



# Userguide for the Wetland Water Budget Model Tutorial

---

**PURPOSE:** This technical note provides step-by-step instructions for using a computerized tutorial to help set up, execute, and calibrate the Wetland Water Budget Model, and to help manipulate the model's output using a post-processing program.

**BACKGROUND:** The Wetlands Research Program has developed a water budget model which can be used to model hydrologic and hydraulic processes in wetlands, and is a useful tool for investigating wetland functions. The model is modular in design, allowing users to run only those modules important to their study. For example, if users are interested in surface water processes, they would run the surface water module. The model also has a horizontal groundwater flow module and a vertical processes module (for precipitation, canopy interception, infiltration, and evapotranspiration). Any combination of the modules can be selected by the user. The number of input files (and the amount of data) needed to run the model depends on which modules are selected. A user could quickly become confused over which input files are needed and what data are required. To eliminate the confusion, a screen-driven, PC-based tutorial was developed to help users select modules, identify all of the necessary input files, and to give users instructions for formatting the data in those input files. The program also provides instructions for calibrating and running the water budget model, and manipulating the model's output.

For more information about the water budget model (including the processes it models, applications of the model, and assessments of the model's capabilities), the tutorial program and a program for manipulating output data, see Walton and Chapman (1993).

**OVERVIEW:** The computerized tutorial for running the water budget model uses a sequence of screens to prompt users for information they need for their particular model application. The following is an overview of the screens.

**SCREEN 1:** Provides instructions for using the screens. To move the cursor around a screen, use the **TAB** or **ARROW** keys. To move from one screen to the next, use the **Page Up/Down** keys. Use the **Enter** key to make selections within screens and the **Control\_H** keys to obtain "Help" at any time.

**SCREEN 2:** Permits users to select one or more of the water budget model's modules for their particular application, and allows them to define a **project name** (less than 9 characters) which becomes the "root" name for all of the input and output files.

**SCREEN 3:** Identifies the necessary input files required for the particular application (set of modules) selected by the user. The necessary files are indicated by a "YES" appearing in the box next to the file name. Users can view examples of the format for a file's data by selecting (**Enter** key) the file while the cursor is positioned by the file name. The user can also get more information regarding data sources by pressing the **Control\_D** keys. The names of the output files that the water budget model will generate are identified by the word "OUT" next to the file name. The "MAKE

**FILE**" option will automatically generate a file list (called "**root**") which the water budget model uses to identify which files are input files and which are output files.

**SCREEN 4:** Tells the user how to setup the input data, calibrate and run the model, and manipulate the output data. When the user exits this screen, the tutorial ends. The user may also exit the tutorial at any time by pressing the **END** key.

The remainder of this TN concentrates on the module components of the tutorial.

**SURFACE WATER MODULE SELECTION:** By selecting the surface water module (from **SCREEN 2**) for their model application, users can simulate surface water processes such as channel flow, over-bank wetting and drying, remote-basin inflow, tidal forcing, local rainfall and wind forcing. **SCREEN 3** of the program will enable the following file names (i.e. a "YES" will appear next to the file names in **SCREEN 3**):

- root.OUT** - general output file to which model input and run information are output.
- root.PAR** - run parameter file where module selection; simulation start date, duration, time step; print initiation and interval; and echo print options are given.
- root.GRD** - grid file where the number, location, size, and shape of nodes and links; the linkage between nodes; and link elevation, inverse side slope, and type are defined.
- root.ELV** - contains the bottom and overbank elevations of each node.
- root.PRP** - defines link properties such as friction formulation, and weir and culvert coefficients.
- root.INI** - specifies the initial nodal stages or water elevations above the model datum.
- root.BCI** - specifies the upstream and downstream boundary conditions (e.g. stage, flow, loop rating, or stage-discharge).
- root.HBC** - time series of stages or surface water elevations at each head boundary node specified in file "root.BCI."
- root.FBC** - provides the time series of flows or discharges at each flow boundary node specified in file "root.BCI."
- root.MET** - contains daily meteorological inputs such as rainfall, temperature, solar radiation and wind speed and direction.
- root.HDS** - nodal head or elevation output file.
- root.FLO** - link flow or discharge output file.
- root.VOL** - nodal volume output file.
- root.VEL** - link velocity output file.

When in SCREEN 3, examples of the input files may be viewed by moving the cursor to the box next to the file of choice and pressing the ENTER key. A sample input file is displayed showing the free-field file format, file delimiters, and a description of each variable within a "Help" window at the bottom of the screen. While a sample input file is being displayed, pressing Control\_D will display brief information on possible data sources for the necessary file data. The "Make File" option generates a file called "root" that contains the file names of all the input and output files required for a surface water module simulation.

Note: While examining the data structure for "root.GRD," the user can press the Control\_F keys to display schematics of the geometry used to describe nodes or links. The figures will be displayed in conjunction with the appropriate line in the data file "root.GRD."

**VERTICAL PROCESSES MODULE SELECTION:** By selecting the vertical processes module (from SCREEN 2) for the model application, users can simulate processes such as canopy interception, ET, and infiltration. To prescribe inputs and boundary conditions for these processes, the following files are enabled (i.e. a "YES" will appear next to the file names on SCREEN 3).

- root.OUT** - general file to which input and run information are output.
- root.PAR** - run parameter file where module selection; simulation start date, duration, time step; print initiation and interval; and echo print options are specified.
- root.GRD** - grid file where the number, location, size, and shape of nodes and links as well as the linkage between nodes are defined.
- root.ELV** - bottom and overbank elevations of each node, and the elevations of the bottoms of each layer at each node.
- root.NOD** - canopy and soil types used in each node layer at each node.
- root.CAN** - canopy type in terms of drainage parameters and monthly values of leaf area index (LAI).
- root.SOL** - soil type in terms of moisture parameters, hydraulic conductivities, and the power function relationship to soil moisture tension.
- root.INI** - initial nodal water surface elevations and groundwater heads in each layer, with respect to the model datum.
- root.MET** - daily meteorological inputs such rainfall, temperature, solar radiation, and wind speed and direction.
- root.HDS** - nodal surface water head output file.
- root.GWH** - nodal groundwater heads output file.
- root.SMC** - nodal soil moisture content output file.
- root.SUM** - vertical water mass balance file.

When in SCREEN 3, examples of the input files may be viewed by moving the cursor to the file of choice and pressing **Enter**. A sample file will appear showing the free-field file format, file delimiters, and a description of each variable within a "Help" window at the bottom of the screen. While the sample input file is displayed, pressing the **Control\_D** keys will show some general information on possible sources for the necessary file data. The "**Make File**" option generates a file called "**root**" that contains the file names of all of the input and output required for a vertical processes module simulation.

**HORIZONTAL GROUNDWATER FLOW MODULE SELECTION:** By selecting the horizontal groundwater flow module, users can include that hydrologic process in their application. The module can be used alone to simulate depth-averaged groundwater flow, although it is important to remember that the module is based on variably saturated groundwater flow theory. More commonly, the module would be used with the vertical processes module (described above) to simulate three-dimension groundwater flow and surface exchanges. In order to prescribe inputs and boundary conditions for these processes, the files "**root.OUT**," "**root.PAR**," "**root.GRD**," "**root.ELV**," "**root.NOD**," "**root.SOL**," "**root.INI**," "**root.GWH**," and "**root.SMC**" are enabled (i.e. a "**YES**" will appear next to the file names in SCREEN 3) as described above for the vertical processes module. In addition, SCREEN 3 enables "**root.BCI**" which specified fixed head boundary conditions and the location of wells.

**MODEL EXECUTION:** Once the necessary input data files are created, the selected modules are executed by typing "**WWBM**" followed by **Enter** at the DOS prompt. (Note: the prompt must be in the directory where the swamp program resides or the directory must be in the PC Path [see DOS manual].) At the program prompt, enter the 'control' file name: "**root**" followed by **Enter**. The program will echo the input and output file names and grid manipulation operations as they are completed. During the model simulation, the program prints a message to the screen each time output data are written. When "**SIMULATION COMPLETED**" appears on the screen, execution ends.

**OUTPUT MANIPULATION:** A post-processing program (called **WWBMAID**) is available with the water budget model. The program allows users to customize the output data from the surface water module output files. After starting the **WWBMAID** program, the following data manipulation options are displayed:

1. Create file with specified node numbers
2. Compare computed and observed heads
3. Compare computed and observed flows
4. Select a node or link for graphics
5. Edit heads for initial conditions
6. Convert grid files to new datum and units
7. Hydroperiod statistics
8. Water surface profiles
9. Add more columns of data to file
10. Groundwater heads for graphics
11. Calculate head from soil moisture content
99. EXIT

Enter selection:

**CONCLUSION:** A computerized tutorial is available to help users setup, calibrate, and run the Wetland Water Budget Model that simulates wetland hydrologic and hydraulic processes. The tutorial also explains how to use a post-processing program to manipulate the model's output data. The tutorial program, the Wetland Water Budget Model, and the post-processing program are available through the WES Engineering Computer Programs Library, Attention Gloria Naylor at (601) 634-2581. The programs' reference number is 722-PD-R0008.

**REFERENCE:**

Walton, R. and Chapman, R. S. 1993. "Development of an Integrated Hydrologic and Hydrodynamic Numerical Model for Wetland Processes and Function Evaluation," prepared for Coastal Engineering Research Center, USAE WES, prepared by Ebasco Environmental, Bellevue, WA, and Ray Chapman and Associates, Vicksburg, MS.

**POINT OF CONTACT FOR ADDITIONAL INFORMATION:** Mr. Jack Davis, U.S. Army Engineer Waterways Experiment Station, ATTN: CEWES-CD-SE, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199, phone: (601) 634-3006, author.