are stored for every segment at every output time

interval through a simulation. Display variables are selected in the Output control screen. Variables checked in the “Output” box will be available to the WASP graphical post-processing software MOVEM. For each variable with a checked “CSV” box, WASP will produce a separate comma-delimited file containing output for all segments and all output times.

A subset of transport variables is included in the output for all of the water quality modules. These are listed in Table 3. Users are encouraged to explore patterns and relationships among these variables to better understand the dynamics controlling transport in their water body.

Table 3 - Transport output variables

Variable	Units	Module					
		Heat	Eutr	Adv Eutr	Toxi	Adv Toxi	Merc
Flow In	m ³ /sec		x	x			
Flow Out	m ³ /sec	x	x	x	x	x	x
Ave Segment Flow	m ³ /sec		x	x			
Water Velocity	m/sec	x	x	x	x	x	x
Volume	m ³	x	x	x	x	x	x
Depth	m	x	x	x	x	x	x
Surface Elevation	m						x
Surface Width	m	x		x			x
Residence Time	days	x	x	x	x	x	
Maximum Time Step	days	x	x	x	x	x	x
Actual Time Step	days	x	x	x	x	x	x

6 - References

Ambrose, R. B., J. L. Martin, and T. A. Wool, 2006. Wasp7 Benthic Algae - Model Theory and User's Guide. U.S. Environmental Protection Agency, Washington, DC, EPA/600/R-06/106 (NTIS PB2007-100139), 2006.

- Wool T.A., R.B. Ambrose, J.L. Martin, and E.A. Comer, 2001. The Water Quality Analysis Simulation Program, WASP6; Part A: Model Documentation. U.S. Environmental Protection Agency, Center for Exposure Assessment Modeling, Athens, GA.
- Ambrose, R.B., Jr., T.A. Wool, and J.L. Martin, 1993. The Water Quality Analysis Simulation Program, WASP5; Part A: Model Documentation. Internal Report Distributed by USEPA Center for Exposure Assessment Modeling, U.S. Environmental Protection Agency, Athens, GA.
- Ambrose R.B., T.A. Wool, J.P. Connolly, and R.W. Schanz, 1988. WASP4, A Hydrodynamic and Water Quality Model—Model Theory, User's Manual, and Programmer's Guide. EPA/600/3-87-039, U.S. Environmental Protection Agency, Athens, GA.
- Di Toro DM, J.J. Fitzpatrick, and R.V. Thomann, 1983. Water Quality Analysis Simulation Program (WASP) and Model Verification Program (MVP) - Documentation. Contract No. 68-01-3872, Hydrosience Inc., Westwood, NY, for U.S. EPA, Duluth, MN.
- Chapra, S.C. 1997. *Surface Water-Quality Modeling*, McGraw-Hill, New York, New York, 844 pp.
- Chapra, S.C. 2003. QUAL2K: A Modeling Framework for Simulating River and Stream Water Quality (Beta Version): Documentation and Users Manual. Civil and Environmental Engineering Dept., Tufts University, Medford, MA.
- Chapra, S.C. and G.J. Pelletier. 2004. QUAL2K: A Modeling Framework for Simulating River and Stream Water Quality, Version 1.3: Documentation and Users Manual. Civil and Environmental Engineering Dept., Tufts University, Medford, MA.
- Feigner, K.D. and H.S. Harris, 1970. Documentation Report – FWQA Dynamic Estuary Model. U.S. Department of Interior, Federal Water Quality Administration.
- Wool, T.A., R.B. Ambrose, and J. L. Martin. 2001. “The Water Analysis Simulation Program, User Documentation for Version 6.0,” Distributed by USEPA Watershed and Water Quality Modeling Technical Support Center, Athens, GA.

7 - Appendix 1: Derivation of Equations

7.1 Hydraulic Exponents for Kinematic Wave Flow

Manning's formula (Equation 19) provides the basis for deriving relationships among the hydraulic exponents. Rearranging terms gives cross-sectional area as a function of flow:

Equation 54

$$A = n^{3/5} S_0^{-3/10} B^{2/5} Q^{3/5}$$

Cross-sectional average depth (hydraulic radius) is given by:

Equation 55

$$d = \frac{A}{B} = n^{3/5} S_0^{-3/10} B^{-3/5} Q^{3/5}$$

Velocity is given by:

Equation 56

$$v = \frac{Q}{A} = n^{-3/5} S_0^{3/10} B^{-2/5} Q^{2/5}$$

Substituting Equation 3 for width as a function of flow gives:

Equation 57

$$d = n^{3/5} S_0^{-3/10} b^{mult-3/5} Q^{3/5(1-bexp)}$$

Equation 58

$$v = n^{-3/5} S_0^{3/10} b^{mult-2/5} Q^{2/5(1-bexp)}$$

Equation 57 and Equation 58 give the depth and velocity hydraulic exponents as a function of the width exponent:

Equation 59

$$d_{xp} = 0.6 (1-bexp) \quad v_{exp} = 0.4 (1-bexp)$$

Comparing the flow exponents confirms that the velocity exponent is 2/3 of the depth exponent, confirming Equation 9.

8 - Appendix 2: Model Verification Tests

Model verification tests were designed to assure that the equations are implemented correctly in the model code. Results are stored in separate folders at:

- \WASP7\QA\Stream Transport\4-Stream Kinematic Wave Flows\

Verification tests are outlined below. Results are detailed in a companion document.

8.1 Kinematic Wave Tests

8.1.1 Stream Transport Test 1

This series tests the kinematic wave flow routines in WASP using the Simple Toxicant module with intermediate slope and steady inflow with an example problem taken from Chapra, Example 14.6, pp. 253, 254. Results are compared with analytical solutions implemented in a spreadsheet, and by simple hand calculations.

- Test 1a – Upstream Inflow Only
- Test 1b – Upstream and Pore Water Inflow
- Test 1c – Upstream and Precipitation Inflow
- Test 1d – Upstream Inflow and Evaporation Outflow

8.1.2 Stream Transport Test 2

This series tests the kinematic wave flow routines in WASP using the Heat module with shallow slope and step changes in inflow. Results for each flow step are compared with analytical calculations calculated in a spreadsheet.

- Test 2a – Rectangular Cross-Section
- Test 2b – U Cross-Section
- Test 2c – V Cross-Section

8.1.3 Stream Transport Test 3

This series tests the kinematic wave flow routines in WASP using the Eutrophication module with shallow slope, U-shape cross-section, step changes in upstream inflow, and inflow or withdrawal at Segment 3.

- Test 3a – Constant Tributary Inflow
- Test 3b - Constant Flow Withdrawal

8.1.4 Stream Transport Test 4

This series tests the kinematic wave flow routines in WASP using the Mercury module with a branching stream system. A medium stream with moderate slope is connected to a small tributary with shallow slope.

- Test 4a – Step Inflows
- Test 4b - Long-Term, Variable Inflows

Test 4a specifies step changes in upstream and tributary inflows. Results for each flow step are compared graphically and to analytical solutions from a spreadsheet. Test 4b uses variable upstream and tributary inflows repeating in a 2-year pattern over a long simulation period. Results are examined graphically for stationary (repeating) output.

8.1.5 Stream Transport Test 5

This tests the kinematic wave flow routines in WASP using the Heat module with a diverging-converging stream system. A small river with steep slope diverges into two branches receiving 40% and 60% of the upstream flow. These branches converge downstream. This test uses step changes in the upstream inflow. Results for each flow step are compared graphically and to analytical solutions from a spreadsheet.

8.2 Weir Overflow Verification Tests

Model verification tests were designed to assure that the equations are implemented correctly in the model code. Results are stored in separate folders at:

- \WASP7\QA\Stream Transport\4-Ponded Weir Flows\

Tests are outlined below.

8.2.1 Weir Overflow Test 1 – Steady Flow

This tests the ponded weir overflow routine in WASP using the Mercury module with steady inflow and sequentially increasing weir heights. Results are compared with analytical calculations.

8.2.2 Weir Overflow Test 2 – Variable Flow

This tests the ponded weir overflow routine in WASP using the Mercury module with sequentially increasing inflow and sequentially increasing weir heights. Results are compared with analytical calculations.

8.2.3 Weir Overflow Test 3 – Long Term Dynamics

This tests weir overflow routines in WASP for long-term performance with variable inflows repeating in a 2-year pattern. Results are examined for long-term drift in depths, volumes, and velocities.

8.3 Dynamic Flow Verification Tests

Model verification tests were designed to assure that the equations are implemented correctly in the model code. Results are stored in separate folders at:

- \WASP7\QA\Stream Transport\4-Dynamic Flows\

Tests are outlined below.

8.3.1 Dynamic Flow Test 1 - Steady Backwater

This tests the dynamic flow routine in WASP with mild slope, steady inflow, and downstream pond with weir. Weir height is set to provide downstream ponded depth equal to the kinematic flow depth. Results are compared with output from an equivalent kinematic wave simulation.

8.3.2 Dynamic Flow Test 2 – Steady Flow, Elevation

This tests the dynamic flow routine in WASP with mild slope, steady inflow, and constant downstream boundary elevation. Results are compared with output from an equivalent DYNHYD simulation linked with WASP.

8.3.3 Dynamic Flow Test 3 – Variable Stream Flow

This tests the dynamic flow routine in WASP using the Mercury module with sequentially increasing flow. Results are compared with output from a DYNHYD simulation linked with WASP.

8.3.4 Dynamic Flow Test 4 – EFDC Stream

This tests the dynamic flow routine in WASP with mild slope, steady inflow, and constant downstream boundary elevation. Results are compared with output from an equivalent EFDC simulation linked with WASP.

8.3.5 Dynamic Flow Test 5 – Tidal Stream

This tests the dynamic flow routine in WASP with flat slope, no inflow, and sinusoidal tidal downstream boundary elevation. Results are compared with output from equivalent DYNHYD and EFDC simulations linked with WASP.

8.3.6 Dynamic Flow Test 6 – 2-D Tidal Stream

This tests the dynamic flow routine in WASP on a 2-dimensional network with flat slope, no inflow, and sinusoidal tidal downstream boundary elevation. Results are compared with output from an equivalent EFDC simulation linked with WASP.