WHI UnSat Suite Tutorial: SESOIL

Modeling Seasonal Transport and Fate of Volatile Organic Contaminants in the Unsaturated Zone

WHI UnSat Suite for Windows 95/98/NT/2000

The WHI UnSat Suite combines three popular unsaturated zone models in a powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. These models are:

SESOIL - a popular US EPA model which is capable of simultaneously modeling water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess contaminant fate and transport for regulatory requirements.

PESTAN (PESTicide **AN**alitycal**)** - a model for assessments of groundwater contamination by pesticides used in agricultural practice.

VS2DT (Variably **S**aturated **2D** flow and **T**ransport**)** - a finite difference numerical model for determining the fate of agricultural chemicals, landfill leachate, and accidental chemical spills in the unsaturated zone.

VLEACH (**V**adose Zone **LEACH**ing model) - a model for evaluating the fate of volatile organic contaminants in the unsaturated zone.

All these popular models are seamlessly integrated with the WHI UnSat Suite and each one is compiled and optimized to run as a 32-Bit, native Windows application.

The WHI UnSat Suite shares the majority of its features with the other WHI software product Visual HELP which, besides its direct function, landfill designing, can be used as an efficient tool for the assessment of soil surface water balance. The two products are connected functionally and logically and in many cases the joint use of two products in consulting projects is very beneficial. Both products are joined together in the software package called WHI UnSat Suite Plus. To learn more about Visual HELP, review the demo exercise for this product.

Learning Objective

In this demonstration exercise, you will use SESOIL as a tool for simulating the transport of benzene in the unsaturated zone and prepare a project report.

The instructions for this exercise are provided in a step-by-step format, which allows you to examine different features of UnSat Suite without having to complete the entire exercise.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

Type	- Type in the given word or value
select	- Click the left mouse button where indicated

Introduction to Visual HELP for Windows 95/98/NT 3
The HELP Model 3
History 3
New Features of Visual HELP version 2.1 4
System Requirements 4
Installing Visual HELP 4
Visual HELP Demonstration Exercise 4
How to Contact Waterloo Hydrogeologic Inc. 5
Demonstration Exercise 6
Learning Objectives 6
Terms and Notation 6
Starting Visual HELP 7
Module I: Creating a New Project 8
Selecting the Location 9
Selecting Units 12
Assigning Authors and Clients 12
Selecting an Existing Profile 13
Module II: Designing a Landfill Profile 15
Viewing the Default Profile 15
Editing the Layer Structure of the Landfill Profile 16
Resizing Layers 17
Deleting a Layer 18
Restoring a Layer 19
Splitting a Layer 19
Inserting a Layer 20
Editing Layer Parameters 22
Setting Initial Moisture Conditions & Runoff Parameters 24
Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting 26
Viewing Weather Data 27
Running Visual HELP 29
Viewing Results 29
Viewing the Output Graphs 29
Viewing Tables 31
Creating a Report 32

Introduction to Visual HELP for Windows 95/98/NT

The HELP Model

HELP (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile model used to test the effectiveness of landfill designs and also to predict landfill hydrologic processes and groundwater recharge rates. Now, HELP has become a requirement for obtaining landfill operation permits in the U.S.A. and is becoming the standard for landfill optimization around the world.

HELP is a quasi-two-dimensional hydrologic model and uses the following input data to create the landfill profile:

- Weather (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil (porosity, field capacity, wilting point, and hydraulic conductivity)
- Engineering design data (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can be multi-layered, consisting of a combination of natural (soil) and artificial materials (waste, geomembranes) with an option to install horizontal drainage. HELP also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

Built-in Database tools include:

- Weather Generator, a tool for synthetic generation for up to 100 years of daily values of precipitation, air temperature and solar radiation.
- Soil, waste and geomembrane database which contains parameters for 42 materials.

History

WHI has developed a Windows 95/98/NT interface for version 3.07 of the HELP model which was released in May 1998 under the name Visual HELP version 1.101. Visual HELP can be used as a pre- and post-processor for the HELP model, allowing you to perform all of the functions of the DOS version 3.08 of HELP. Visual HELP was presented at the trade show at SWANA's WASTECON 1998 and ISWA World Congress 1998 in Charlotte, North Carolina. Currently, more than 300 copies of this product are being used by consulting companies, government regulators and Universities in U.S.A., Canada, U.K., Germany, Australia, Sweden, and Hungary.

After running through this demonstration tutorial, we trust you will agree that Visual HELP is now the most complete and easy-to-use modeling tool for designing & optimizing landfills.

Visual HELP is also now available as part of the **WHI UnSat Suite Plus**. This ultimate suite of 1-D unsaturated flow models is comprised of Visual HELP, VS2DT, VLEACH, and PESTAN. Now you can design landfills, predict leachate & contaminat migration, and show degradation of pesticides in one package! For details about the **WHI UnSat Suite Plus,** please call (519) 746-1798 or visit our web page at www.flowpath.com.

New Features of Visual HELP version 2.1

Profile Viewing and Editing:

- A layer can now be split into two separate layers
- Layers are systematically numbered, and labelled in the Project Tree, which simplifies interpreation of the results

Weather Generating:

- New databases and GIS searching tools have been developed for the major regions of the world (more than 3000 weather stations)
- Data in NOAA format can now be imported automatically
- Diagnostics are provided to detect missing records in NOAA files

Output Presentation:

- Output units can be changed now without restarting the project
- Water Balance Tables have been added
- New tree-like interface structure allows the user to easily select results
- Report Generator (a new feature of Visual HELP) allows you to display, print and export to Microsoft Word project input settings and output graphs and tables.

System Requirements

To run Visual HELP, you will need the following minimum system configuration:

•Windows 95/98, or Windows NT 3.5.1 or later;

•IBM PC or compatible,

•Pentium 100 processor;

•CD-ROM drive for software installation;

•32 Mb of RAM;

•60 Mb of free hard disk space; and

• Minimum display resolution of 1024 x 768

The following fonts should be installed on your computer: MS Sans Serif, Arial, and Courier New.

Installing Visual HELP

The Visual HELP 2.1 demo must be installed on your computers hard disk. Please ensure that your system meets the minimum requirements before proceeding with the installation. To install, simply navigate to the Visual HELP page found on the Demo CD ROM and select the appropriate button.

Visual HELP Demonstration Exercise

We have included example data with the installation to help you familiarize yourself with the Visual HELP 2.1 interface. The examples are found as the default projects in Visual HELP. Since this is a demonstration installation, you are restricted to only running the default projects, however are free to browse the entire Visual HELP interface.

How to Contact Waterloo Hydrogeologic Inc.

If after reviewing the Visual HELP demo, you would like to contact Waterloo Hydrogeologic Inc.(WHI), please use the address below:

Waterloo Hydrogeologic, Inc.
180 Columbia Street West, Unit 1104
Waterloo, Ontario, CANADA
N2L 3L3
Phone: (519) 746-1798
Fax: (519) 885-5262

To purchase the product: contact the WHI Sales Department at sales@flowpath.com or visit our web page order form at: http://www.flowpath.com/order_form.html.

Please forward addtional comments or suggestions to the Visual HELP development team at:

Mikhail Gogolev, Product Manager mgogolev@flowpath.com

Dmitri Eidenzon, Chief Programmer deidenzon@flowpath.com

Demonstration Exercise

Learning Objectives

In this demonstration exercise, you will:

- Use Visual HELP as a tool for landfill design and hydrologic simulation;
- Use the Weather Generator to predict meteorological data for a twenty year period and use the predicted data as input to Visual HELP;
- Run the Visual HELP simulation, view the water balance of the landfill, assess the performance of the drainage and leachate collection system, and
- Prepare a report.

The instructions for this demo exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

The default landfill profile used in this exercise was designed in accordance with the EPA's recommendations. This design is also used in the official HELP 3.0 User's Manual. The layer properties for the default landfill profile were taken from the HELP database. The simulated landfill is located near Toronto, Ontario, Canada. The area of the landfill is 1 acre.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type- Type in the given word or value

select- Click the left mouse button where indicated

a - Press the **<Tab>** key

↵ - Press the **<Enter>** key

☞ - Click the left mouse button where indicated

☞☞- Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

NOTE: All theoretical parts of the demo tutorial are italicized

Starting Visual HELP

To start Visual HELP, you must have it installed on your hard disk. Once installed, click Windows **Start** button, choose **Programs**, navigate to the folder that contains Visual HELP, and click **Visual HELP**.

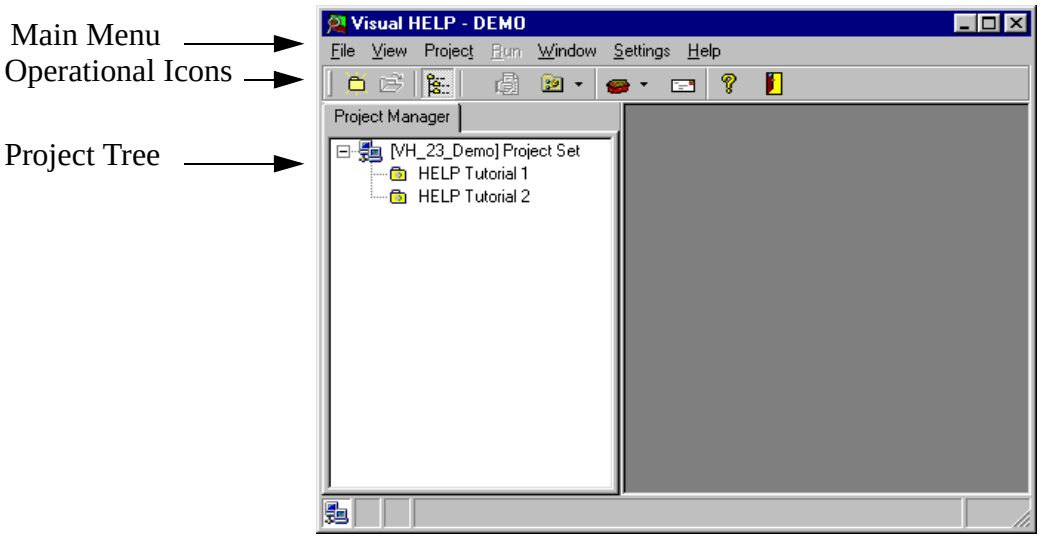


The installation program will also place an icon on your desktop, which you can also click to start the Visual HELP demo exercise.

An introductory screen will be displayed when you open Visual HELP



After the program is loaded, the main Visual HELP window will appear on your display:



The Visual HELP interface structure has been designed to help you navigate through the program with ease. The main menu is located at the top, similar to our other programs. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

The default projects **HELP Tutorial 1**, and **HELP Tutorial 2** are snapshots of a HELP Tutorial project taken at different stages of model development. Since the demo version does not allow you to save or edit new projects, these default projects have already been prepared for you to use. In the full working version of Visual HELP, all results of the editing process are saved within the same project.

Module I: Creating a New Project

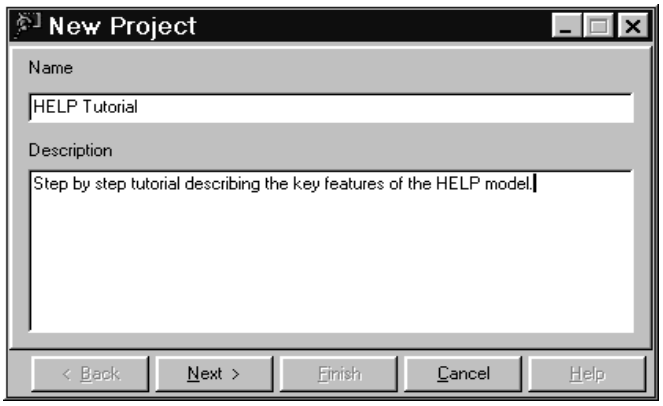
To create a new project, go the the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new Visual HELP project.

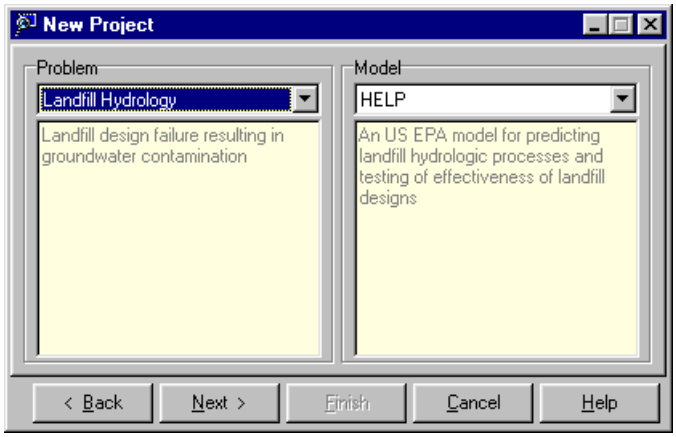
In the **Name** box type: **HELP Tutorial**.

In the Description box type: **Visual HELP step-by-step tutorial.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Problem class that should be used with the select Problem type.

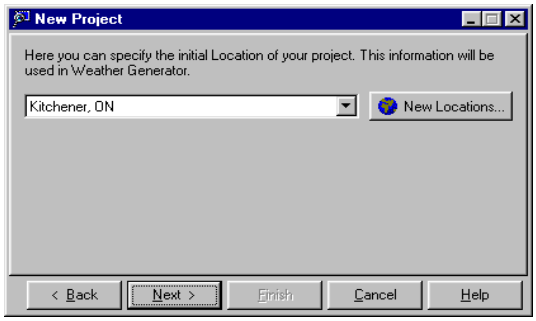


☞ **[Next]**

Once completed, the site location must be defined.

Selecting the Location

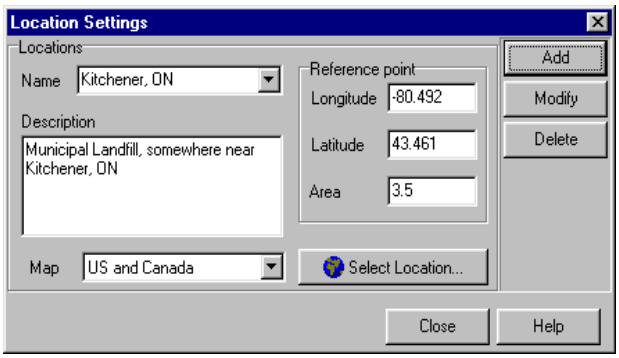
We will now select the location of our project using the GIS - WHI Locator. The following dialog box should already appear on your screen.



At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



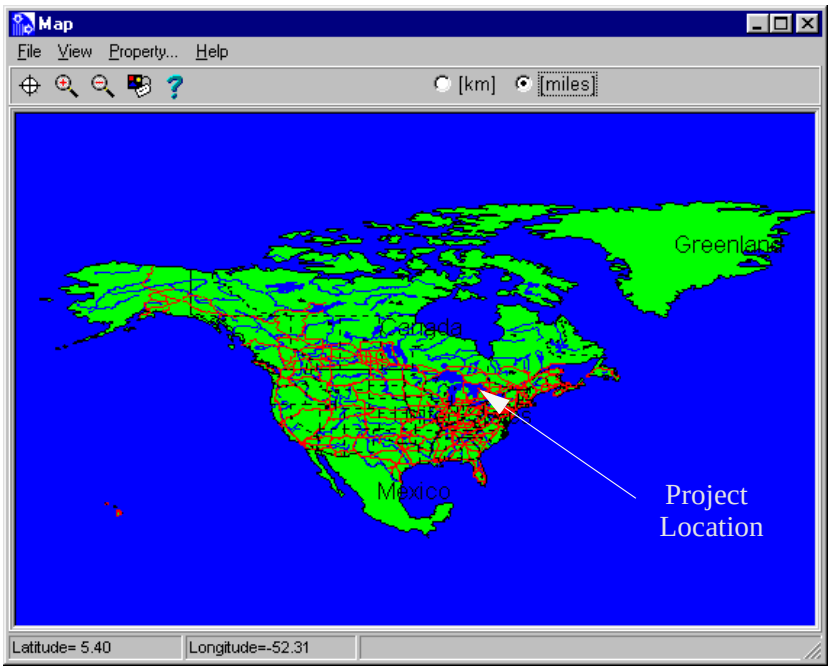
From the **Map** drop-down list,

☞ **North America,** to enable the WHI Locator, developed by WHI. The additoinal maps found in the list refer to various global regions that will help you with build projects from around the world!



☞ **[Select Location...],** to define our specific project location using the WHI Locator map.

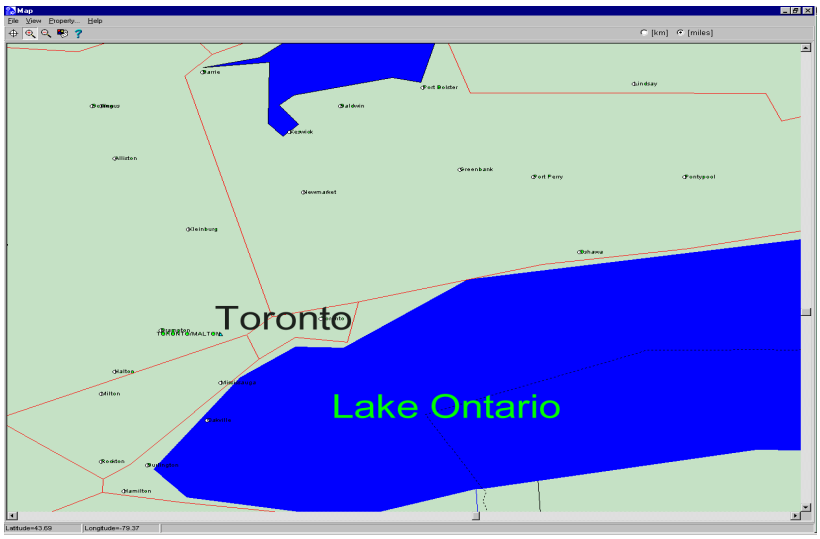
As shown below, a map of USA, Canada, and Mexico appears on your display:.



Now lets select Toronto, Ontario, Canada as our site location.

☞  , to zoom into our location.

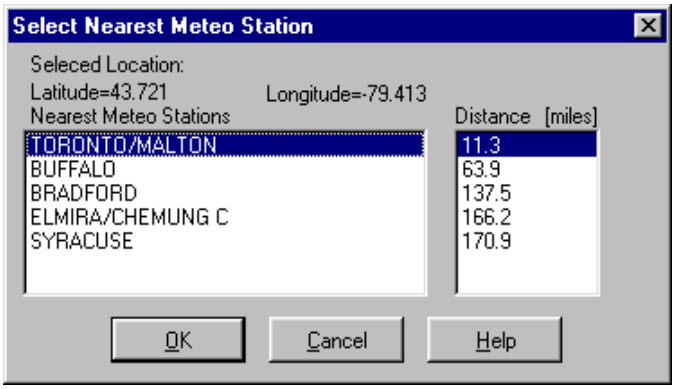
Move the mouse to the right of the region where the landfill project is located (in center of the Great Lakes region). The coordinates, seen in the bottom left of the screen, should read approximatelly Latitude:46.8, Longitude:-84.7. Press the left mouse button and stretch a zoom rectangle to the right of the Lake Ontario, release the button (approximate Latitude:43.3, Longitude:-76.2). After one or two zooms, figure below should resemble your screen.



Now select Toronto to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

Move the crosshair to the spot on the map where Toronto is located and click on it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you wish to calculate the distance in kilometers, simply click the appropriate option button at the top of the map.

In the "Select Nearest Meteo Station" dialog box,

☞ **[TORONTO/MALTON],** as the nearest weather station.

☞ **[OK]**

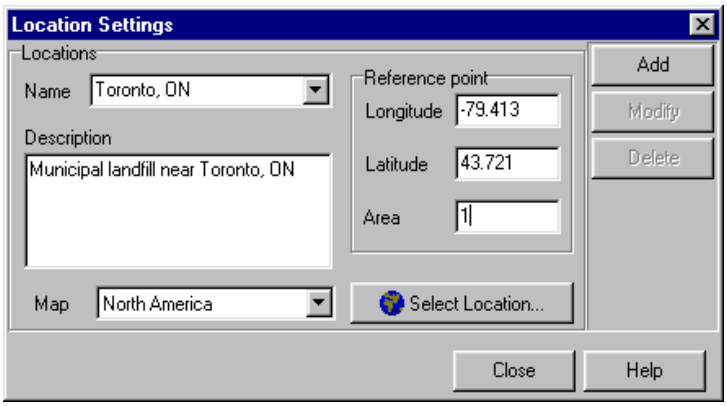
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: Toronto, ON., in the **Name** box.

Type: Municipal landfill near Toronto, ON., in the **Description** text box.

Type: 1 , in the landfill area text box: **1**.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database for future use.

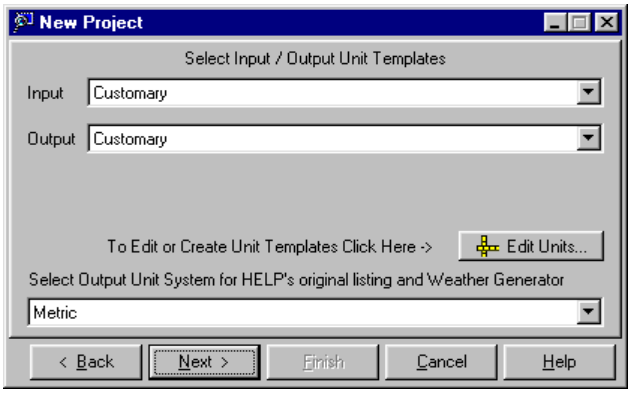
☞ **[Close]**

In the **New Project** dropdown list, select **Toronto, ON**.

☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.



The Visual HELP Unit Converter was developed to maximize the flexibility for data input and output unit conversion. To show you the range of possibilities, just view the list of available units for hydraulic conductivity, by opening the **[Edit Units]** option.



For this demo excercise we will specify that all input units will be in **Metric**, and output will be in **Customary** units.

From the **Input** drop-down list,

☞ **Metric**

From the **Output** drop-down list

☞ **Customary (in)**.

From the **Select Output Unit System for original listing and Weather Generator** list box,

☞ **Metric**

☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

All the project information will be presented, as seen in the figure below.



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings were specified, the **New Profile Wizard** will open.

Profiles and Multiple Profiles in Visual HELP. *For the purpose of hydrological simulation, a profile represents a part of a landfill that is assumed to have the same cross-section throughout. The profile contains all of the layers of the landfill that it represents. It may also contain details of engineering components, such as subsurface drainage, leachate recirculation systems, geomembranes, geonets, and composite liners. The surface of the profile can be sloped, which is typical for the peripheral parts of the landfill, or flat, which is typical for the central part of the landfill. The slope of the landfill usually mimics the shape of the waste body. At the time of landfill closure, the waste layer at the periphery of the landfill is usually sloped and the entire landfill site is capped by several layers, which form the landfill cap. Although drainage pipes are not usually installed in the landfill cap, permeable sand layers in the cap can transport water to the bottom drain or sump that surrounds the landfill. This process can be simulated with Visual HELP 2.1.*

A typical landfill can be represented as a set of profiles. This is done by creating several profiles in one project. This is useful because it allows you to use one profile to simulate the middle of the landfill, and several other profiles to simulate the edges, where the cross-section is tapering.

Selecting an Existing Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:

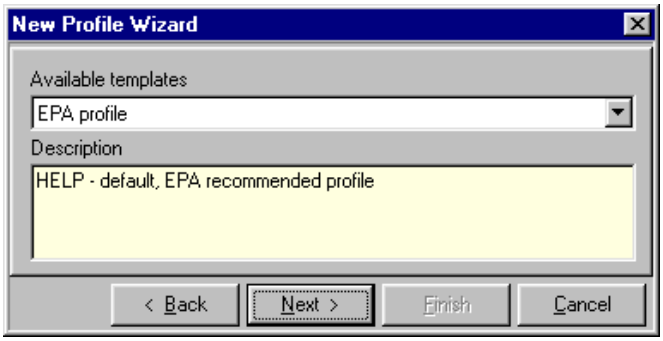


In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the default profile template.

☞ **[use existing profile template]**

☞ **[Next]**

The **Available templates** list will appear:



☞ **EPA Profile**

☞ **[Next]**

All the profile information will be presented:.



☞ **[Finish]** to load the profile.

The new project details will be added to the project tree. This concludes creating a new project using Visual HELP. We will now continue with designing the landfill profile.

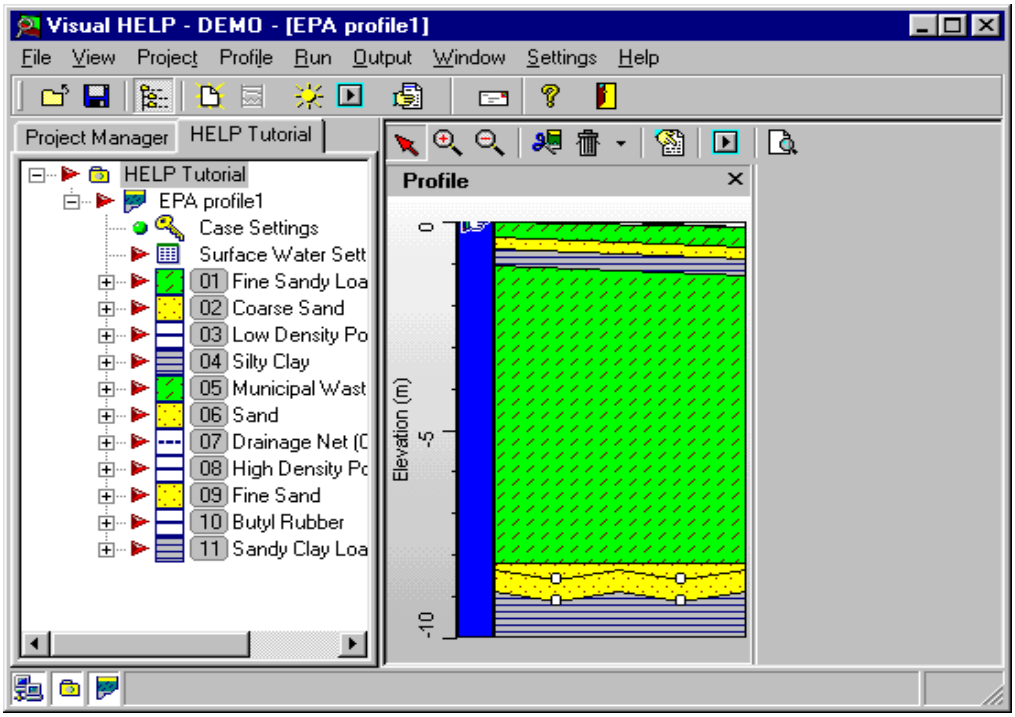
Module II: Designing a Landfill Profile

If you have skipped the steps in Module I and wish to start the Visual HELP demo exercise from this point,

☞☞ **HELP Tutorial 1,** from the Project Tree.

If you have run through Module I, you may continue with the project **HELP Tutorial.**

In both cases, the following picture will appear on your display.



Viewing the Default Profile

In the Profile View, located in the center part of the Visual HELP window, you will see the loaded default landfill profile with a depth scale.

A description of each landfill layer will appear when the mouse cursor is pointed at the layer.

As you can see, this profile consists of:

- a sloped landfill cap in the depth interval 0.0 - 1.2 m, consisting of fine sandy loam, coarse sand and silty clay
- a 7 m thick waste layer in the depth interval 1.2 - 8.2 m; and
- a double liner leachate collection and removal system in the depth interval 8.2-9.6 m.

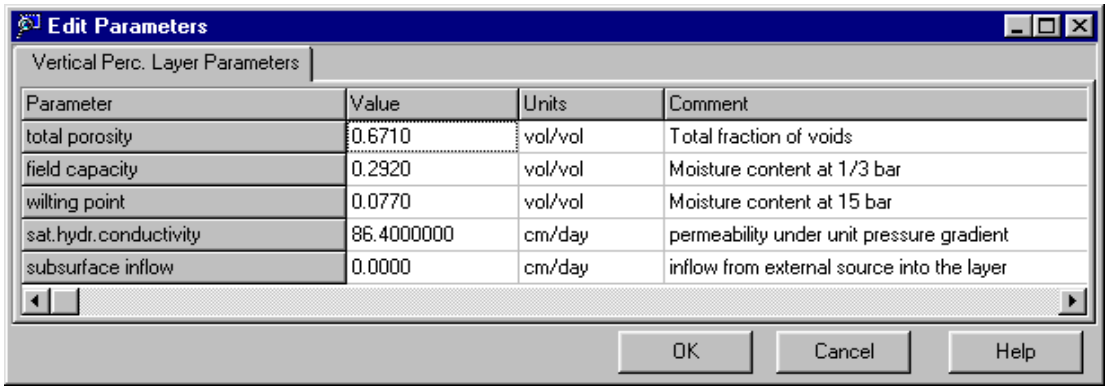
The profile construction can also be seen in the Project Tree View in the left part of the screen.

The Project Tree View is also used to view and modify the parameters of specific layers.

☞ **'+'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

☞☞ **Vertical Percolation Layer Parameters**. The **Edit Parameters** dialog box will appear.

The **Edit Parameters** menu allows you to modify the values of specific parameters.



☞ **[Cancel],** in the **Edit Parameter** dialog box to exit.

☞ **'-'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

As you can see, the Project Tree View can be used to easily access specific Profiles, Profile Layers, and Layer Parameters.

Now that you have become familiar with some viewing functions of Visual HELP, you will now learn how to edit the profile. In this part of the exercise we will:

- learn how to edit the layer structure of the landfill profile
- learn how to edit parameters of individual layers
- learn how to set initial moisture conditions and runoff parameters of the profile

Editing the Layer Structure of the Landfill Profile

To assist you in designing a landfill, Visual HELP offers many convenient tools for resizing, splitting, inserting and deleting layers. Visual HELP is a graphical user interface for the US EPA HELP model and, as such, obeys all conventions of the original HELP model.

The layers in the landfill are classified by the hydraulic function that they perform. Four categories of layers are available in original HELP:

•vertical percolation layers,

•lateral drainage layers,

•barrier soil liners, and

•geomembrane liners.

The topsoil and waste layers are generally vertical percolation layers. Sand layers above liners are typically lateral drainage layers; compacted clay layers are typically barrier soil liners. Geomembranes are typed as geomembrane liners. Composite liners are modeled as two layers. In the original HELP, geotextiles are not considered as layers unless they perform a unique hydraulic function.

In Visual HELP, taking into account that this class of landfill materials is growing extensively, geotextiles and geonets were defined as a separate category.

Although HELP allows a wide range of layer combinations, there are some basic rules and restrictions for the arrangement of layers in a profile that should be followed. These rules are obeyed by Visual HELP and you will get a warning if you try to delete or insert a layer incorrectly.

Resizing Layers

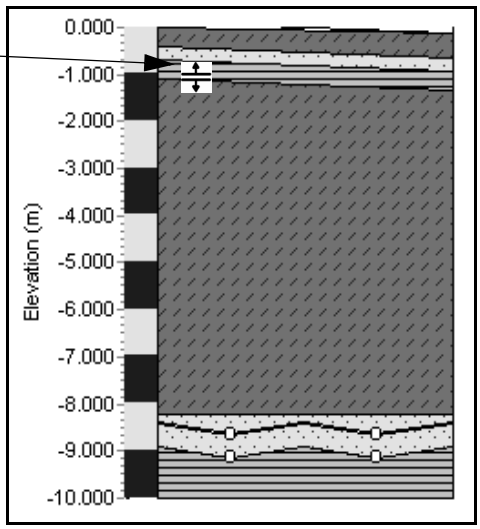
In Visual HELP, the layers are either resizable or non-resizable.

Resizable Layers *Layers built from soil and waste, grouped in the Vertical Percolation, Lateral Drainage, Barrier Soil design categories, are resizable.*

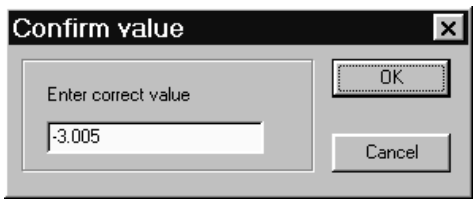
Non-Resizable Layers *Layers built from industrially produced materials with fixed thickness, grouped in the Geomembrane Liner and Geotextile and Geonet categories, are non-resizable.*

To test the resizing feature, move the mouse arrow on the **Municipal Waste (312 kg/cub.m)** layer in the profile view and move the arrow upwards to the top of the layer. As soon as the cursor encounters the layer's boundary, the cursor symbol changes:

New Cursor Symbol



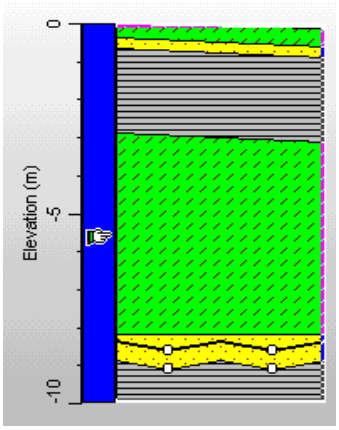
Press the left mouse button and, holding the mouse button, move the cursor downwards to approximately **-3.000** and release the mouse button. A **Confirm Value** dialog box will appear.



Type: -3.000 in the **Enter correct value** box.

☞ **[OK]**

The profile will change according to the new setting specified.



Note: ***Moving the layer's boundary resizes both layers which are separated by this boundary, however it does not change the total thickness of the profile. You can also change the thickness of an individual layer. This feature will be described later in this tutorial.***

Now resize the municipal waste layer back to its original size.

Place the mouse arrow on the **municipal waste** layer in the profile view, and move the arrow up until the cursor symbol changes.

Click and drag the boundary to approximately -1.200.

Type: -1.2, in the **Enter correct value** text box.

☞ **[OK]**

Deleting a Layer

We will now examine how to delete a layer.

☞ **Silty Clay**, in the Project Tree View.



.

Click the right mouse button. A shortcut menu will appear

☞ **Delete**

The layer **Silty Clay** will disappear from the Profile View. Notice an 'X' beside the silt clay layer to represent a deleted layer.

Restoring a Layer

Now let us restore the **Silty Clay layer**.

<Right click> the layer's name in the Project Tree View.

Click **Restore,** the original layer returns.

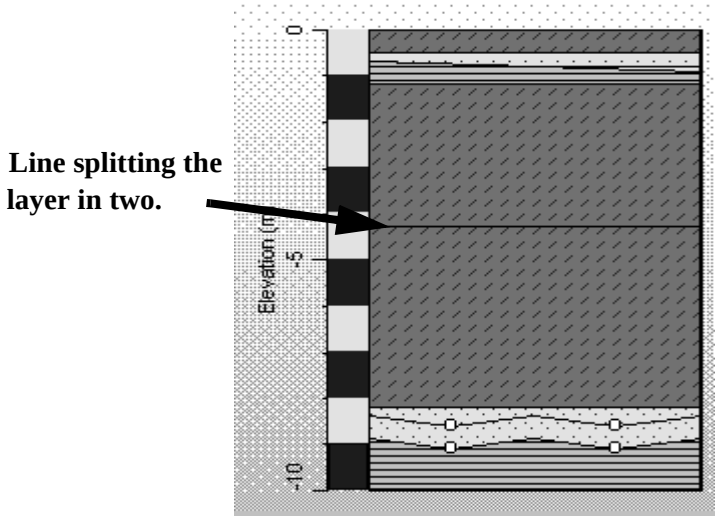
Splitting a Layer

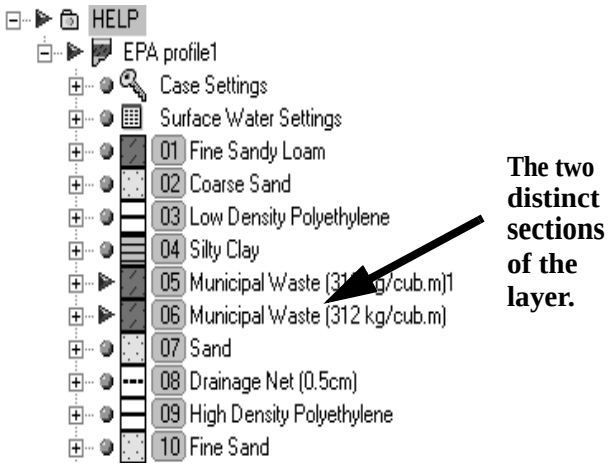
You can split a layer up into multiple sections and substitute materials for each section or assign different values of parameters for each section.

<right click> on the **Municipal Waste (312 kg/cub.m)** layer in the profile view.

☞ **Layer/Split**

A line will appear through the layer at cursor position and a new layer will appear in the project tree. Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.



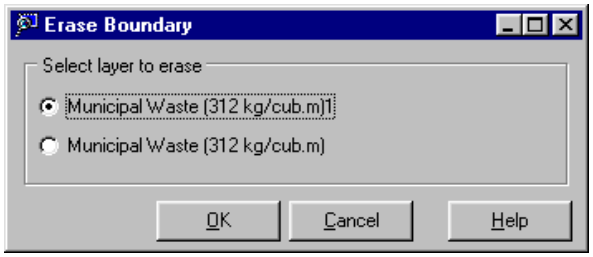


You may also substitute a material in the new layer or insert another layer between two separate parts of the layer. To return to the original profile construction, simply merge the two layers.

Place a cursor at the boundary between two parts of a layer and **<right click>**,

☞ **Merge Layers**.

In the **Erase Boundary** dialog box select the layer to erase: **Municipal Waste (312 kg/cub.m)1**



☞ **[OK]**.

Inserting a Layer

<right click> on **Sand**, in the Project Tree View

The following shortcut menu will appear



:

☞ **[Insert Layer]**

An empty **Profile Material Properties** dialogue box will appear. You can now specify the new material to insert.

Note: The new layer will always be inserted above the current layer.

Note: To insert the new layer inside the existing layer, split the layer first with the split function.

Material Category Click a category from the **Material Category** list. The list will be limited to those categories that are specified by the HELP Model only.

☞ **[HELP] Barrier Soil Liner.**

Material Click the appropriate material from the **Material** list. Each layer category is associated with several textures that can be used in the design of the layer. The texture number in Visual HELP corresponds directly to the texture number in the original DOS HELP.

☞ **[Clay Loam**]

Once the material is selected, the top and bottom settings are activated.

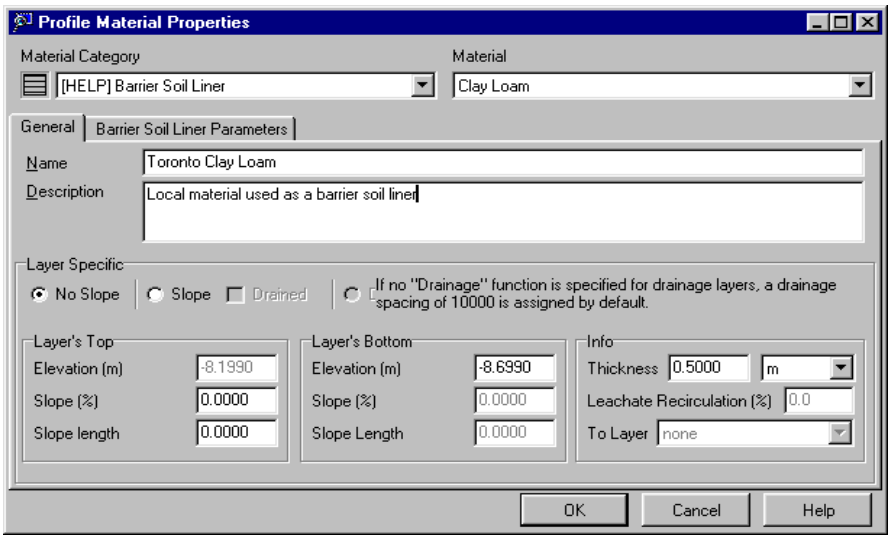
Name **Type: Toronto Clay Loam,** as the unique name for the material.

Description **Type: Local material used as a barrier soil liner**, in the comments field.

No Slope ☞ **No Slope**

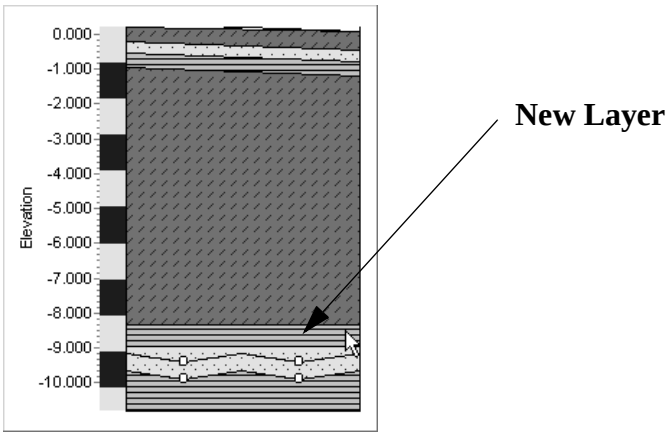
Type: 0.5 in the **Thickness** box.

Once completed, your **Profile Material Properties** dialogue box will appear as the figure below:



☞ **[OK]**

The new layer is now defined::



☞  icon, to save changes.

☞ **[OK],** to confirm the changes

The **Toronto Clay Loam** has now become part of the profile and you can modify it like the rest of profile layers.

To return to the original profile construction, which will be used for the rest of this tutorial, delete the recently inserted layer.

<right click> on **Toronto Clay Loam,** in the Project Tree View.

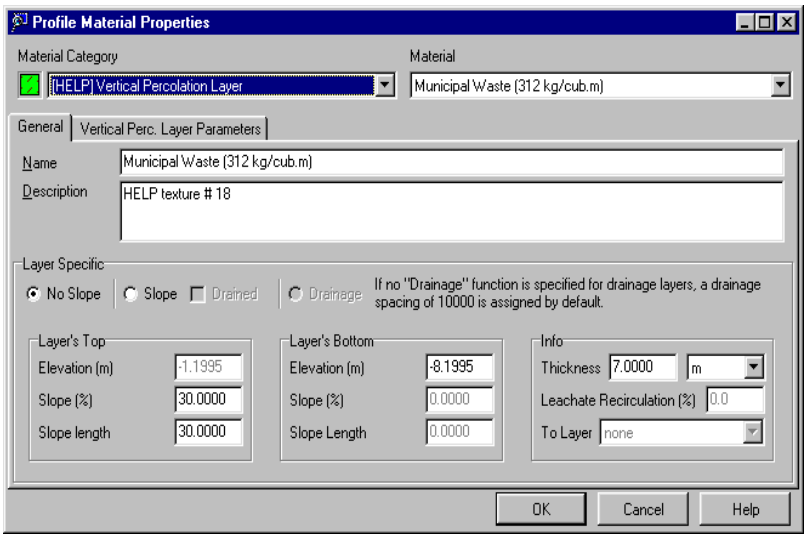
☞ **[Delete]**.

Editing Layer Parameters

The Visual HELP parameters of a landfill material depend on the material category.

☞ ☞ **Municipal Waste (312 kg/cub.m),** in the Project Tree.

The **Profile Material Properties** dialogue box will appear.



In this dialogue box, you can view and edit information about the layer including;

category, material texture, slope, top or bottom elevation, and thickness.

☞ **Vertical Perc. layer Parameters** tab to edit the material properties of particular layer.

☞ **Value** beside **total porosity**

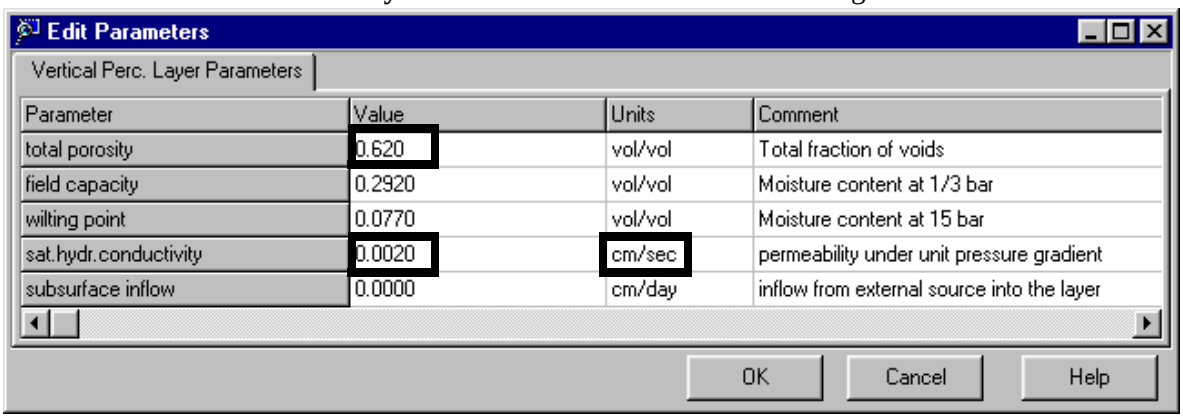
Type: 0.62

☞ **Units** beside the **sat. hydr. conductivity**, a drop-down list with available units will appear.

☞ **cm/sec**

Type: 0.002, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-] ,**beside **Municipal Waste (312 kg/cub.m)** in the Project Tree.

Now lets edit the properties of the Lateral Drainage Layer. Layers from this category have the same hydraulic parameters as Vertical Percolation Layers. In addition, the drainage and leachate recirculation parameters may be specified.

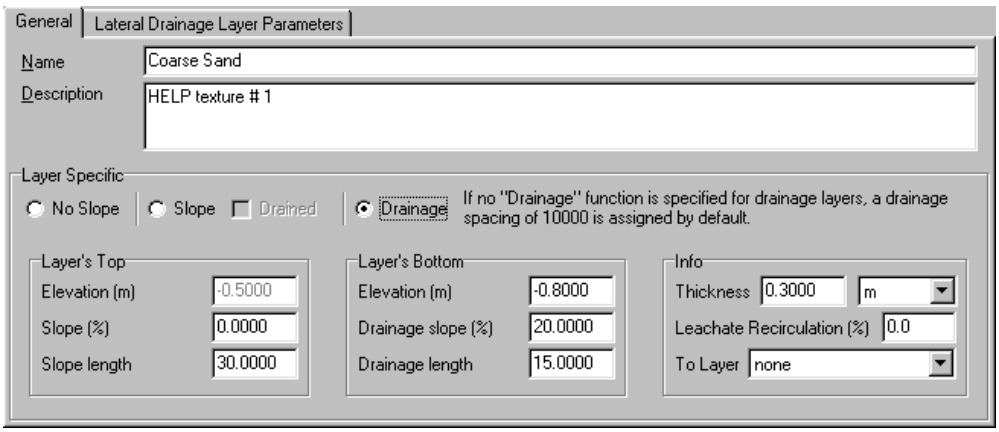
☞☞ **Coarse Sand** in the Project Tree. By default, the **Slope** function is selected for this layer.

Select **Drainage** by clicking the appropriate radial button. The name of parameter **Slope** in the **Layer's Bottom** box will change to **Drainage Slope** and the name of parameter **Slope Length** will change to **Drainage Slope**.

Type: 20 (%) for **Drainage Slope**

Type: 15 (m) for **Drainage Length**.

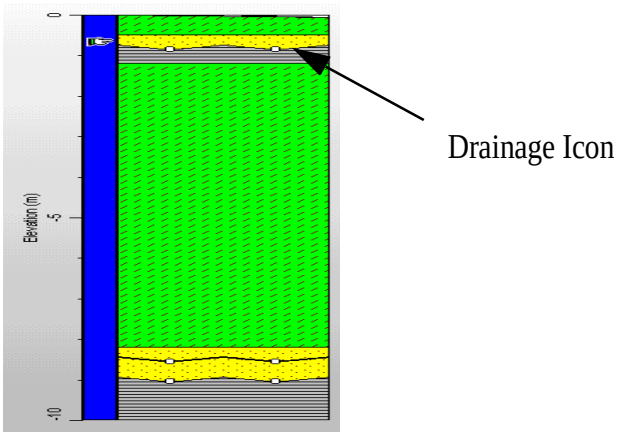
The **General** tab for **Coarse Sand** should be similar to the figure below.



☞ **[OK]**

☞ **[-],** beside **Coarse Sand** in the Project Tree.

The Profile View will change.



Finally, we will edit parameters of a Geomembrane Liner.

☞ **[+],** beside the **Low Density Polyethylene**

☞☞ **[Geomembrane Liner Parameters]**

☞ **Units,** beside the **sat. hydr. conductivity**

☞ **[cm/sec],** from the drop-down list.

Type: 7.00E-13, in the **Value** field.

☞ **[Units]**, beside the **pinhole density.**

☞ **[# / acre],** from the drop-down list.

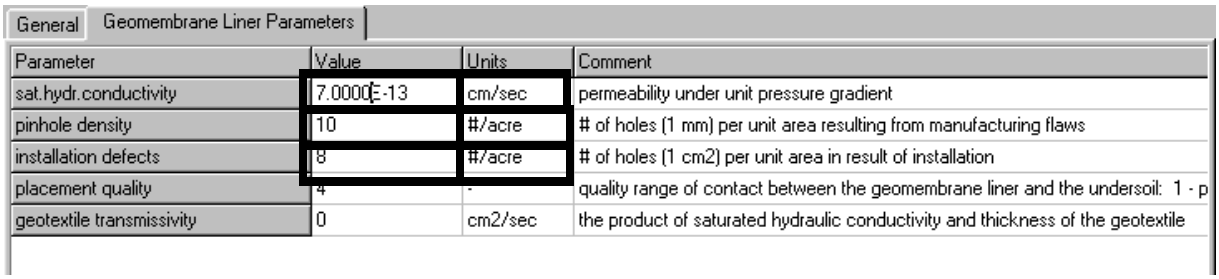
Type: 10, in the **Value** field.

☞ **Units,** beside the **installation defects.**

☞ **[# / acre],** from the drop-down list.

Type: 8, in the **Value** field.

The **Vertical Perc. layer Parameters** tab for LDPE liner will look like the figure below.



☞ **[OK]**

☞ **[-],** beside **Low Density Polyethylene** in the Project Tree.

With this action completed, we have finished editing the parameters of individual layers of our profile. However, there are more ways to do this, including the possibility of reassigning geometrical properties (slope, slope length) or drainage properties for a set of layers that perform a specific hydraulic function within a landfill (e.g. landfill cap).

Setting Initial Moisture Conditions & Runoff Parameters

Finally, we will now define our landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** parameter group located in the project tree..

As with the original HELP model, Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- *the model simulate values, or*
- *the user specify values.*

Depending on the selection made, runoff can be:

- *calculated by the model which will account for the type of material, slope, slope length and type of land cover,*
- *specified by the user,*
- *specified by the user and further adjusted to the slope and slope length.*

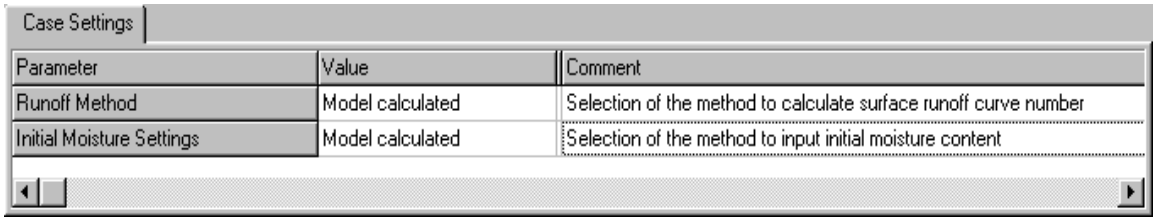
In the first case, which is the default, the model will assign realistic values for the initial moisture storage and simulate one year of landfill life. The values of moisture storage obtained from this simulation will be used as initial values.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. To set the automatic calculation,

☞☞ **Case Settings**, found in the Project Tree**.**

In the **Value** field beside **Runoff Method,** select **Model Calculated.** The program will automatically pick up the slope and slope length values for the top layer and use them in the runoff calculation.

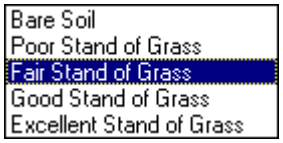
We will also leave the **Initial Moisture Settings** as **Model Calculated**



☞ **[OK]**

To specify additional parameters for surface runoff,

☞☞ **Surface Water Settings** from the Project Tree View.



To set the various vegetation classes, click in the **Value field**, found beside the **Vegetation Class.** The drop down list of available types of the land cover will appear.

☞ **Fair Stand of Grass**

☞ **[OK]**

Note: Surface Slope and Slope Length which regulate the runoff also, may be editied at the General Tab of the top profile layer.

You have now have completed creating your specific landfill profile. With a fully functional version of Visual HELP, you could save your work at this point however, as a demo, you will need to load a pre-run profile included with this demo. To continue,

☞ **File** from the main menu

☞ **Close Project**

☞ **[NO]** to close the project without saving your changes.

The next step is to generate your site specific weather using the USDA Weather Generating program that synthetically generates daily values for precipitation, temperature and solar radiation.

NOTE: Visual HELP has also an effective tool for generating aquifer recharge values that can be used in other models such as Visual MODFLOW. To calculate accurate recharge rates for your project, simply set your model profile to represent you site specific hydrogeological conditions and run!

Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting

We will now create site specific weather data that will be considered when modeling our landfill profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

☞☞ **HELP Tutorial 2** in the Project Tree. Note that two profiles (DEMO A & B) appear. **DEMO profile A** is a continuation fo the previous changes made in Module II.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill, and simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

For synthetic generation of daily values of precipitation, mean temperature, and solar radiation DOS HELP and Visual HELP version 1 included a Weather Generator developed by the Agricultural Research Service of the USDA (U.S. Department of Agriculture), as well as parameters for generating synthetic data for 139 U.S. cities. After Visual HELP 1.01 was released, WHI received requests from our clients worldwide to expand the area of Weather Generator application to other regions of the world.

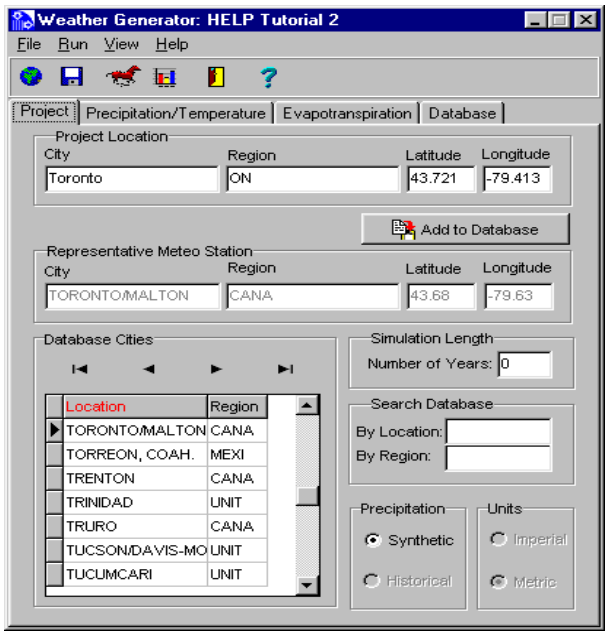
Trying to meet these requests, WHI developed a global database that includes more than 3000 stations and a GIS feature for searching the nearest stations globally for the Visual HELP version 2.1. If your landfill site is not located in our database, we recommend that you choose the closest city to your site and use the generated data for that location for your simulations. Visual HELP will search for the closest location of a given set of co-ordinates. This co-ordinate set can be entered as actual values or interactively with a map.

To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the main menu

☞ **Weather Generator**

After an introductory splash screen, the Weather Generator dialogue box will appear.



For this tutorial, you will be using data for the TORONTO/MALTON weather station. The TORONTO/MALTON station was selected among five stations located nearest to our landfill site.

Parameters for TORONTO/MALTON appear automatically in the text boxes throughout the **Weather Generator** dialogue boxes after it starts (if you have time, click **Precipitation/ Temperature, Evapotranspiration and Database** tabs to see the options to customize the weather parameters).

Let's generate weather data for our site for the next 20 years.

☞ **Number of Years** text box,

Type: 20, to represent the number of years you wish to build weather data for.

To run the Weather Generator,

☞  ,on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  ,to save the generated files

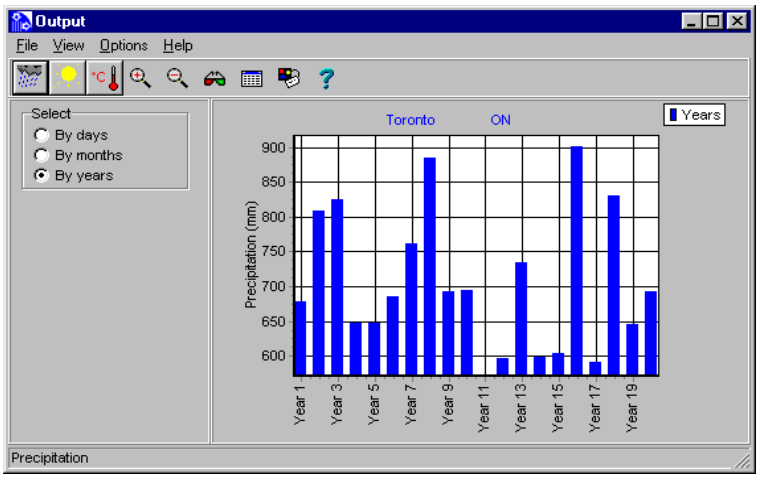
Viewing Weather Data

To view the results for the weather data you just generated,

☞  ,on the toolbar to view the **Output** dialogue box.

☞  ,on the toolbar to view generated precipitation values

By default annual totals will be presented, as shown below.

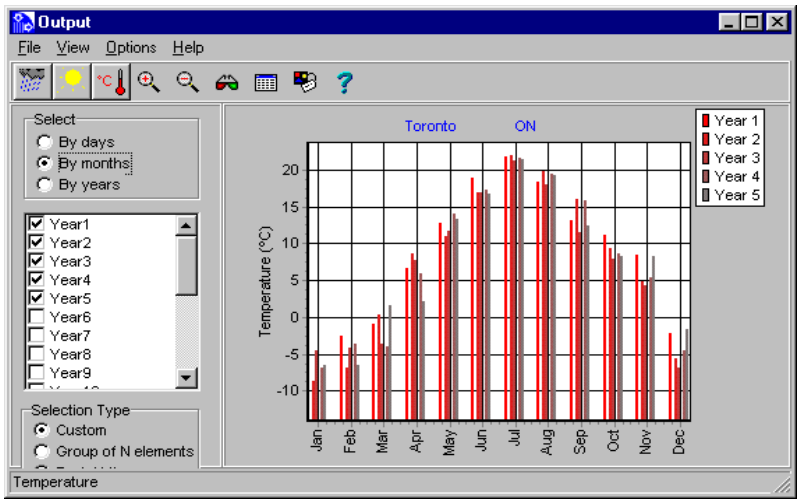


☞  ,to view the generated solar radiation data.

☞  ,to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  ,to view the generated raw data in table format.

☞  ,to close the table.

☞  ,to close the graphics.

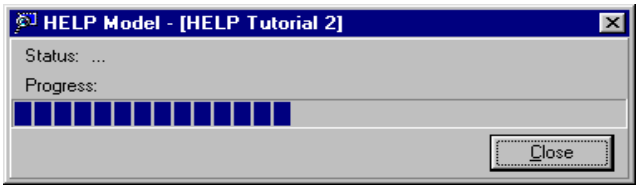
☞  ,to exit the Weather Generator.

You are now ready to run Visual HELP for 20 years using the landfill profile you created.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

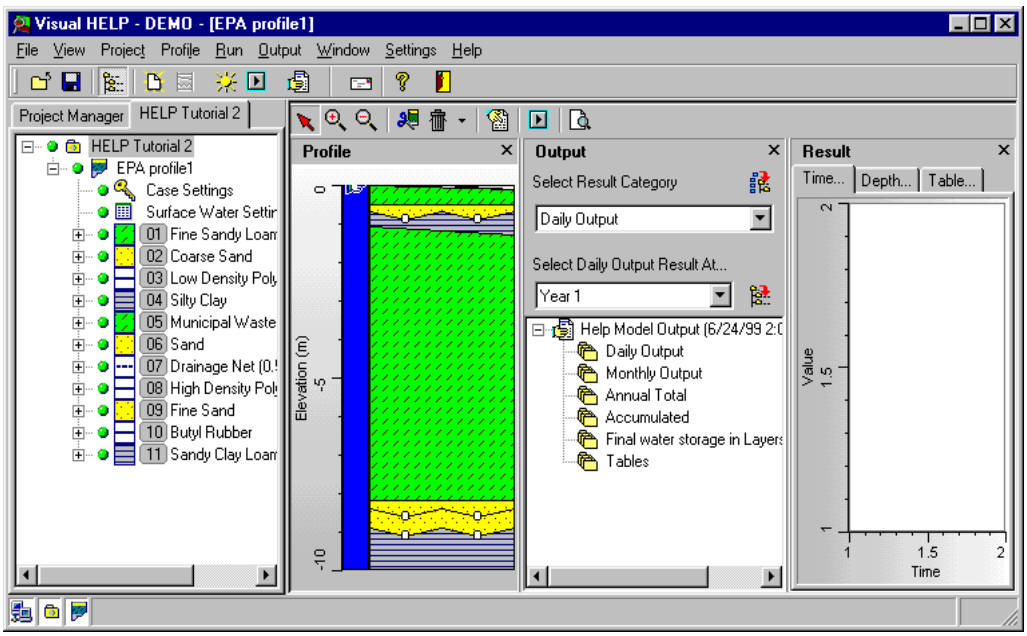


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

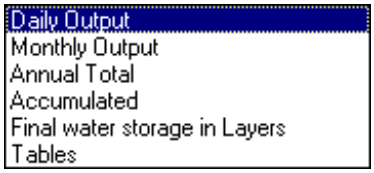
After the model has successfully run, Visual HELP will display the Output View and Result View windows.



☞  ,to enlarge the graphs viewing area

☞ **X** ,in the Profile View to close the window.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



You may select the category you wish to view however for this exercise, we recommend viewing your results with an **Accumulated** category. The option to graph **Accumulated Volumes** is a new feature, offered only in Visual HELP 2.1. This plot will show you the total volumes of water that were drained during the simulation period. This allows you to examine the total volume of leachate percolated through the landfill bottom during the simulated time and assess the total volumes of other water balance constituents.

☞ **Accumulated**

The results for the the **Accumulated** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox.

☞ **Select Accumulated Result at...** , to view the available balance types. ie) rate, volume

Rate means the annual rate of the balance constituent, and **volume** means the volume for the area represented by the profile.

☞ **rate**

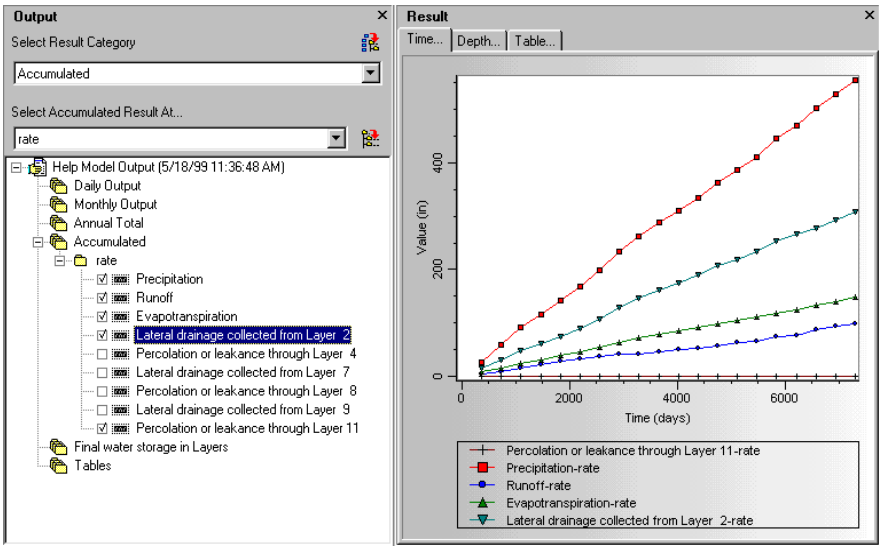


To view all the available results for this specific type of balance, click the icon to the right of the **Select Accumulated Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Lateral Drainage collected from Layer 2** (the first drainage layer), **Percolation or leakance through Layer 11** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



To erase a variable from the Result View window, deselect the corresponding check box in the Result Tree.

From the graph, one may conclude that Runoff, Evapotranspiration, and Lateral Drainage collected from Layer 2 play an important role in the landfill balance while Percolation through Layer 11 (the landfill bottom) is equal or close to 0. For specific values, the landfill balance constituents may be viewed in table format.

Viewing Tables

To view HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Accumulated rate**, from the **Select Table Result at..** lower listbox.

☞  **Add to Output Tree** icon to the right of the lower output listbox

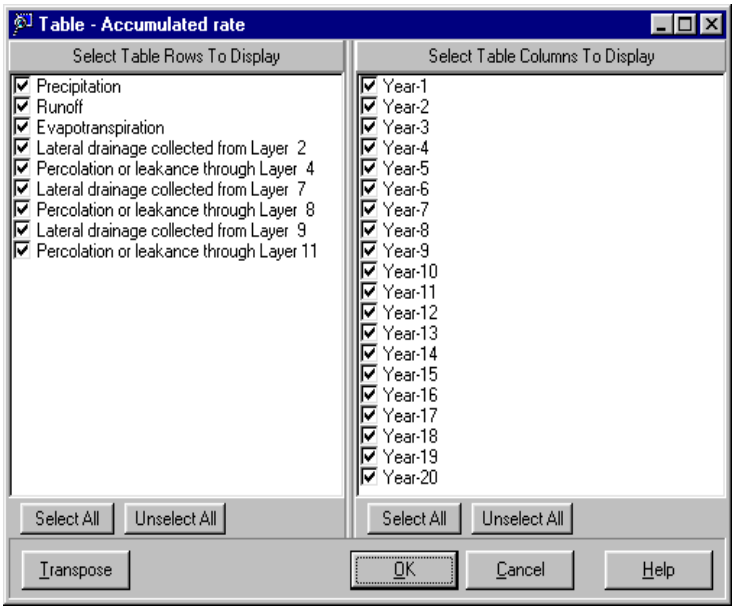
The table name **Accumulated rate** will appear in the Output Tree. For future use, let's place the table results into a new Result Window.

To do this, **<right click>** the name, **Accumulated rate** in the Output Tree. The following menu will appear:



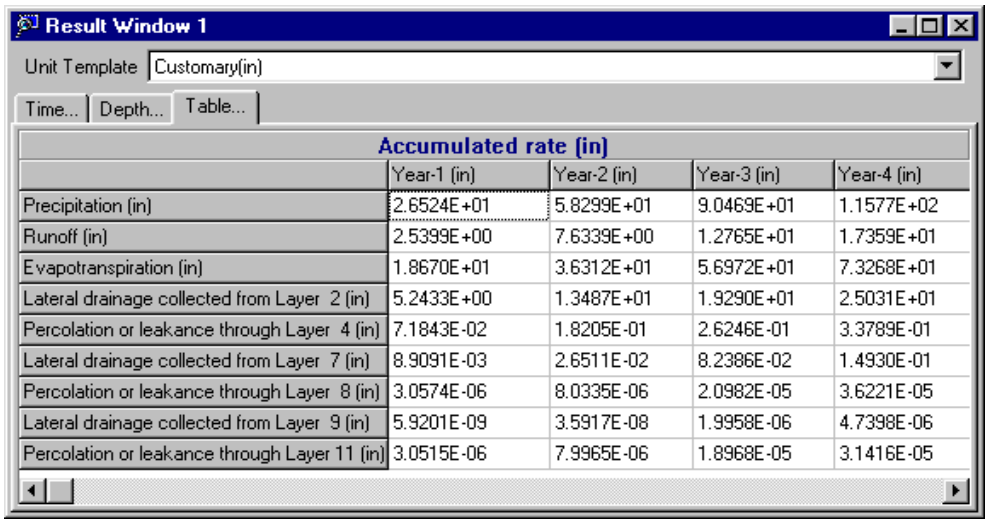
Choose **New Result Window**.

The New **Result Window 1** will open at the background and the table editing dialog box will appear.



☞ **[OK]**

The table will appear in **Result Window 1**:



Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation (in)	2.6524E+01	5.8299E+01	9.0469E+01	1.1577E+02
Runoff (in)	2.5399E+00	7.6339E+00	1.2765E+01	1.7359E+01
Evapotranspiration (in)	1.8670E+01	3.6312E+01	5.6972E+01	7.3268E+01
Lateral drainage collected from Layer 2 (in)	5.2433E+00	1.3487E+01	1.9290E+01	2.5031E+01
Percolation or leakance through Layer 4 (in)	7.1843E-02	1.8205E-01	2.6246E-01	3.3789E-01
Lateral drainage collected from Layer 7 (in)	8.9091E-03	2.6511E-02	8.2386E-02	1.4930E-01
Percolation or leakance through Layer 8 (in)	3.0574E-06	8.0335E-06	2.0982E-05	3.6221E-05
Lateral drainage collected from Layer 9 (in)	5.9201E-09	3.5917E-08	1.9958E-06	4.7398E-06
Percolation or leakance through Layer 11 (in)	3.0515E-06	7.9965E-06	1.8968E-05	3.1416E-05

Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation **553.98 in**

Runoff **74.84 in**

Evapotranspiration **359.14 in**

Lateral Drainage collected from Layer 2 **119.43 in**

Percolation or leakance through Layer 11 **0.00 in**

These values show that the default EPA landfill design which was slightly modified by you during the exercise, provides good ground water protection for our project location. Having made such an optimistic conclusion, lets prepare the Visual HELP report.

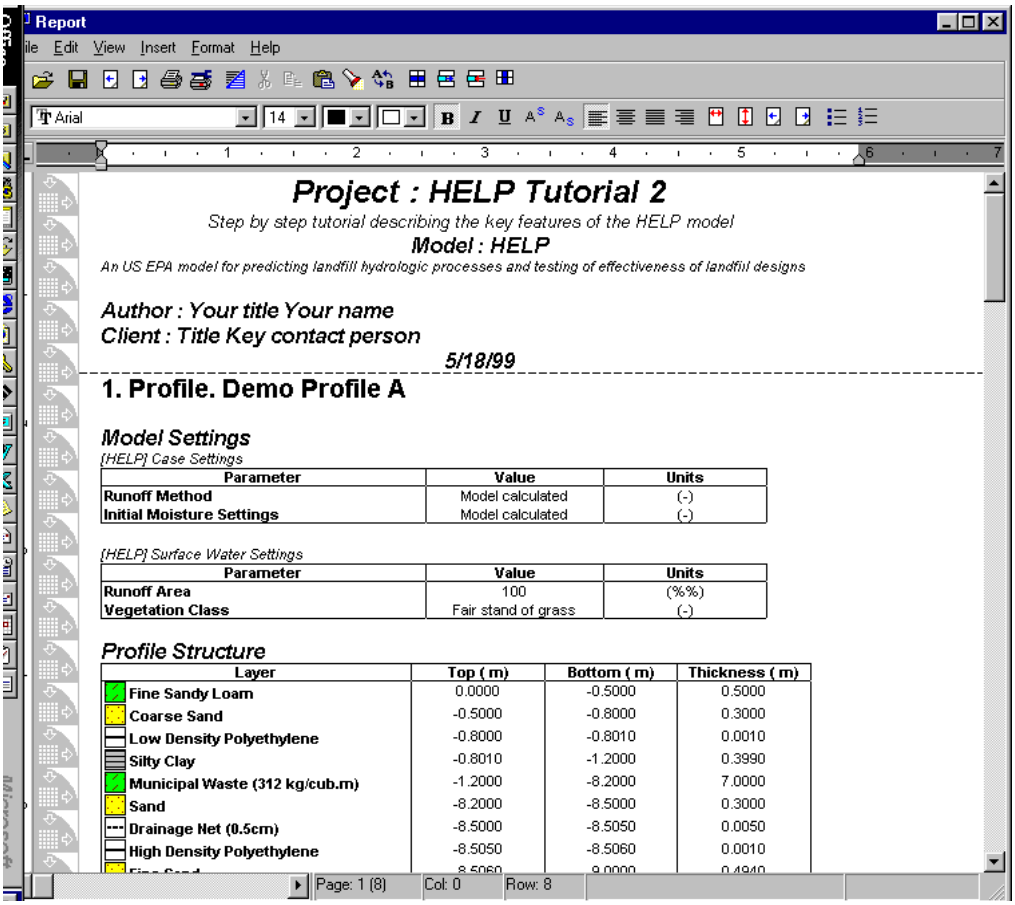
☞ , to close Result Window 1.

Creating a Report

As an added feature of Visual HELP, we have developed a time saving, report generator that will help you prepare profession reports of the model simulation.

To create a report and add the project input data, click the icon from the Operational Icons toolbar.

The Visual HELP Report Generator will display your report in a separate window.



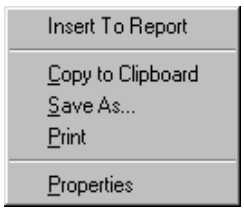
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add a graph form the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the graph in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The graph will be inserted into the report.

Go to the "Project" in the main menu and select **[view report]** to continue editing. To re-size the graph you just inserted, click the graph in the **Report** window and stretch it until it reaches the proper size.

To add a table from the **Result Window 1** to the report:

In the **Report** window place the cursor below the imported graph.

<right click> the table in the **Result Window 1**

3) ☞ **Insert To Report**. The table will appear in the report. Again, go to the "Project" in the main menu and select **[view report]** to continue editing or to view your report at any time.

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to , insert a headers /footers, change fonts/ letters size, etc..

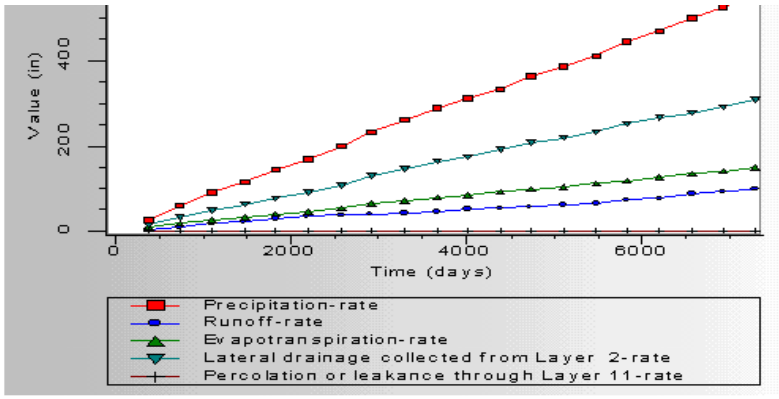


Fig.1 Accumulated balance constituents.

Table 1. Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation	2.6669E+01	5.8476E+01	9.0929E+01	1.1642E+02
Runoff	3.9219E+00	9.5287E+00	1.6193E+01	2.2387E+01
Evapotranspiration	8.2591E+00	1.5974E+01	2.4371E+01	3.1858E+01
Lateral drainage collected from Layer 2	1.4484E+01	3.1264E+01	4.8366E+01	6.2130E+01
Percolation or leakance through Layer 4	3.7569E-03	6.5516E-03	9.2846E-03	1.2289E-02
Lateral drainage collected from Layer 7	3.7532E-03	6.5417E-03	9.2717E-03	1.2271E-02
Percolation or leakance through Layer 8	3.7459E-06	8.9520E-06	1.2983E-05	1.7380E-05
Lateral drainage collected from Layer 9	3.5588E-06	8.4902E-06	1.2188E-05	1.6448E-05
Percolation or leakance through Layer 11	1.8707E-07	4.6209E-07	7.9524E-07	9.3185E-07

Once you are satisfied with your report, you may print the report and/or save it for future use. Let's take a look at additional output features available with Visual HELP.

Peak Daily Values

To assess the landfill hydrologic performance in extreme conditions, it is important to know when the peak hydrologic events occured. The **Peak Daily Values** table picks up the dates of such events.

☞ **[Tables]** from the **Select Result Category** drop-down listbox.

☞ **[Peak daily values]** from the **Select Table Result at..**

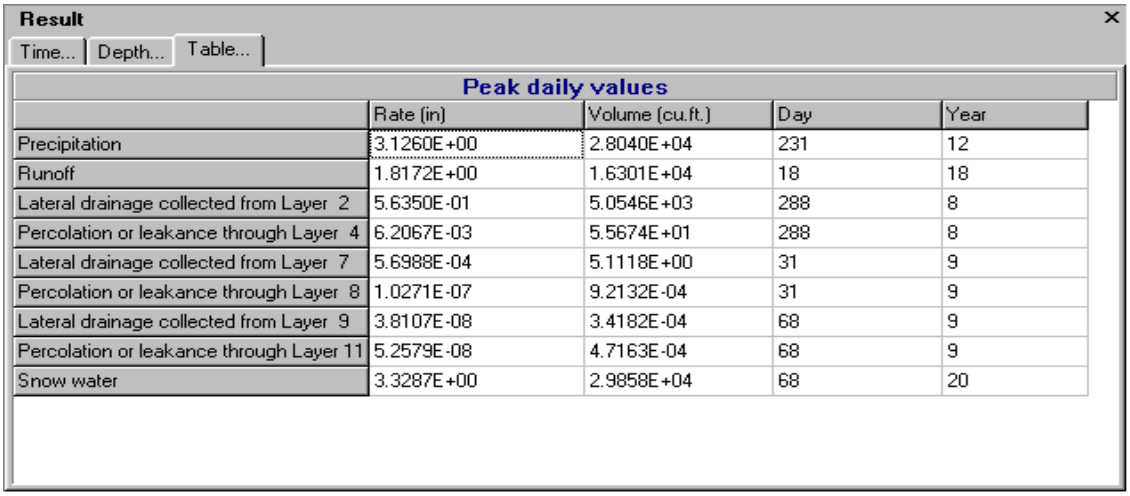


☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name "**Peak daily values**" will appear in the Output Tree. Click the check box beside it.

☞ **[OK]** in the table editing dialog box.

The following table will appear:



Peak daily values

	Rate (in)	Volume (cu.ft.)	Day	Year
Precipitation	3.1260E+00	2.8040E+04	231	12
Runoff	1.8172E+00	1.6301E+04	18	18
Lateral drainage collected from Layer 2	5.6350E-01	5.0546E+03	288	8
Percolation or leakance through Layer 4	6.2067E-03	5.5674E+01	288	8
Lateral drainage collected from Layer 7	5.6988E-04	5.1118E+00	31	9
Percolation or leakance through Layer 8	1.0271E-07	9.2132E-04	31	9
Lateral drainage collected from Layer 9	3.8107E-08	3.4182E-04	68	9
Percolation or leakance through Layer 11	5.2579E-08	4.7163E-04	68	9
Snow water	3.3287E+00	2.9858E+04	68	20

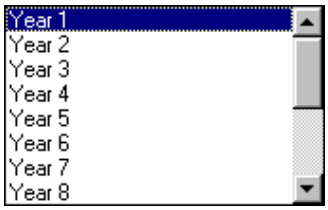
You may conclude that no leakage occurred through the bottom of the landfill (Peak daily percolation/leakage through layer 11). You may also find that the peak precipitation (3.12 in) happened on day 231 of year 12 and that peak runoff (1.81 in) happened on day 18 year 18.

To learn more about the circumstances of these events, Daily Output graphs may be studied.

Daily Output

☞ **[Daily Output]** from the **Select Result Category** drop-down listbox.

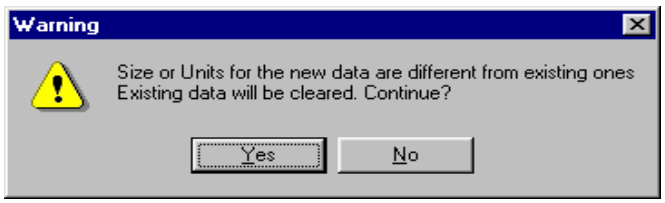
Next select the year for which you would like to have the results to be displayed from in the listbox.





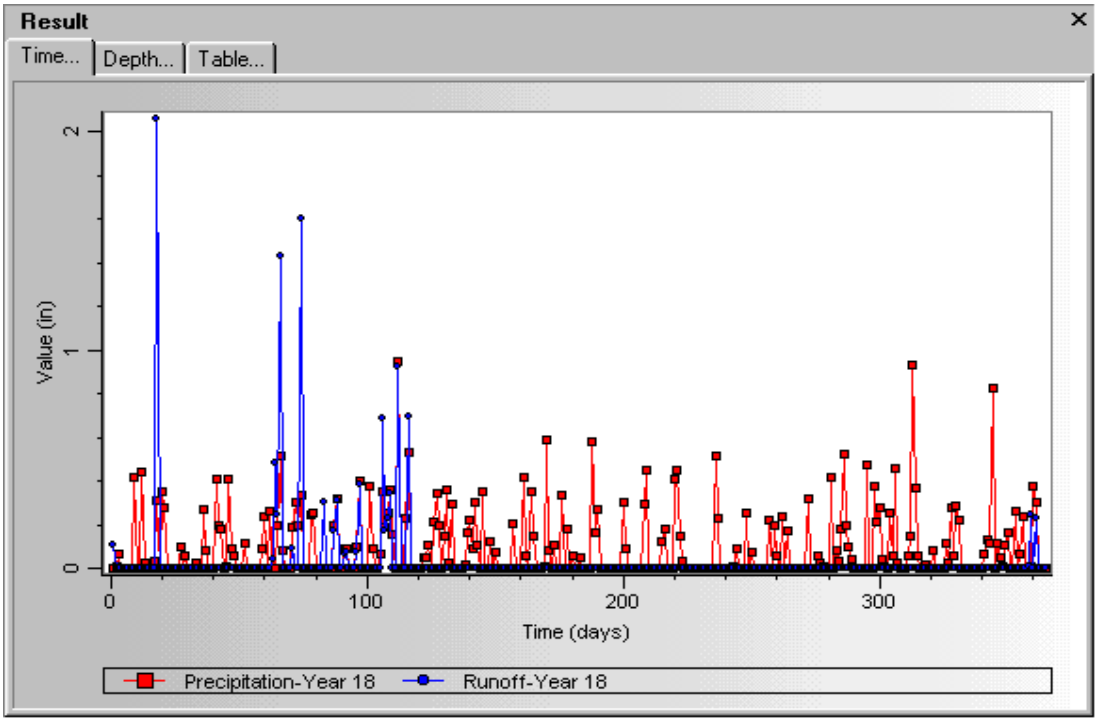
Scroll the list of years, and select the desired year: **18**. To view the available results for this specific type of balance, click the **Add to Output Tree** icon to the right of the lower output listbox.

The tree of available variables for year 18 will appear. Select **Precipitation** and **Runoff** by clicking the appropriate check boxes. This means that you are going to place these graphs to the Main Result Window. A warning will appear:



☞ **[Yes]**

The graph for the daily values of these variables will appear:



This graph shows that the maximum runoff event was not preceded by very intensive precipitation. However, after reviewing the weather data, it appears that day 18 was the first warm day with an average temperature of 5 degrees C (41 Fahrenheit) after a long span of frosty days. The peak runoff was caused by the intensive thaw of snow accumulated during the previous cold period.

Click to close the graph.

This concludes the Visual HELP demonstration exercise. Close Visual HELP by clicking in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers. If you have time, we would encourage that you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us.

Visual HELP Lab Exercise: Recharge

Estimating Groundwater Recharge with Visual HELP

The HELP Model

The **HELP** model (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile program used to design, evaluate and optimize landfill hydrology, and estimate groundwater recharge rates. The HELP model is used and recognized all over the world as the accepted standard for modeling landfill hydrology, and has become an integral component for projects involving landfill operating and closure permits.

The HELP model is a quasi-two-dimensional, multi-layer hydrologic model requiring the following input data for each model profile:

- Weather data (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil properties (porosity, field capacity, wilting point, and hydraulic conductivity)
- Design information (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can consist of a combination of natural (soil) and artificial materials (waste, geomembranes) with options to install horizontal drainage layers. The HELP model also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

Learning Objectives

After completion of this exercise you should be able to…

- Apply Visual HELP to obtain an estimate of a groundwater recharge rate from precipitation through the unsaturated zone;
- Understand the basic principals and limitations of the water balance method for estimating recharge from precipitation;
- Apply some of the many useful features of Visual HELP and the Weather Generator utility to actual projects.

Introduction

Visual HELP can be used to calculate a groundwater recharge rate from precipitation (rain or snow, or both) under the influence of site-specific soil properties, solar radiation, air temperature, wind speed, and vegetative properties. The HELP model (Schroeder, et al., 1994) predicts an infiltration rate under quasi two-dimensional influences through one or more uniform soil layers. The HELP model (Schroeder, et al., 1994) has been applied by contemporary groundwater practitioners to substantiate values of net recharge from precipitation obtained from historical

studies for use in groundwater flow and contaminant transport models (see Abraham, B., et al., USAE Waterways Experimental Station, 1998).

In this exercise you will use the HELP model (Schroeder, et al., 1994) in Visual HELP to calculate the potential net recharge to groundwater from precipitation under site-specific soil, vegetative, and climatic conditions for a fictitious site near San Francisco, California using the water balance method. The water balance method calculates potential net recharge as a residual of all other water balance components and is expressed as:

$$R = P - D - ET_a - \Delta W \qquad (1)$$

where,

R is potential net recharge, P is precipitation, D is net runoff, ET_a is actual evapotranspiration, and ΔW is the change in soil water storage.

Limitations of the Water Balance Method

In arid and semi-arid regions application of the water balance method is more difficult to apply than in humid regions because precipitation in these regions is only slightly larger than evapotranspiration. Therefore, small measurement errors in either of these two components of the water balance equation can cause large errors in recharge rate estimates (Simmers, I., et al., 1998; Lerner, D. et al., 1990). Care should therefore be taken when applying the water balance method in water scarce regions, and should be used to calculate recharge events for one- to ten-day intervals to detect ephemeral recharge events common to these regions.

Note: All theoretical components of this tutorial are italicized.

Terms and Notations

For this tutorial, the following terms and notations will be used:

type	- type in the given word or value
☞ , select	- click the left mouse button where indicated
√	- press the <Tab> key
↵	- press the <Enter> key
+	- click the left mouse button where indicated
☞☞	- double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click or values to type in.

[….**]** denotes a button to click, either in a window, or in the side or bottom menu bars.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Background Information

Based on laboratory analysis of numerous soil samples collected at the site from split spoon and Shelby tube sampling, test pit soil profile descriptions, and a comprehensive vegetation survey, the following characterization of the unsaturated soil profile and vegetative cover was made:

1. The unsaturated zone exists to a depth of 10 feet, where the water table is encountered;
2. The ground surface is covered with an excellent stand of mature grass;
3. The root zone extends to a depth of 12 inches;
4. The soil is classified as a fine sandy loam;
5. Surface slope is 12 percent;
6. Leaf area index is 4;
7. Total Porosity: 0.48 (vol/vol)
8. Field Capacity: 0.25 (vol/vol)
9. Wilting point: 012 (vol/vol)
10. Initial soil water content: 0.28 (vol/vol)
11. Saturated hydraulic conductivity: 5.0 x 10^{-4} (cm/sec)

Starting Visual HELP

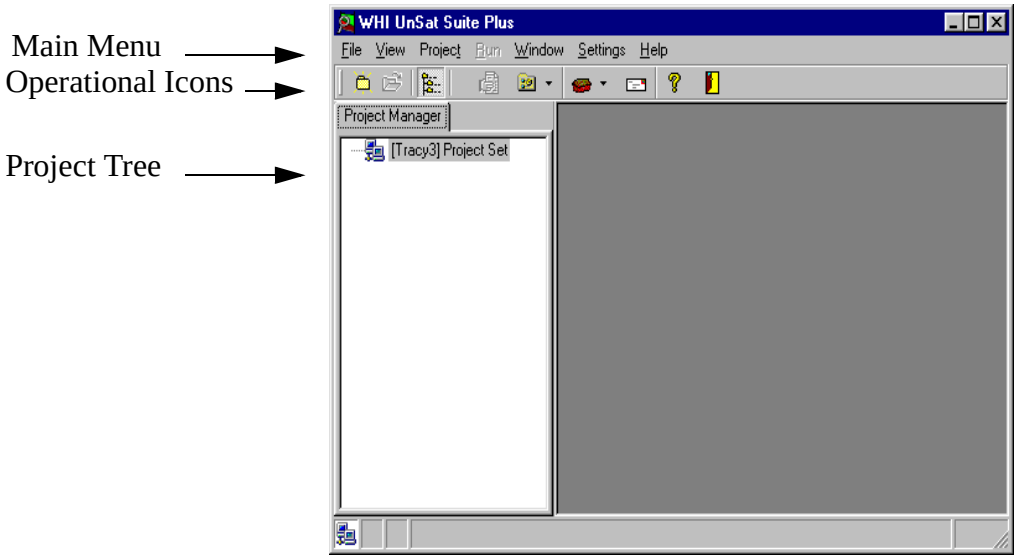
Visual HELP is a component of the WHI UnSat Suite Plus package. To start the WHI UnSat Suite Plus, double click the following icon on your desktop.



An introductory screen will be displayed when you open WHI Visual HELP/UnSat Suite



After the program is loaded, the main WHI Visual HELP/UnSat Suite Plus window will appear on your display:



The WHI UnSat Suite Plus interface structure has been designed to help you navigate through the program with ease. The main menu is located at the top, similar to our other programs. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

Creating a New Project

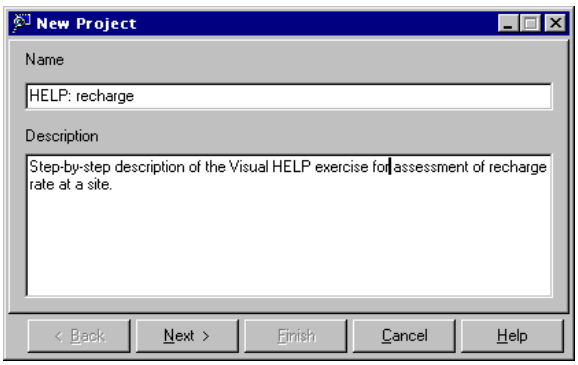
To create a new project, go the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new Visual HELP project.

In the **Name** box type: **HELP: recharge**.

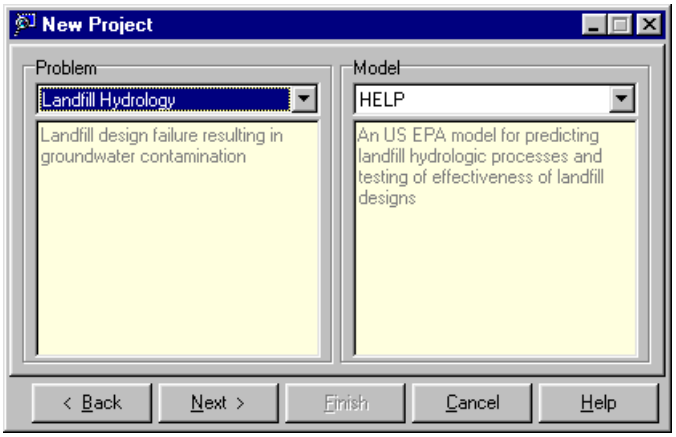
In the Description box type: **Step-by-step description of the Visual HELP exercise for assessment of recharge rate at a site.**



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Problem class that should be used with the selected Problem type.



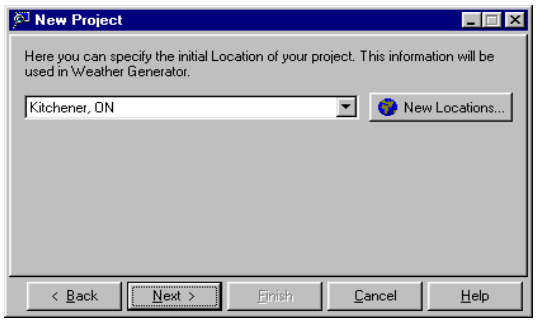
☞ **[Next]**

Once completed, the site location must be defined.

Selecting the Location

We will now select the location of our project using the GIS - WHI Locator.

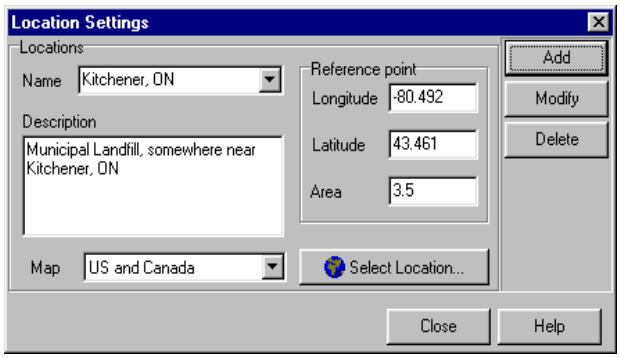
The following dialog box should already appear on your screen.



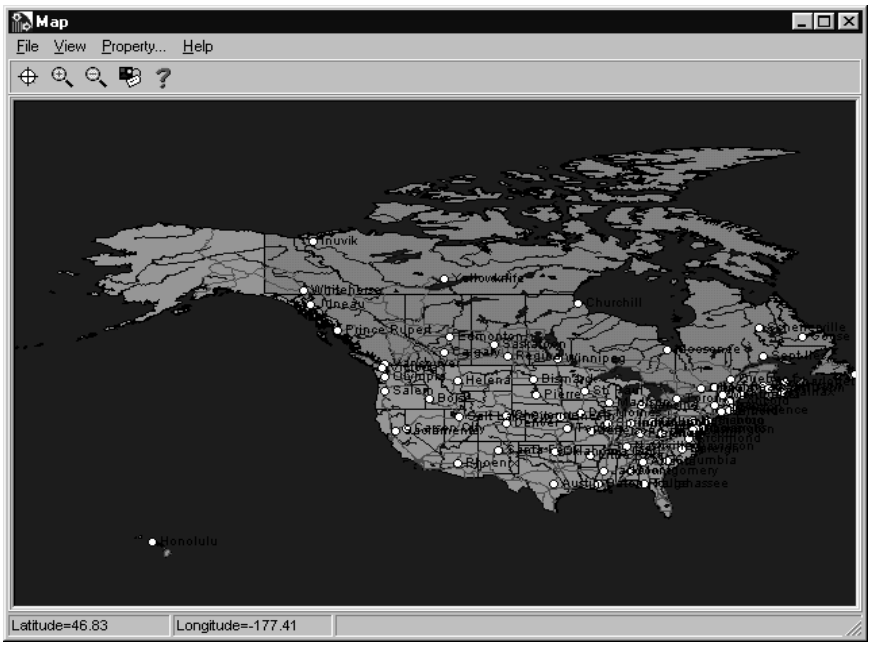
At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



☞ **[Select Location..]** button**,** to enable the Site Locator, developed by WHI. As shown below, a map of USA, Canada, and Mexico appears on your display:



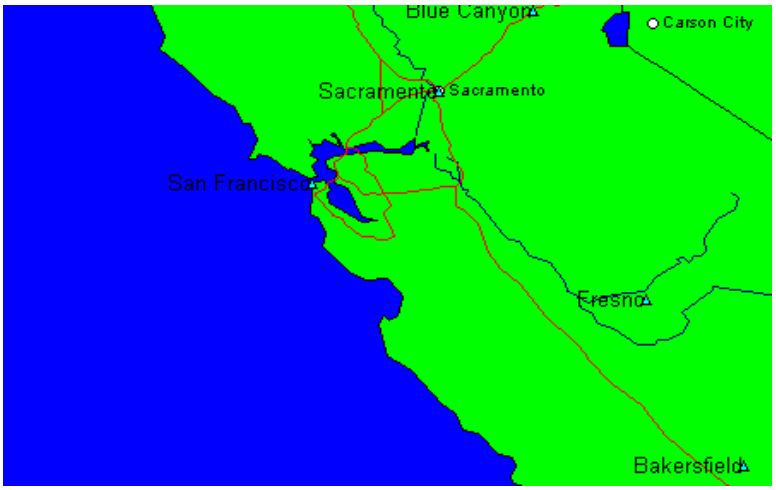
Now lets select San Francisco as your site location.

☞  , to zoom into our location.

Move the mouse to the right of the region where the landfill project is located (the West Coast). The coordinates, seen in the bottom left of the screen, should read approximately Latitude: 41, Longitude:-127. Press the left mouse button and stretch a zoom rectangle to the bottom and right,

 Copyright 2000 Waterloo Hydrogeologic, Inc.

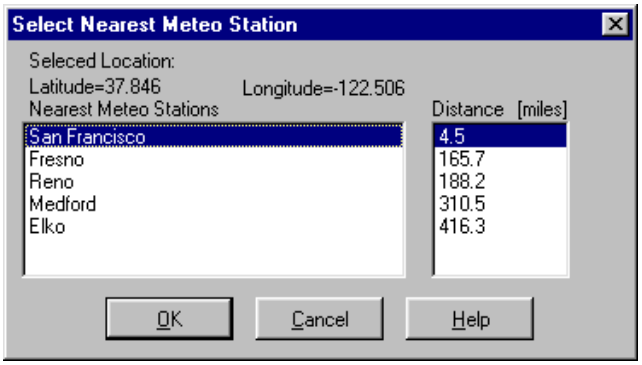
release the button (approximate Latitude: 33, Longitude:-115). After one or two zooms, the figure below should resemble your screen.



Next you will select San Francisco to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

Move the crosshair to the spot on the map where San Francisco is located and click to the north of it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you click directly on the city, the location will be forwarded to the database automatically. No nearest stations will show up.

In the "Select Nearest Meteo Station" dialog box,

☞ **[San Francisco],** as the nearest weather station.

☞ **[OK]**

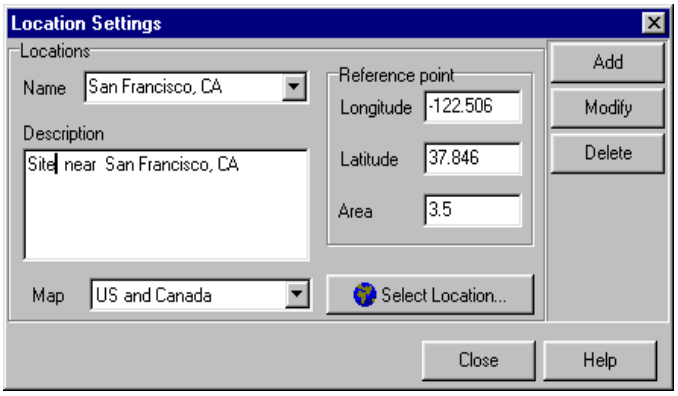
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: San Francisco, CA in the **Name** box.

Type: Site near San Francisco, in the **Description** text box.

Type: 1 , in the landfill area text box.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database.

☞ **[Close]**

While back in the **New Project**, select **San Francisco, CA** from the drop-down list.

☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.

The Visual HELP Unit Converter has been developed to maximize the flexibility for data input and output unit conversion. To show you the range of possibilities, just view the list of available units for hydraulic conductivity:



For this demo exercise we will specify that all input units will be in **Customary**, and output will be in **Customary (in)** units. The difference between the two unit templates is that Length in **Customary** template is set in feet while in **Customary (in)** it is set in inches.

From the **Input** drop-down list,

☞ **Customary**

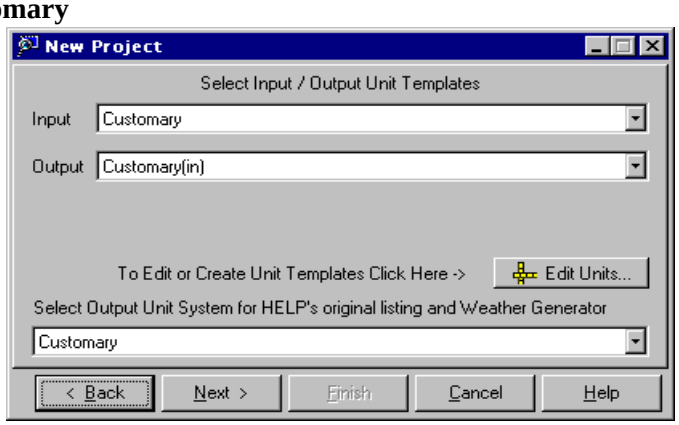
From the **Output** drop-down list

☞ **Customary (in)**

From the **Select Output Unit System for original listing and Weather Generator** list box,

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **Customary**



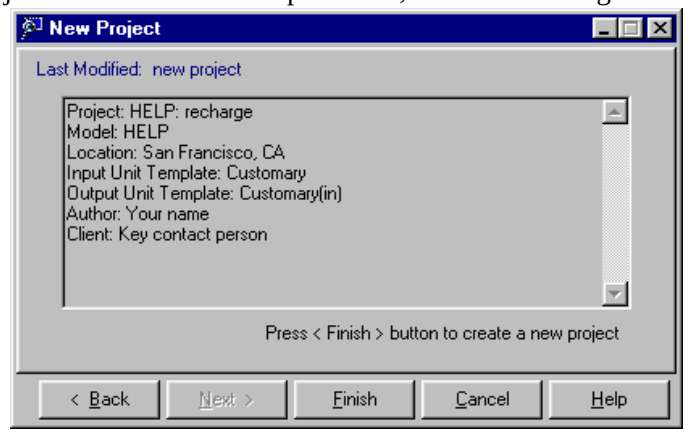
☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

All the project information will be presented, as seen in the figure below:



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings are specified, the **New Profile Wizard** will open.

Creating the Soil Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:



In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the later option.

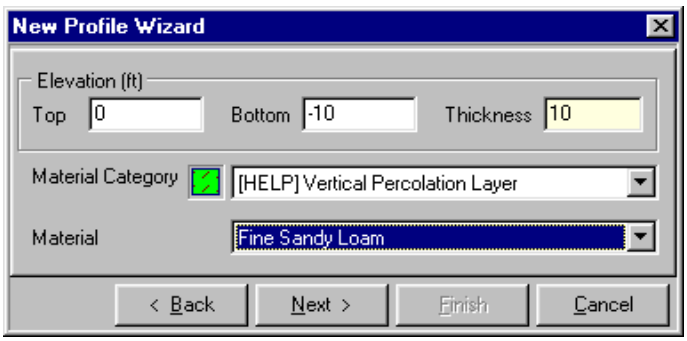
☞ **[create new profile]**

☞ **[Next]**

The dialog box with default profile parameters will appear. In this dialog box you may assign the parameters of the simulated profile. In your case the default profile thickness matches your profile characteristic.

☞ the drop-down arrow in the **Material** drop-down list box.

Select: **Fine Sandy Loam**. As a result of editing, the **New Profile Wizard** window will appear as shown below:



☞ **[Next]**

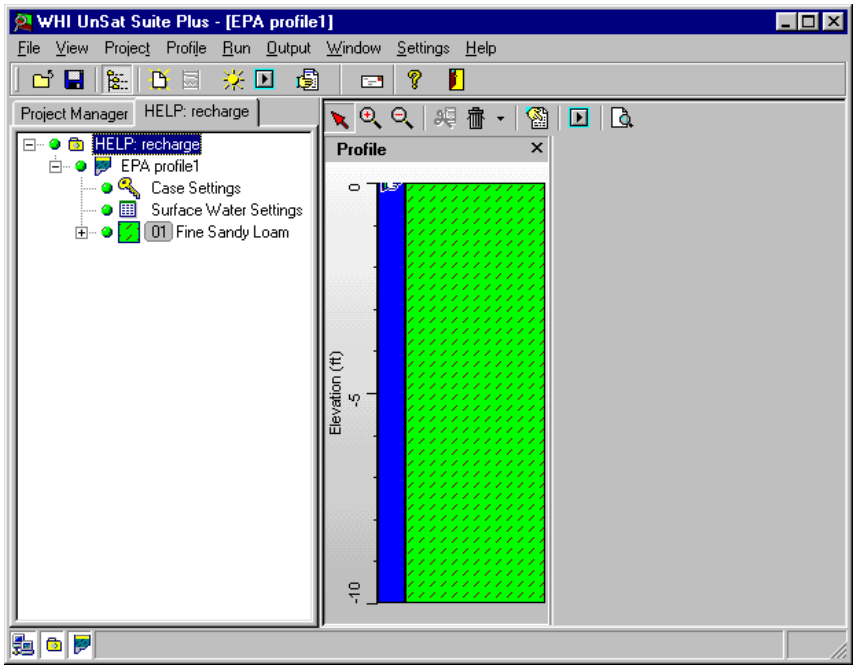
All the profile information will be presented:



☞ **[Finish]** to load the profile.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The new project details will be added to the project tree. The following picture will appear on your display:



This concludes creating a new project using Visual HELP

Editing Layer Parameters

To edit parameters of the Fine Sandy Loam layer,

☞ ☞ **Fine Sandy Loam,** in the Project Tree.

The **Profile Material Properties** dialog box will appear. In this dialog box, you can view and edit information about the layer including:

category, material texture, slope, top or bottom elevation, and thickness.

Under **Layer Specific**,

☞ **Slope** option button to enable surface runoff simulation.

Under **Layer Top**,

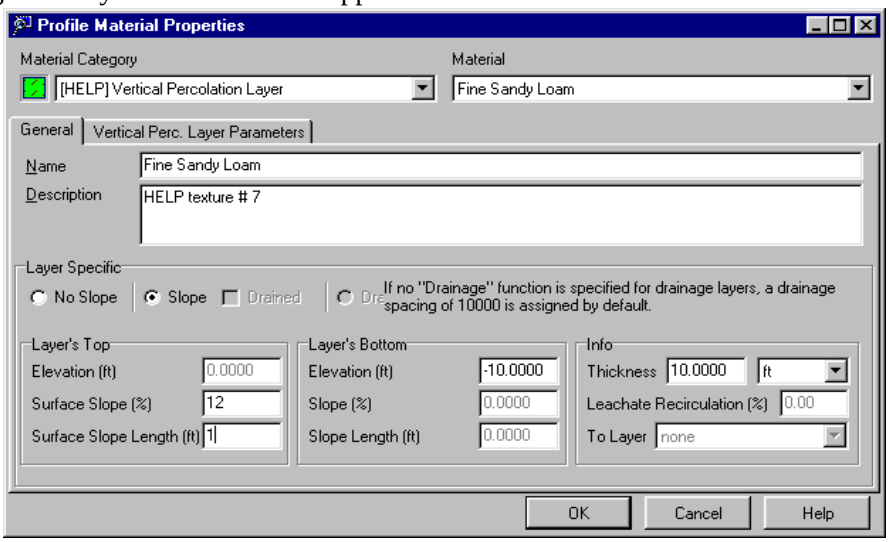
In the **Surface Slope (%)** text box:

type: **12**

In the **Surface Slope Length (ft)** text box:

type: **1** (to set unit length of slope for this exercise)

Finally, the dialog box on your screen should appear as shown below:



☞ **Vertical Perc. layer Parameters** tab to edit the material properties of a particular layer.

☞ **Value** beside **total porosity**

Type: 0.48

☞ **Value** beside **field capacity**

Type: 0.25

☞ **Value** beside **wilting point**

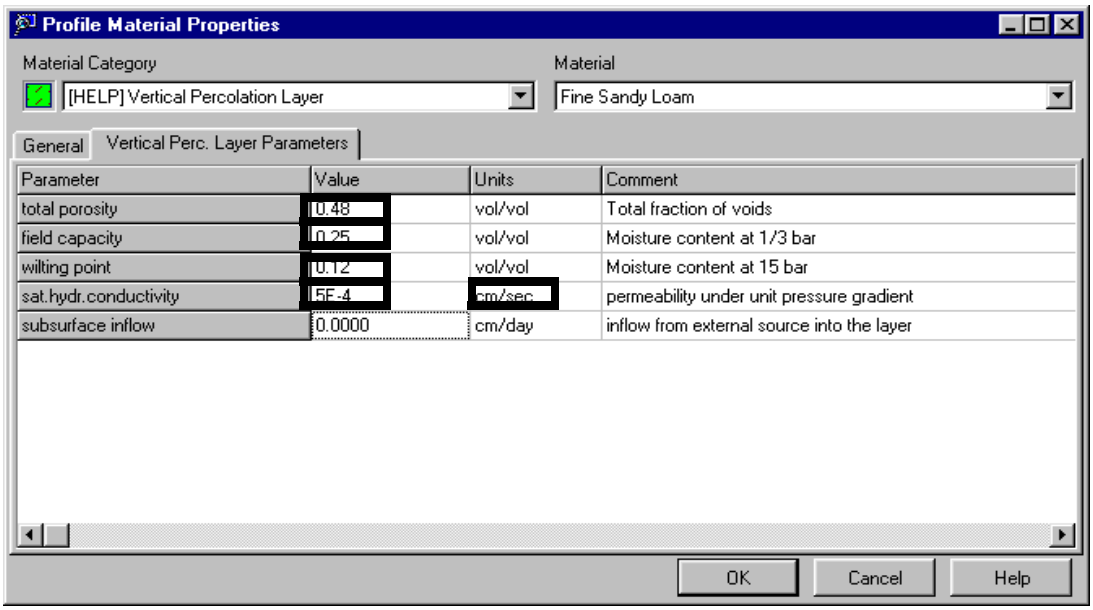
Type: 0.12

☞ **Units** beside the **sat. hydr. conductivity**, a drop-down list with available units will appear.

☞ **cm/sec**

Type: 5E-4, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **[OK]**

☞ **[-] ,**beside **Fine Sandy Loam** in the Project Tree.

Setting Initial Moisture Conditions & Runoff Parameters

Finally, you will define your landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** and **Surface Water Settings** parameter groups located in the project tree.

Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- the model simulated values, or
- the user specified values.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. These settings, accessible through the **Case Settings** parameter group in the Project Tree are default and do not require your intervention.

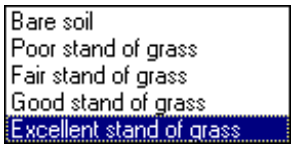
To edit additional parameters for the surface runoff simulation:

☞☞ **Surface Water Settings** from the Project Tree View.

To set the vegetation class,

☞ in the **Value** field, found beside the **Vegetation Class.** The drop down list of available types of the land cover will appear.

☞ **Excellent Stand of Grass:**



☞ **[OK]**

This completes the data preparation process. With a fully functional version of Visual HELP you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **HELP Recharge tutorial 1**, which was specifically developed for you and contains all changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

☞ ☞ project **HELP Recharge tutorial 1.**

You will now proceed with generating the required weather data and, subsequently, running Visual HELP.

Generating Weather Data

We will now create site specific weather data that will be considered when modeling your profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill, and simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

For synthetic generation of daily values of precipitation, mean temperature, and solar radiation DOS HELP and Visual HELP version 1 included a Weather Generator developed by the Agricultural Research Service of the USDA (U.S. Department of Agriculture), as well as parameters for generating synthetic data for 139 U.S. cities. After Visual HELP 1.01 was released, WHI received requests from our clients worldwide to expand the area of Weather Generator application to other regions of the world.

Trying to meet these requests, WHI developed a global database that includes more than 3000 stations and a GIS feature for searching the nearest stations globally for the Visual HELP version 2.1. If your landfill site is not located in our database, we recommend that you choose the closest city to your site and use the generated data for that location for your simulations. Visual HELP will search for the closest location of a given set of co-ordinates. This co-ordinate set can be entered as actual values or interactively with a map.

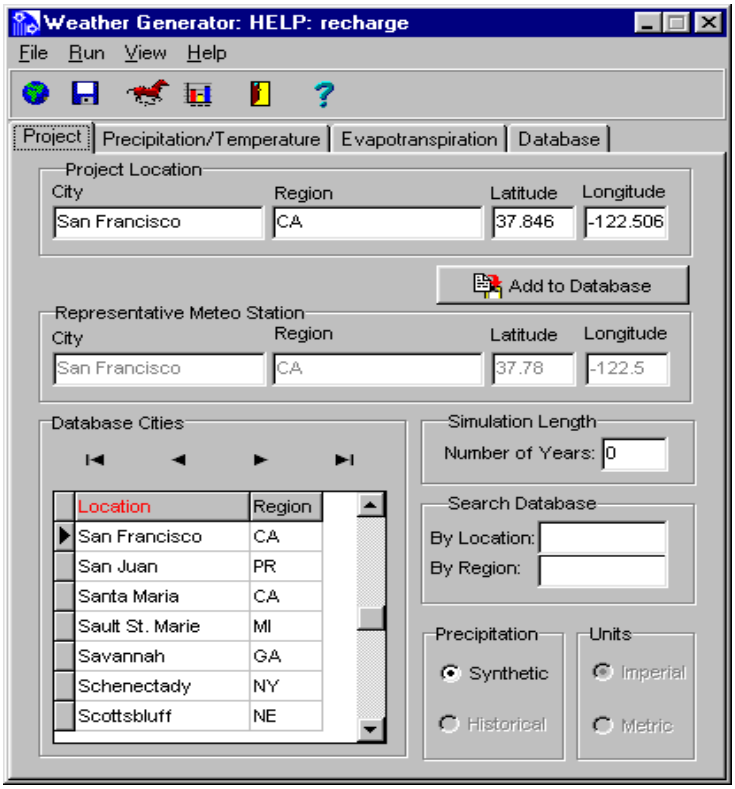
To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the main menu

☞ **Weather Generator**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

After an introductory splash screen, the Weather Generator dialog box will appear.



For this tutorial, you will be using data for the San Francisco weather station.

Parameters for San Francisco appear automatically in the text boxes throughout the **Weather Generator** dialog boxes after it starts (if you have time, click **Precipitation/Temperature and Database** tabs to see the options to customize the weather parameters).

According to the exercise scenario, you have to edit values for evaporative zone depth and leaf area index.

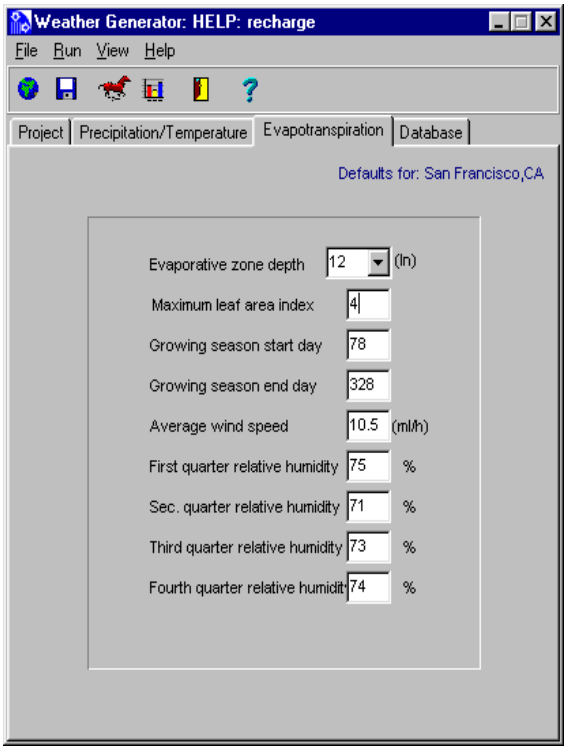
☞ **Evapotranspiration** tab.

In the text box for **Evaporative zone depth**:

type: 12

In the text box for **Maximum leaf area index**:

type: **4**



Let's generate weather data for our site for a 100 year period.

☞ **Project** tab

☞ **Number of Years** text box,

Type: 100, to represent the number of years you wish to build daily weather data for.

To run the Weather Generator,

☞  ,on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  ,to save the generated files

Viewing Weather Data

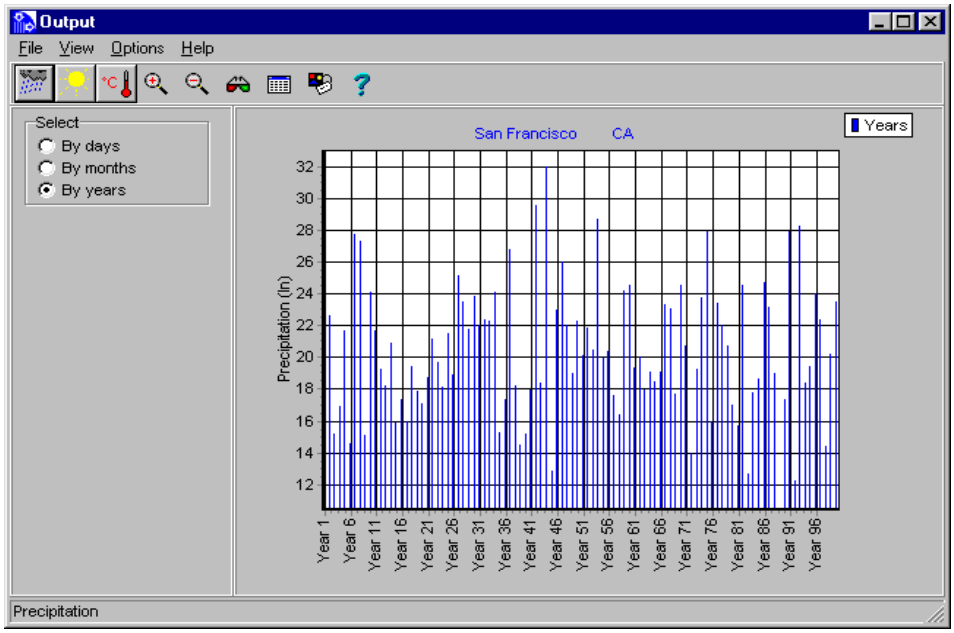
To view the results for the weather data you just generated,

☞  ,on the toolbar to view the **Output** dialog box.

☞  ,on the toolbar to view generated precipitation values

 Copyright 2000 Waterloo Hydrogeologic, Inc.

By default annual totals will be presented, as shown below.

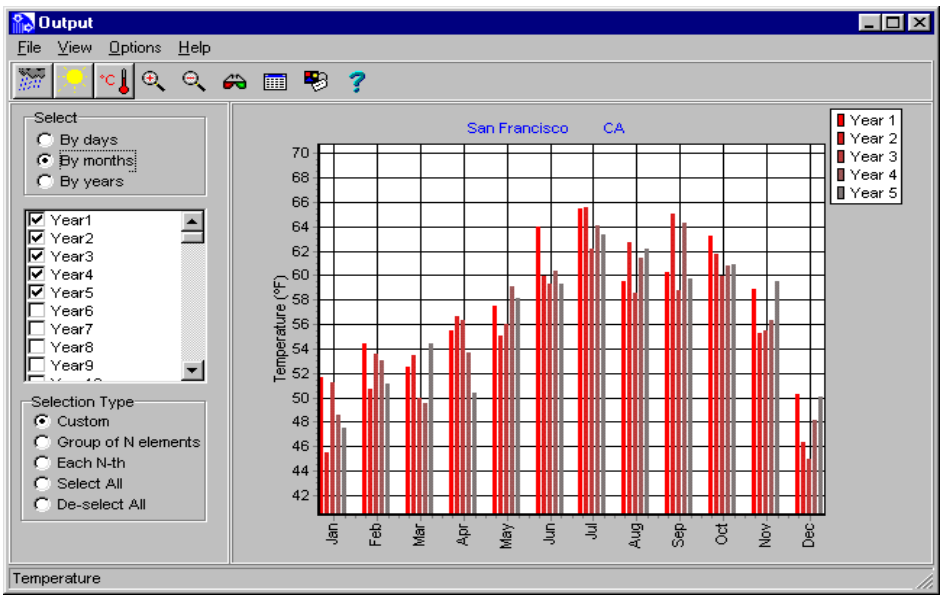


☞  ,to view the generated solar radiation data.

☞  ,to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  ,to view the generated raw data in table format.

☞  ,to close the table.

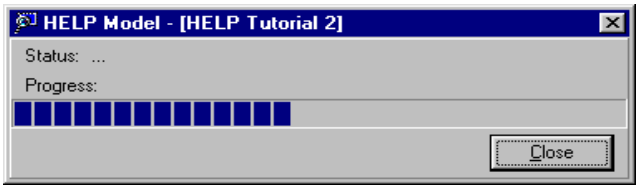
☞  ,to close the graphics.

☞  ,to exit the Weather Generator.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

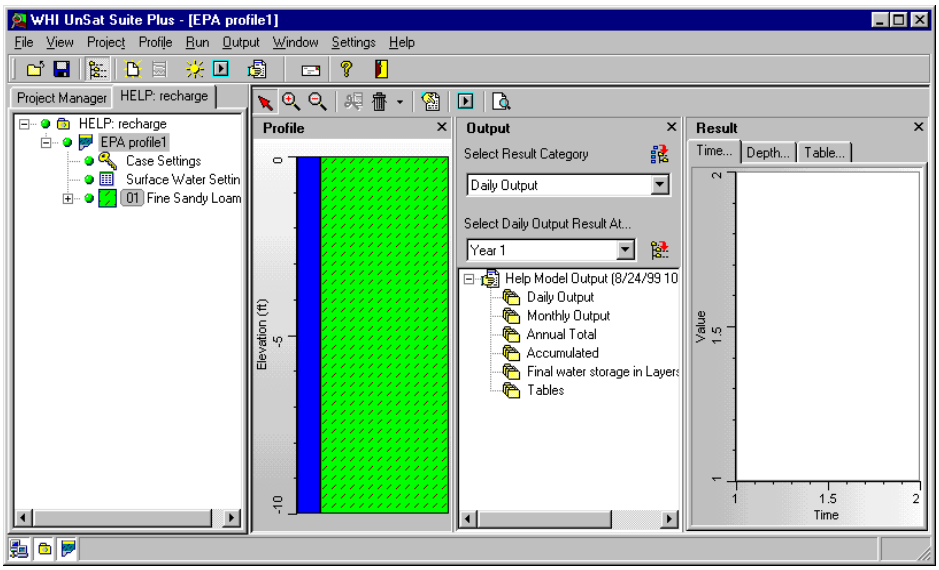


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

After the model has successfully run, Visual HELP will display the Output View and Result View windows.

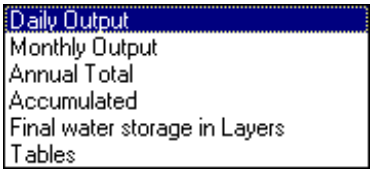


☞  ,to enlarge the graphs viewing area

☞ **X** ,in the Profile View to close the window.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



You may select the category you wish to view, however for this exercise, we recommend viewing your results with an **Annual Total** category. This plot will show you the annual amounts of water that were drained during the simulation period and assess volumes of other water balance constituents.

☞ **Annual Total**

The result for the **Annual Total** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox or

☞ **Select Annual Total Result at...** You will see: rate, volume, percent

Rate means the annual rate of the balance constituent, **volume** means the volume for the area represented by the profile, and **percent** means the percent of the amount of water entering the soil surface.

☞ **rate**

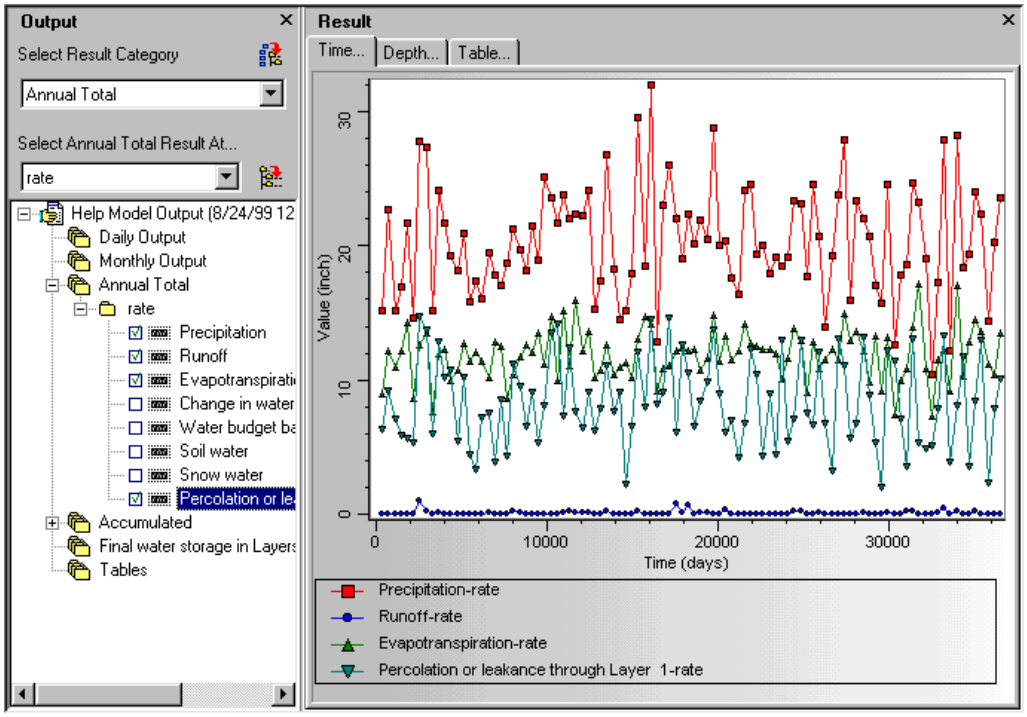


To view all the available results for this specific type of balance, click the icon to the right of the **Select Annual Total Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



This plot illustrates the following:

- At the end of first year, approximately 6.3 inches of recharge (labeled as percolation in the time series plot) is predicted to reach the groundwater.
- Peak recharge is seen to occur in year 7 of the simulation with an annual recharge rate of approximately 14.8 inches for the year.
- Annual recharge can be as low as 2.0 in. (year 81).
- A strong correlation exists between the amount of precipitation and recharge. This can be seen in the similar shapes of the precipitation and recharge plots over the 100-year simulation.
- The level of evapotranspiration activity affects the recharge rate. The recharge rate is higher when periods of high precipitation and low evapotranspiration activity correspond.
- Surface runoff does not play an important role in the profile water balance.

To get information about the principal proportion of the profile water balance constituents, lets examine the output data presented in **Accumulated** category.

From the top menu:

☞ **Output**

☞ **Clear Display Results**

☞ in the **Select Result Category** drop-down listbox.

☞ **Accumulated** from the drop-down list:

☞ in the **Select Annual Total Result at...** drop-down listbox.

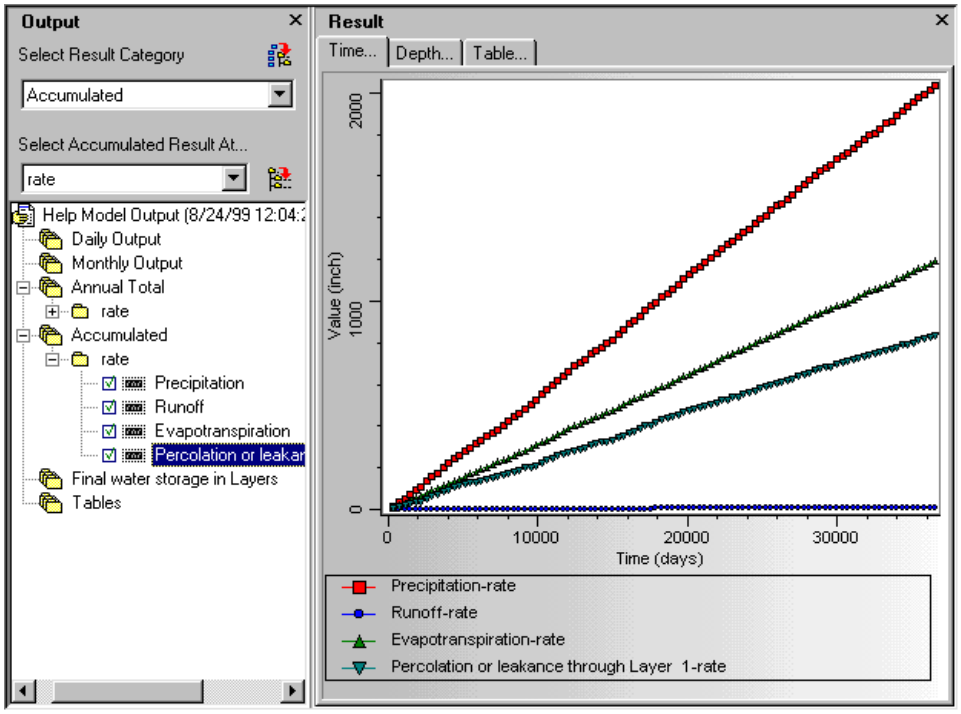
☞ **rate**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

To view all the available results for this specific type of balance, click the icon to the right of the **Select Annual Total Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1** (percolation through the bottom).



This plot illustrates that recharge is growing gradually totalling around 840 inches in 100 years.

For specific values, the profile balance constituents may be viewed in table format.

Viewing Tables

To view Visual HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Annual Totals rate**, from the **Select Table Result at..** lower listbox.

☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name **Annual Totals rate** will appear in the Output Tree.

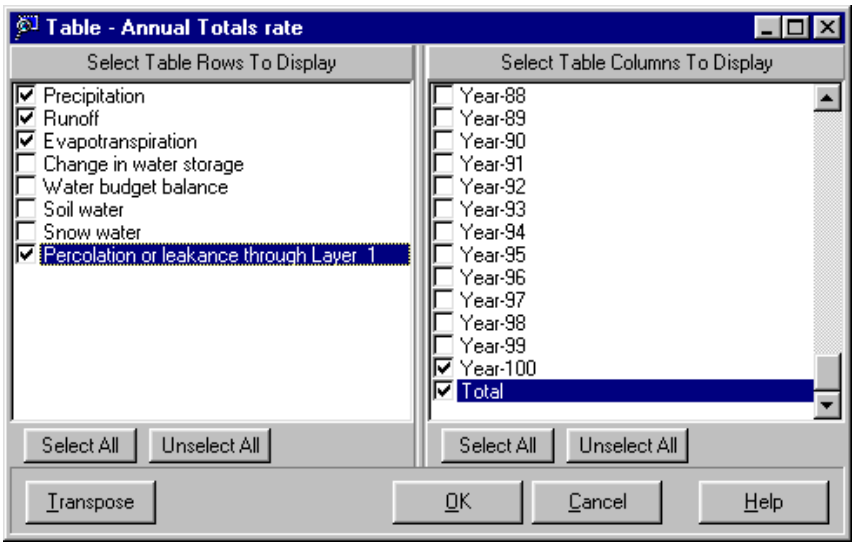
Click the check boxes beside **Annual Totals rate.** The table editing dialog box will appear.

☞ **[Unselect All]** button below the left panel.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1**

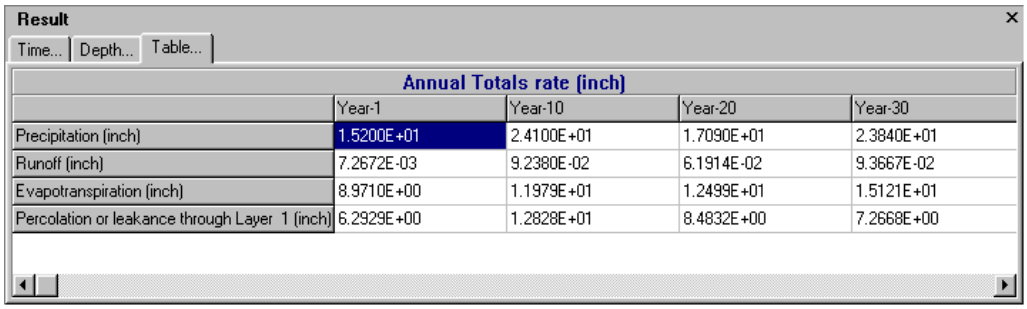
☞ **[Unselect All]** button below the right panel.

Click the check boxes beside the following output times: **Year-1, Year-10, Year-20, Year-30, Year-50, Year-100 and Total.**



☞ **[OK]**

The table will appear in the Output Window:



Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation **2034.6 in**

Runoff **8.56 in**

Evapotranspiration **1192.0 in**

Percolation or leakance through Layer 1 **834.78 in**

Because the values presented above are totals for the 100 year period, the average annual recharge will be equal to 8.34 inches at your project location. This number can be used for long term groundwater simulations. If you want to increase the level of detail in your groundwater model, you may use specific recharge values for individual years. To get these values, prepare the **Annual Totals rate** table for all years. To study the seasonal dynamics of the groundwater, you may use **Monthly** option of the output. These and other options are described in the Visual HELP manual.

Finally, lets prepare the Visual HELP report.

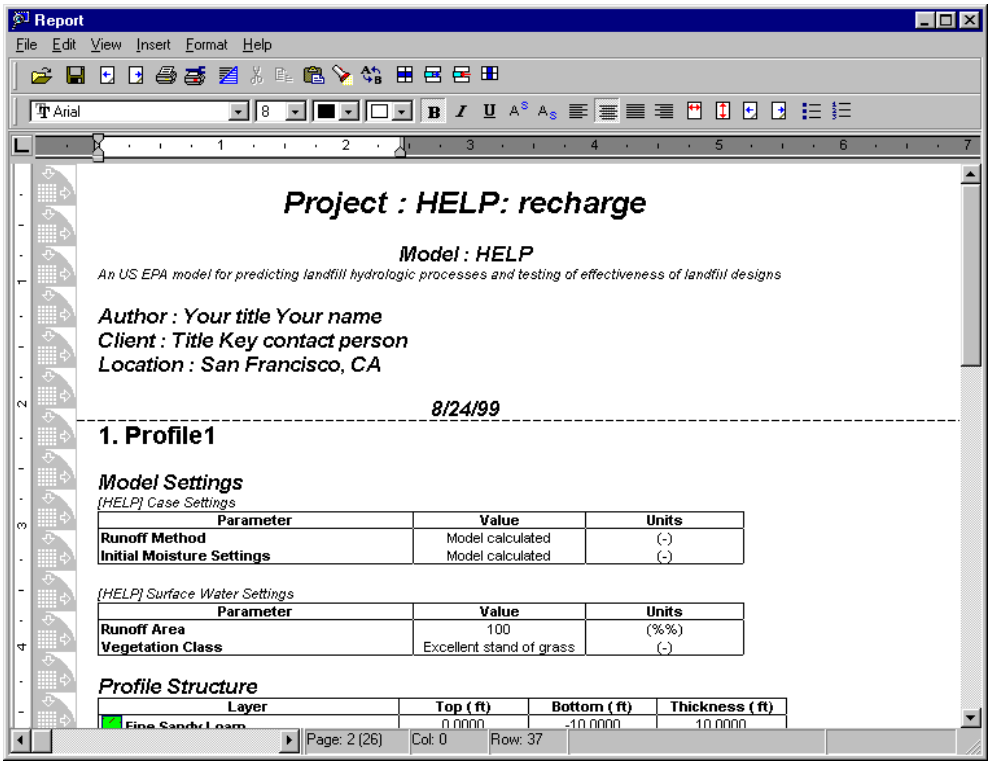
 Copyright 2000 Waterloo Hydrogeologic, Inc.

Creating a Report

As an added feature of Visual HELP, we have developed a time saving, report generator that will help you prepare professional reports of the model simulation.

To create a report and add the project input data, click the  icon from the Operational Icons toolbar.

The Visual HELP Report Generator will display your report in a separate window.



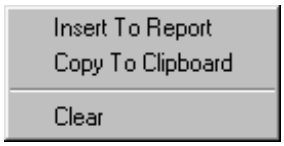
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add the table from the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the table in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The table will be inserted into the report.

To access the report (it will be hidden behind your actual Visual HELP window) in the main menu:

☞ **Project** and then select **[view report]** .

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to, insert headers /footers, change fonts/ letters size, etc.

Once you are satisfied with your report, you may print the report and/or save it for future use.

This concludes the Visual HELP demonstration exercise. Close WHI UnSat Suite Plus by clicking  button in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers for assessment of groundwater recharge. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us at (519) 746-1798 or visit our website at www.flowpath.com.

Conclusions

1) When applied intelligently, the water balance method can be a useful methodology for estimating recharge to groundwater from precipitation.
2) The HELP model (Schroeder, P., et al., 1994) in Visual HELP has been successfully applied by contemporary groundwater practitioners to substantiate historical estimates of groundwater recharge using site-specific data.
3) The results of the 100-year simulation indicate a peak recharge rate of approximately 15-inches will be observed in year 7, and a low of approximately 2-inches is expected in year 81. Average recharge to groundwater will be 8.34-inches.

References

Abraham, B., et al., 1998, A Three Dimensional Groundwater Flow Model of the Peele Dixie Well Field, Including the Florida Reprocessors Superfund Site, Ft. Lauderdale and Davie, Florida, *United States Army Corp of Engineers, Waterways Experimental Station.*

Lerner, D. et al., 1990, Groundwater Recharge: A guide to understanding and estimating natural recharge, *International Contributions to Hydrogeology: Vol. 8, International Association of Hydrogeologists. Heise Publisher, Hannover, Germany.*

Schroeder, Paul R., et al., 19994, The Hydrological Evaluation of Landfill Performance (HELP) Model, Version 3. *Environmental Protection Agency Report EPA/600/R-94/166.*

Simmers, I (ed.), 1997, Recharge of Phreatic Aquifers in (Semi-) Arid Areas, *International Association of Hydrogeologists, International Contributions to Hydrogeology: Vol. 19. A.A. Balkema Publisher, Rotterdam, Netherlands.*

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Introduction to Visual HELP

Modeling Landfill Hydrology with Visual HELP for Windows 95/98/NT/2000

The HELP Model

The **HELP** model (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile program used to design, evaluate and optimize landfill hydrology and groundwater recharge rates. The HELP model is used and recognized all over the world as the accepted standard for modeling landfill hydrology, and has become an integral component for projects involving landfill operating and closure permits.

The HELP model is a quasi-two-dimensional, multi-layer hydrologic model requiring the following input data for each model profile:

- Weather data (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil properties (porosity, field capacity, wilting point, and hydraulic conductivity)
- Design information (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can consist of a combination of natural (soil) and artificial materials (waste, geomembranes) with options to install horizontal drainage layers. The HELP model also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

About Visual HELP

Visual HELP for Windows 95/98/NT/2000 is the most advanced hydrological modeling environment available for designing landfills, predicting leachate mounding and evaluating potential leachate contamination. Visual HELP combines the latest version of the HELP model (v.3.07) with an easy-to use interface and powerful graphical features for designing the model and evaluating the modeling results.

Visual HELP's user-friendly interface and flexible data handling procedures provide you with convenient access to both the basic and advanced features of the HELP model. This completely-integrated modeling environment allows the user to:

- Graphically create several profiles representing different parts of a landfill,
- Automatically generate statistically reliable weather data (or create your own),
- Run complex model simulations,
- Visualize full-color, high-resolution results, ***and***
- Prepare graphical and document materials for your report.

Built-in Database tools include:

- An International Weather Generator for synthetic generation of up to 100 years of daily values of precipitation, air temperature and solar radiation.
- Soil, waste and geomembrane database which contains properties for 42 common landfill materials.

For professional applications in landfill design, Visual HELP is the only software package you will ever need! After running through this demonstration tutorial, we trust you will agree that Visual HELP is now the most complete and easy-to-use modeling tool for designing & optimizing landfill hydrology.

Visual HELP is now available as part of the **WHI UnSat Suite Plus**. This suite of 1-D unsaturated flow models is comprised of Visual HELP, VS2DT, VLEACH, and PESTAN. Now you can model landfill hydrology, predict leachate & contaminant migration, and show degradation of pesticides all with the same easy-to-use software package! For details about the **WHI UnSat Suite Plus,** please call (519) 746-1798 or visit our web page at www.flowpath.com.

New Features of Visual HELP versions 2.1 and 2.2

Profile Viewing and Editing:

- A layer can now be split into two separate layers
- Layers are systematically numbered, and labelled in the Project Tree, which simplifies interpreation of the results

Weather Generating:

- New databases and GIS searching tools have been developed for the major regions of the world (more than 3000 weather stations)
- Data in NOAA format can now be directly imported
- Diagnostics are provided to detect missing records in NOAA files

Output Presentation:

- Output units can be changed without restarting the project
- Water Balance Tables have been added
- New tree-like interface structure allows the user to easily select results
- Report Generator (a new feature of Visual HELP) allows you to display, print and export to Microsoft Word, project input settings and output graphs and tables.

System Requirements

To run Visual HELP, you will need the following minimum system configuration:

- Windows 95/98/2000, or Windows NT 3.5.1 or later;
- IBM PC or compatible,
- Pentium 100 processor;
- CD-ROM drive for software installation;
- 32 Mb of RAM;
- 60 Mb of free hard disk space; and

 Copyright 2000 Waterloo Hydrogeologic, Inc.

- Minimum display resolution of 1024 x 768

The following fonts should be installed on your computer: MS Sans Serif, Arial, and Courier New.

How to Contact Waterloo Hydrogeologic Inc.

To contact Waterloo Hydrogeologic Inc.(WHI), please use the address below:

Waterloo Hydrogeologic, Inc.
180 Columbia Street West, Unit 1104
Waterloo, Ontario, CANADA
N2L 3L3
Phone: (519) 746-1798
Fax: (519) 885-5262

To purchase Visual HELP contact the WHI Sales Department at sales@flowpath.com or visit our web page order form at: http://www.flowpath.com/order_form.html.

Please forward additional comments or suggestions to the Visual HELP development team at:

Mikhail Gogolev, Product Manager mgogolev@flowpath.com

Demonstration Exercise

Learning Objectives

In this demonstration exercise, you will:

- Use Visual HELP as a tool for landfill design and hydrologic simulation;
- Use the Weather Generator to predict meteorological data for a twenty year period and use the predicted data as input to Visual HELP;
- Run the Visual HELP simulation, view the water balance of the landfill, assess the performance of the drainage and leachate collection system, and
- Prepare a report of the simulation data and results.

The instructions for this demo exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

The default landfill profile used in this exercise was designed in accordance with the EPA's recommendations. This design is also used in the official HELP 3.0 User's Manual. The layer properties for the default landfill profile were taken from the HELP database. The simulated landfill is located near Toronto, Ontario, Canada. The area of the landfill is 1 acre.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type- Type in the given word or value

select- Click the left mouse button where indicated

a - Press the **<Tab>** key

↵ - Press the **<Enter>** key

☞ - Click the left mouse button where indicated

☞☞- Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

NOTE: All theoretical parts of the demo tutorial are italicized

Starting Visual HELP

To start Visual HELP, you must have it installed on your computer's hard disk. Once installed, click the Windows **Start** button, choose **Programs**, navigate to the folder that contains Visual HELP, and **Visual HELP**.



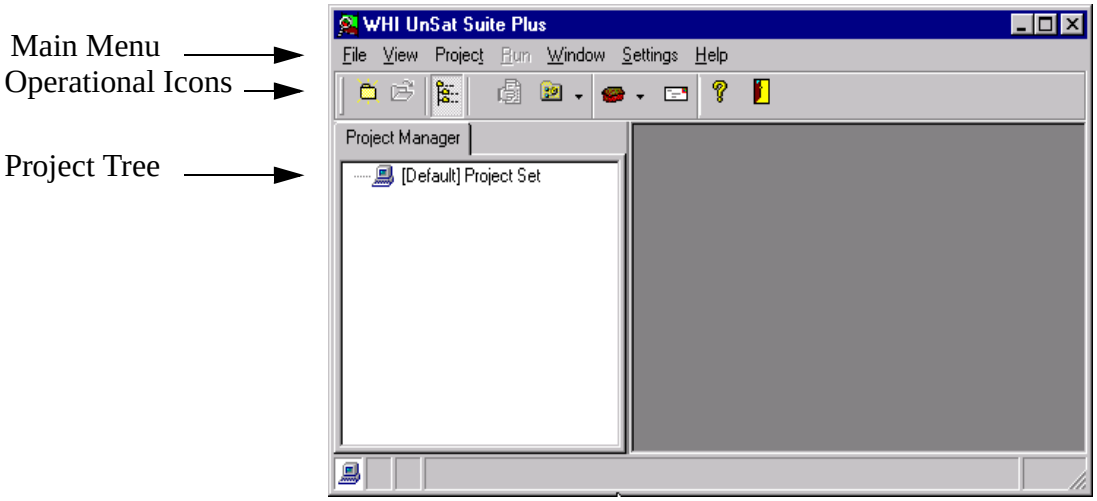
The installation program will also place an icon on your desktop, which you can also double click to start the Visual HELP demo exercise.

When Visual HELP is started an introductory screen will be displayed as shown below.



 Copyright 2000 Waterloo Hydrogeologic, Inc.

After the program is loaded, the main Visual HELP window will appear on your screen as shown in the following figure.



The Visual HELP interface structure has been designed to help you navigate through the program with ease. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the available projects, and the structure of the model profiles in each project.

Module I: Creating a New Project

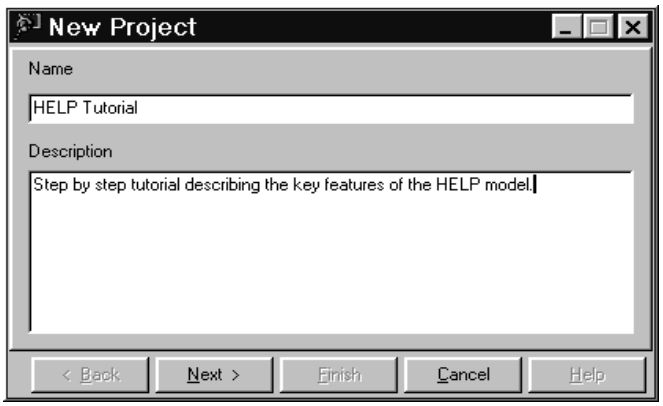
To create a new project,

☞ **File/New Project**, and the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to create a new Visual HELP project.

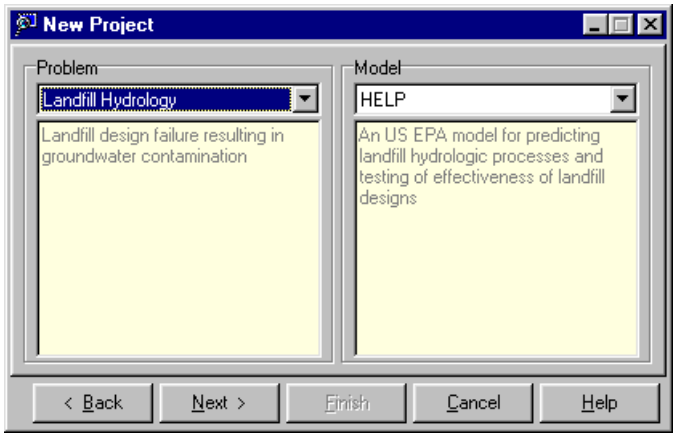
Type: HELP Tutorial, in the **Name** box.

Type: Visual HELP step-by-step tutorial in the **Description** box **.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on, and which model you would like to use to solve your problem.

☞ **[Problem]** dropdown menu and select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Model that should be used with the selected Problem type.



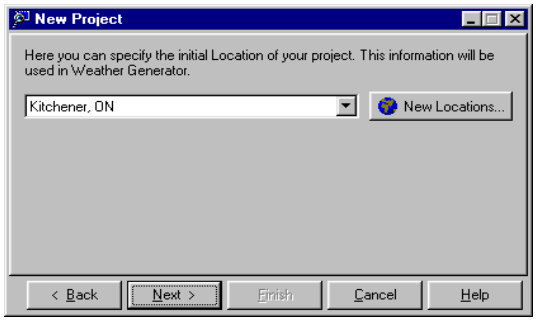
☞ **[Next]**

Once completed, the site location must be defined.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Selecting the Location

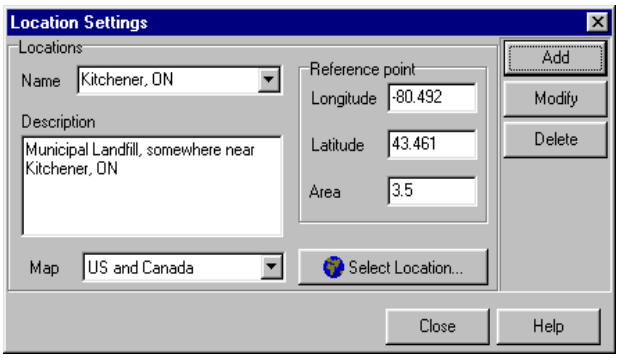
You will now select the location of your project using the GIS Location selection utility. The following dialog box should already appear on your screen.



At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



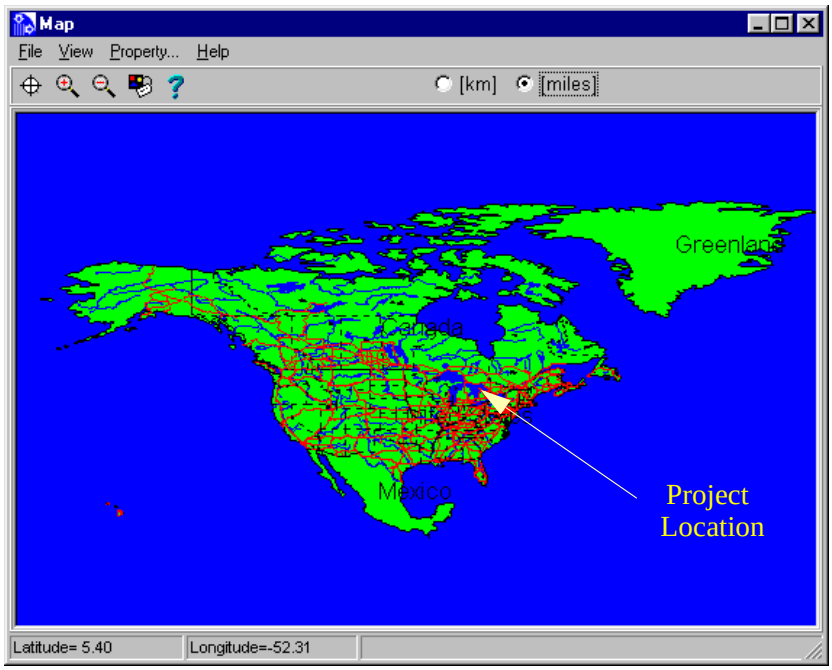
From the **Map** drop-down list,

☞ **North America,** to enable the WHI Locator, developed by WHI. The additional maps found in the list refer to various global regions that will help you to build projects from around the world!



☞ **[Select Location...],** to define our specific project location using the WHI Locator map.

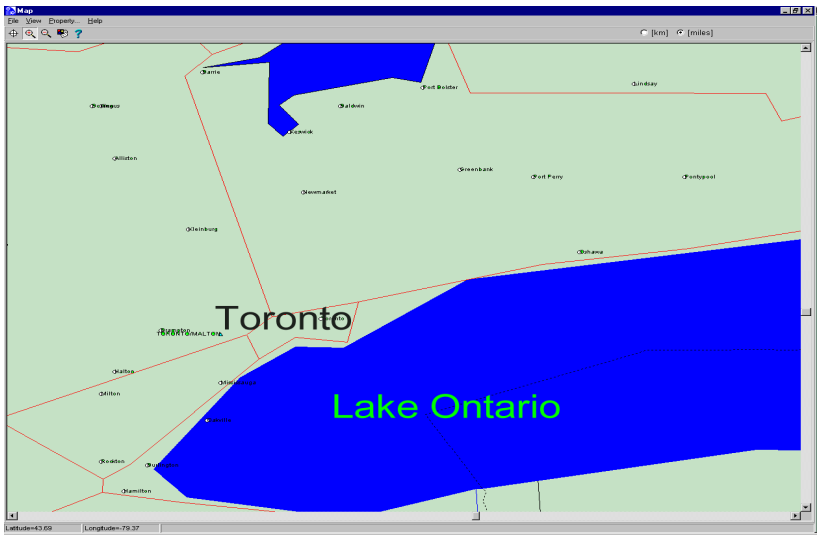
As shown below, a map of USA, Canada, and Mexico appears on your display:.



Now select Toronto, Ontario, Canada as your site location.

☞  , to zoom into the location.

Move the mouse to the right of the region where the landfill project is located (in center of the Great Lakes region). The coordinates, seen in the bottom left of the screen, should read approximately Latitude: 46.8, Longitude:-84.7. Press the left mouse button and stretch a zoom rectangle to the right of the Lake Ontario, release the button (approximate Latitude: 43.3, Longitude:-76.2). After one or two zooms, your screen should resemble the figure below.

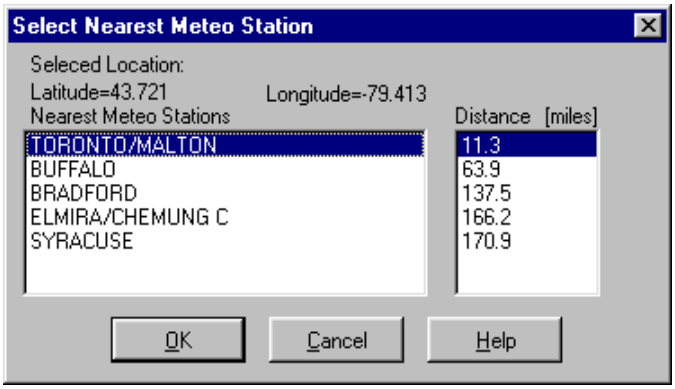


Now select **Toronto** to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Move the crosshair to the spot on the map where **Toronto** is located and click on it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you wish to calculate the distance in kilometers, simply click the appropriate option button at the top of the map.

In the "Select Nearest Meteo Station" dialog box,

☞ **[TORONTO/MALTON],** as the nearest weather station.

☞ **[OK]**

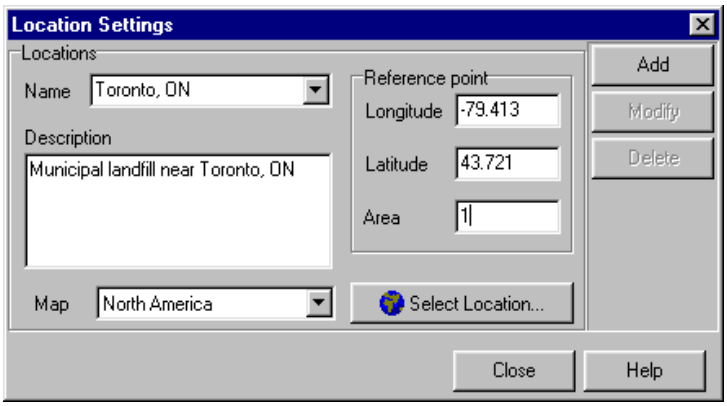
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: Toronto, ON., in the **Name** box.

Type: Municipal landfill near Toronto, ON., in the **Description** text box.

Type: 1 , in the landfill area text box: **1**.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database for future use.

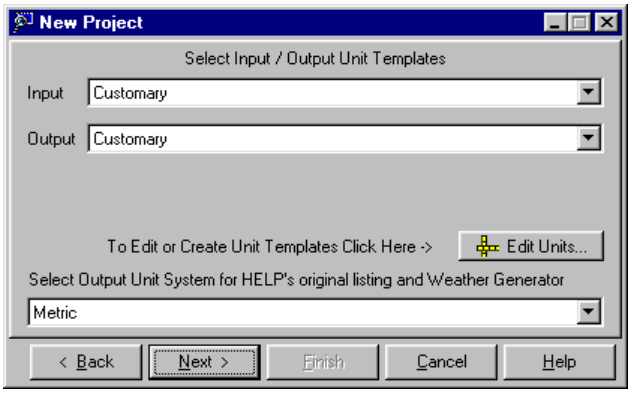
☞ **[Close]**

In the **New Project** dropdown list, select **Toronto, ON**.

☞ **[Next]**

Selecting Units

The next step is to define the units for your project (e.g. Metric or Imperial). This is completed in the "**Select Input/Output Unit Templates**" dialog box. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.



Visual HELP's unit converter capabilities were developed to maximize the flexibility for data input and output unit conversion. To see the range of possibilities, just view the list of available units for hydraulic conductivity by selecting the **[Edit Units]** option.



For this demo exercise we will specify all input units as **Metric**, and output will be in **Customary** units.

From the **Input** drop-down list,

☞ **Metric**

From the **Output** drop-down list

☞ **Customary (in)**.

From the **Select Output Unit System for original listing and Weather Generator** list box,

☞ **Metric**

☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

All the project information will be presented, as seen in the figure below.



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings are specified, the **New Profile Wizard** will open.

Profiles and Multiple Profiles in Visual HELP. *For the purpose of hydrological simulations, a profile represents a part of a landfill that is assumed to have the same cross-section throughout. The profile contains all of the layers of the landfill that it represents. It may also contain details of engineering components such as subsurface drainage, leachate recirculation systems, geomembranes, geonets, and composite liners. The surface of the profile can be sloped, which is typical for the peripheral parts of the landfill, or flat, which is typical for the central part of the landfill. The slope of the landfill usually mimics the shape of the waste body. At the time of landfill closure, the waste layer at the periphery of the landfill is usually sloped and the entire landfill site is capped by several layers, which form the landfill cap. Although drainage pipes are not usually installed in the landfill cap, permeable sand layers in the cap can transport water to the bottom drain or sump that surrounds the landfill. This process can be simulated with Visual HELP.*

A typical landfill can be represented as a set of profiles. This is done by creating several profiles in one project. This is useful because it allows you to use one profile to simulate the middle of the landfill, and several other profiles to simulate the edges, where the cross-section is tapering.

Selecting an Existing Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:

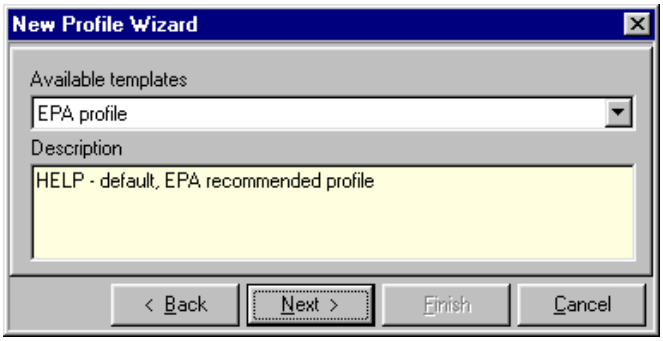


In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the default profile template.

☞ **[use existing profile template]**

☞ **[Next]**

The **Available templates** list will appear:



☞ **EPA Profile**

☞ **[Next]**

All the profile information will be presented:.



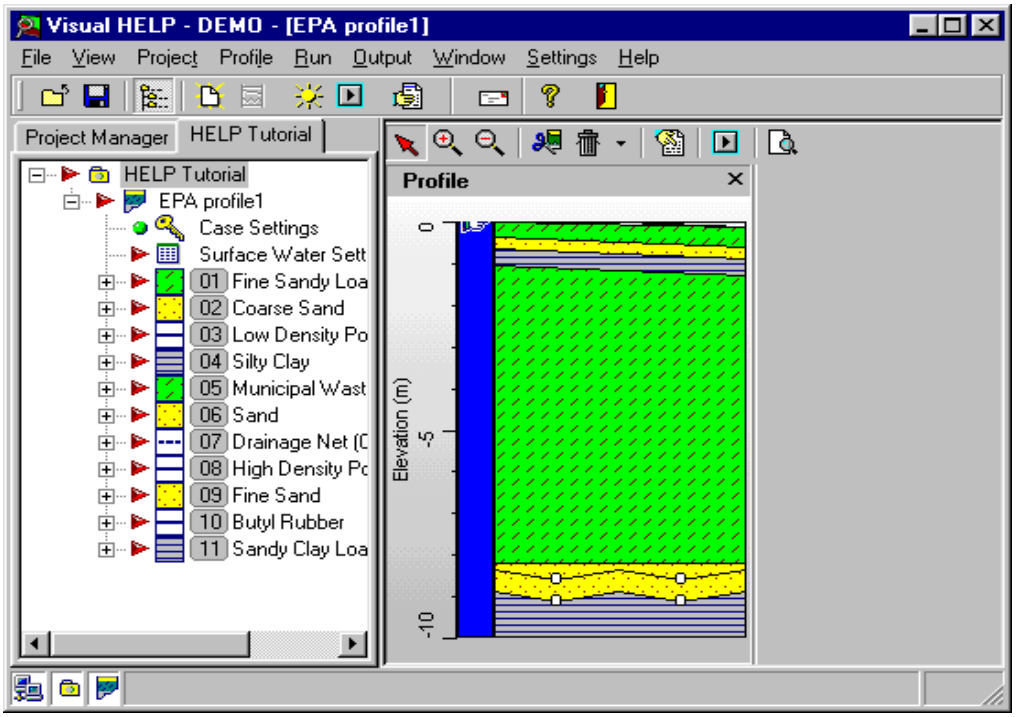
☞ **[Finish]** to load the profile.

The new project details will be added to the project tree. This concludes creating a new project using Visual HELP. We will now continue with designing the landfill profile.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Module II: Designing a Landfill Profile

After you create the project, the following picture will appear on your display.



Viewing the Default Profile

In the Profile View, located in the center part of the Visual HELP window, you will see the loaded default landfill profile with a depth scale.

A description of each landfill layer will appear when the mouse cursor is pointed at the layer.

As you can see, this profile consists of:

- a sloped landfill cap in the depth interval 0.0 - 1.2 m, consisting of fine sandy loam, coarse sand and silty clay
- a 7 m thick waste layer in the depth interval 1.2 - 8.2 m; and
- a double liner leachate collection and removal system in the depth interval 8.2-9.6 m.

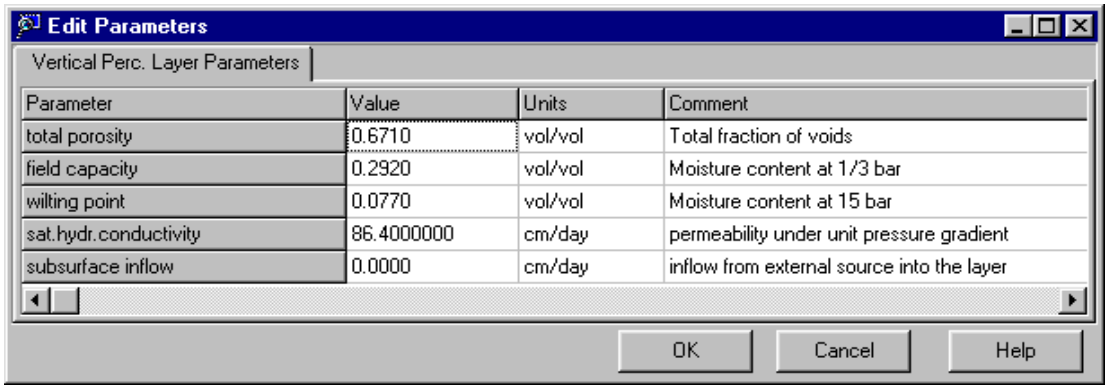
The profile construction can also be seen in the **Project Tree View** in the left part of the screen.

The Project Tree View is also used to view and modify the parameters of specific layers.

☞ **'+'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

☞ ☞ **Vertical Percolation Layer Parameters**. The **Edit Parameters** dialog box will appear.

The **Edit Parameters** menu allows you to modify the values of the parameters associated with the selected layer.



☞ **[Cancel],** in the **Edit Parameter** dialog box to exit.

☞ **'-'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

As you can see, the Project Tree View can be used to easily access specific Profiles, Profile Layers, and Layer Parameters.

Now that you have become familiar with some viewing functions of Visual HELP, you will learn how to edit the profile. In this part of the exercise you will:

- learn how to edit the layer structure of the landfill profile
- learn how to edit parameters of individual layers
- learn how to set initial moisture conditions and runoff parameters of the profile

Editing the Layer Structure of the Landfill Profile

To assist you in designing a landfill, Visual HELP offers many convenient tools for resizing, splitting, inserting and deleting layers. Visual HELP is a graphical user interface for the US EPA HELP model and, as such, obeys all conventions of the original HELP model.

The layers in the landfill are classified by the hydraulic function that they perform. Four categories of layers are available in original HELP:

- *vertical percolation layers,*
- *lateral drainage layers,*
- *barrier soil liners, and*
- *geomembrane liners.*

The topsoil and waste layers are generally vertical percolation layers. Sand layers above liners are typically lateral drainage layers; compacted clay layers are typically barrier soil liners. Geomembranes are classified as geomembrane liners. Composite liners are modeled as two layers. In the original HELP, geotextiles are not considered as layers unless they perform a unique hydraulic function. In Visual HELP, taking into account that this class of landfill material is growing extensively, geotextiles and geonets are defined as a separate category.

Although HELP allows a wide range of layer combinations, there are some basic rules and restrictions for the arrangement of layers in a profile that should be followed. These rules are obeyed by Visual HELP and you will get a warning if you try to delete or insert a layer incorrectly.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Resizing Layers

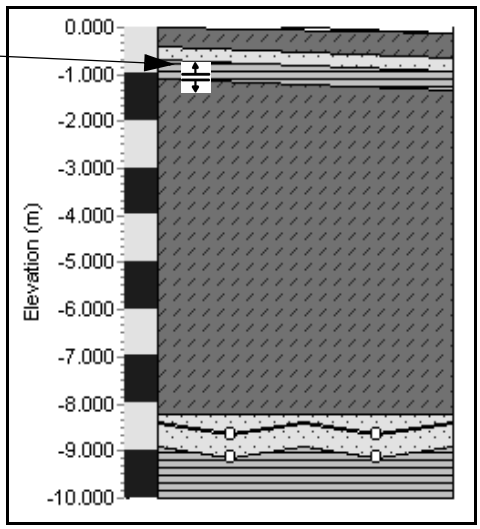
In Visual HELP, the layers are either resizable or non-resizable.

Resizable Layers *Layers built from soil and waste, grouped in the Vertical Percolation, Lateral Drainage, Barrier Soil design categories, are resizable.*

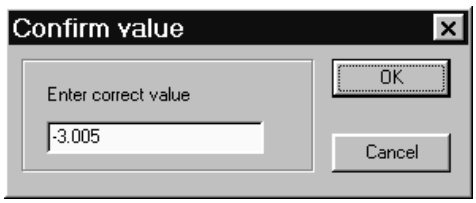
Non-Resizable Layers *Layers built from industrially produced materials with fixed thickness, grouped in the Geomembrane Liner and Geotextile and Geonet categories, are non-resizable.*

To test the resizing feature, move the mouse arrow on the **Municipal Waste (312 kg/cub.m)** layer in the profile view and move the arrow upwards to the top of the layer. As soon as the cursor encounters the layer's boundary, the cursor symbol changes:

New Cursor Symbol



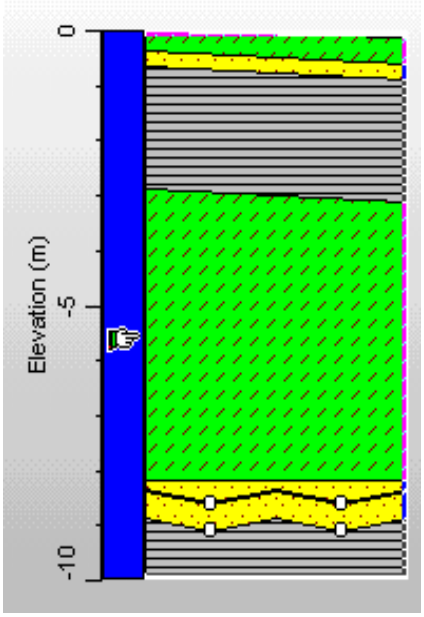
Press the left mouse button and, holding the mouse button, move the cursor downwards to approximately **-3.000** and release the mouse button. A **Confirm Value** dialog box will appear.



Type: -3.000 in the **Enter correct value** box.

☞ **[OK]**

The profile will change according to the new setting specified.



Note: Moving the layer's boundary resizes both layers which are separated by this boundary, however it does not change the total thickness of the profile. You can also change the thickness of an individual layer.

Now resize the municipal waste layer back to its original size.

Place the mouse arrow on the **municipal waste** layer in the profile view, and move the arrow up until the cursor symbol changes.

Click and drag the boundary to approximately -1.200.

Type: -1.2, in the **Enter correct value** text box.

☞ **[OK]**

Deleting a Layer

We will now examine how to delete a layer.

<Right click> the layer's name **Silty Clay**, in the Project Tree View.

.



Click the right mouse button. A shortcut menu will appear

☞ **Delete**

The layer **Silty Clay** will disappear from the Profile View. Notice an 'X' beside the **Silty Clay** layer to represent a deleted layer.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Restoring a Layer

Now let us restore the **Silty Clay** layer.

<Right click> the layer's name **Silty Clay** in the Project Tree View.

☞ **Restore** and the original layer returns.

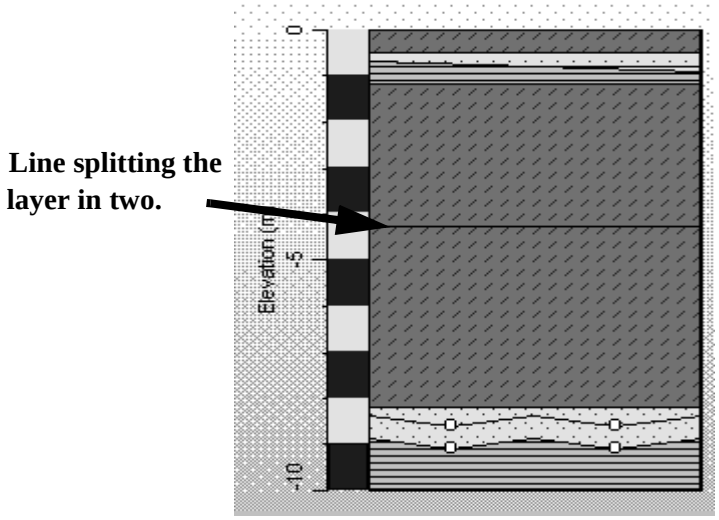
Splitting a Layer

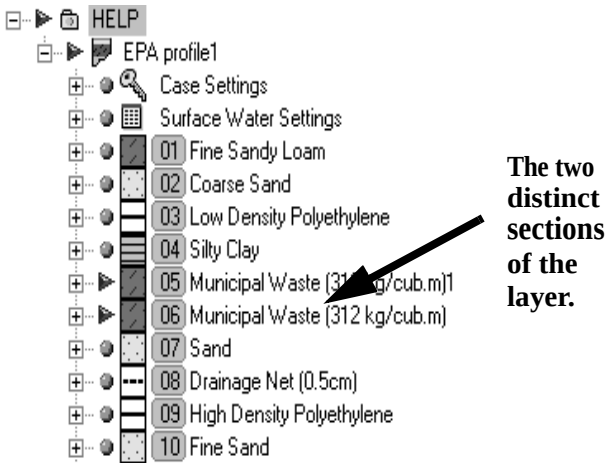
You can split a layer up into multiple sections and substitute materials for each section or assign different values of parameters for each section.

<right click> on the **Municipal Waste (312 kg/cub.m)** layer in the profile view.

☞ **Layer/Split**

A line will appear through the layer at the cursor position and a new layer will appear in the project tree. Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.



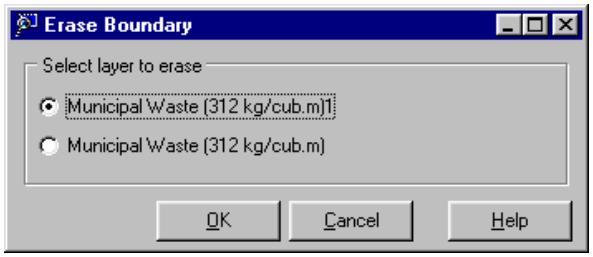


You may also substitute a material in the new layer or insert another layer between two separate parts of the layer. To return to the original profile construction, simply merge the two layers.

Place a cursor at the boundary between two parts of a layer and **<right click>**,

☞ **Merge Layers**.

In the **Erase Boundary** dialog box select the layer to erase: **Municipal Waste (312 kg/cub.m)1**



☞ **[OK]**.

Inserting a Layer

<right click> on **Sand**, in the Project Tree View

The following shortcut menu will appear



:

☞ **[Insert Layer]**

An empty **Profile Material Properties** dialog box will appear. You can now specify the new material to insert.

Note: The new layer will always be inserted above the current layer.

Note: To insert the new layer inside the existing layer, split the layer first with the split function.

Material Category Click a category from the **Material Category** list. The list will be limited to those categories that are specified by the HELP Model only.

☞ **[HELP] Barrier Soil Liner.**

Material Click the appropriate material from the **Material** list. Each layer category is associated with several textures that can be used in the design of the layer. The texture number in Visual HELP corresponds directly to the texture number in the original DOS HELP.

☞ **[Clay Loam**]

Once the material is selected, the top and bottom settings are activated.

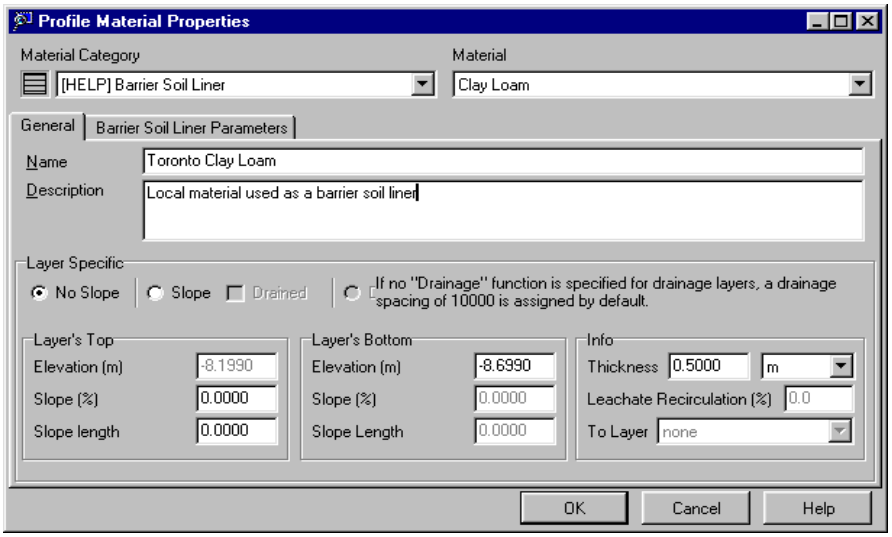
Name **Type: Toronto Clay Loam,** as the unique name for the material.

Description **Type: Local material used as a barrier soil liner**, in the comments field.

Type: 0.5 in the **Thickness** box.

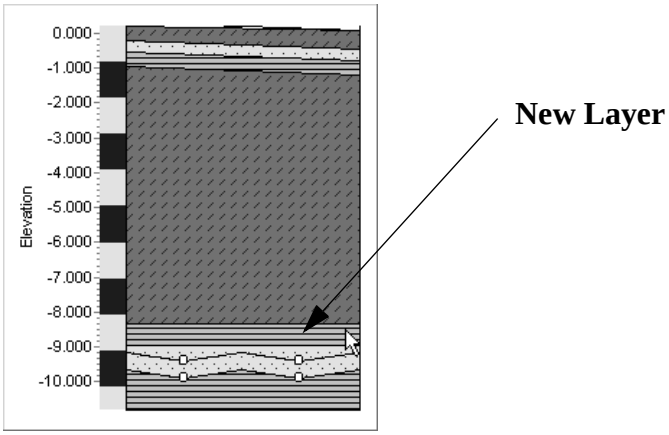
Once completed, your **Profile Material Properties** dialog box will appear as shown in the figure below:

 Copyright 2000 Waterloo Hydrogeologic, Inc.



☞ **[OK]**

The new layer is now defined:



The **Toronto Clay Loam** has now become part of the profile and you can modify it like the rest of profile layers.

To return to the original profile construction, which will be used for the rest of this tutorial, delete the recently inserted layer.

<right click> on **Toronto Clay Loam,** in the Project Tree View.

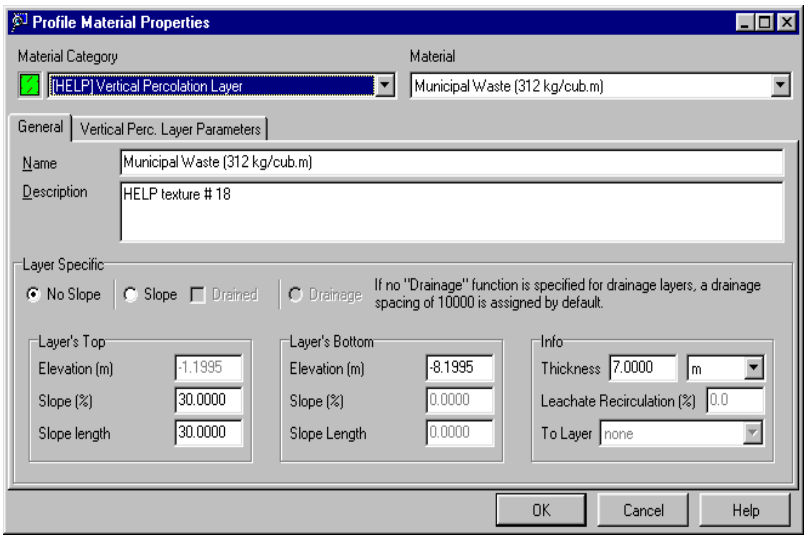
☞ **[Delete]**.

Editing Layer Parameters

The properties of a landfill material depend on the material category.

☞ ☞ the layer name **Municipal Waste (312 kg/cub.m),** in the Project Tree.

The **Profile Material Properties** dialog box will appear.



In this dialog box, you can view and edit information about the layer including;

category, material texture, slope, top or bottom elevation, and thickness.

☞ **Vertical Perc. layer Parameters** tab to edit the material properties of particular layer.

☞ **Value** beside **total porosity**

Type: 0.62

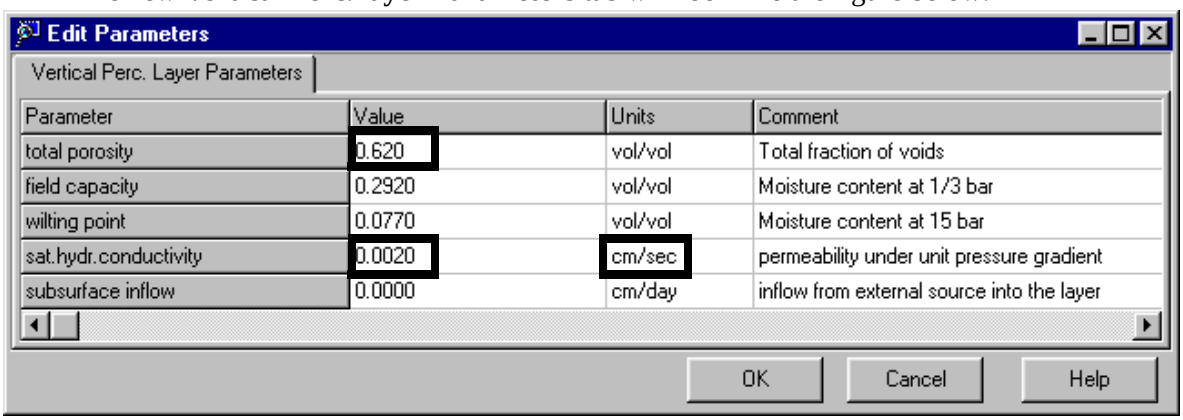
☞ **Units** beside the **sat. hydr. conductivity**, a drop-down arrow will appear.

☞ the drop-down arrow

☞ **cm/sec** from drop-down list of available units

Type: 0.002, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-] ,**beside **Municipal Waste (312 kg/cub.m)** in the Project Tree.

Now edit the properties of the Lateral Drainage layer. The layers from this category have the same hydraulic parameters as Vertical Percolation layers. In addition, the drainage and leachate recirculation parameters may be specified.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

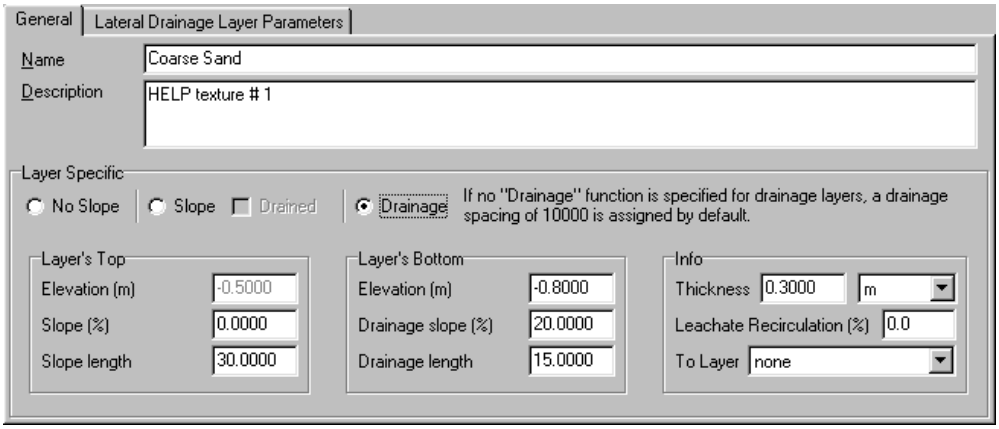
☞☞ **Coarse Sand** in the Project Tree. By default, the **Slope** function is selected for this layer.

Select **Drainage** by clicking the appropriate radial button to set the Drainage function.

Type: 20 (%) for **Drainage Slope**

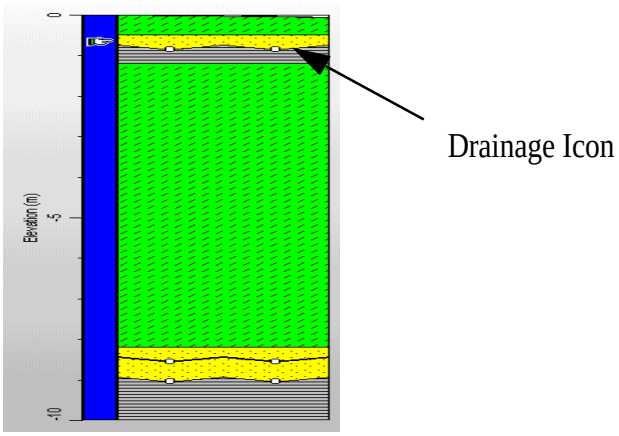
Type: 15 (m) for **Drainage Length**

Once you have made these changes, the properties for the **Coarse Sand** material should be similar to the figure below:



☞ **[OK]**

The Profile View will change to reflect the change of the **Coarse Sand** layer's status.



☞ **[-] ,**beside **Coarse Sand** in the Project Tree.

Finally, you will edit parameters of a Geomembrane Liner.

☞ **[+],** beside the **Low Density Polyethylene**

☞☞ **[Geomembrane Liner Parameters]**

☞ **Units,** beside the **sat. hydr. conductivity**

☞ the drop-down arrow

☞ **[cm/sec],** from the drop-down list.

Type: 7.00E-13, in the **Value** field.

☞ **[Units]**, beside the **pinhole density.**

☞ the drop-down arrow

☞ **[# / acre],** from the drop-down list.

Type: 10, in the **Value** field.

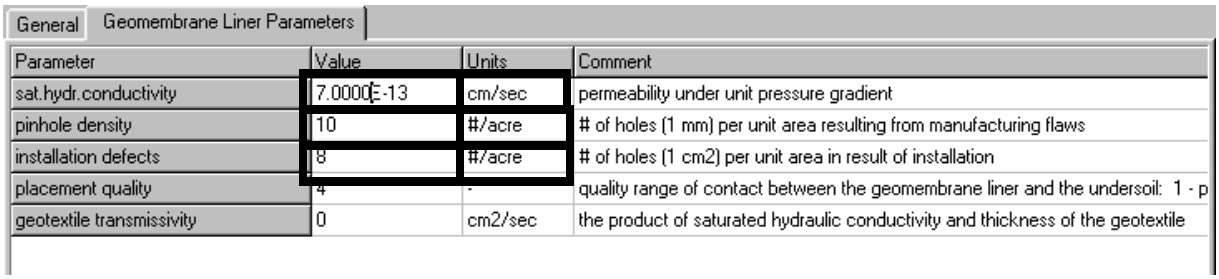
☞ **Units,** beside the **installation defects.**

☞ the drop-down arrow

☞ **[# / acre],** from the drop-down list.

Type: 8, in the **Value** field.

The **Vertical Perc. layer Parameters** tab for LDPE liner will look like the figure below.



☞ **[OK]**

☞ **[-],** beside **Low Density Polyethylene** in the Project Tree.

With this action completed, you have finished editing the parameters of individual layers of your profile.

These sample actions provide only a brief overview of the many profile design features provided by Visual HELP. Once you learn how to fully utilize all of the graphical data manipulation tools in Visual HELP, you will begin to truly appreciate the benefits of this powerful modeling tool.

Setting Initial Moisture Conditions & Runoff Parameters

Finally, you will define our landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** parameter group located in the Project Tree View.

As with the original HELP model, Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- *the model simulated values, or*
- *the user specied values.*

Depending on the selection made, runoff can be:

- *calculated by the model which will account for the type of material, slope, slope length and type of land cover,*
- *specified by the user,*
- *specified by the user and further adjusted to the slope and slope length.*

In the first case, which is the default, the model will assign realistic values for the initial moisture storage and simulate one year of landfill life. The values of moisture storage obtained from this simulation will be used as initial values.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. These functions are selected by default. To view other available options,

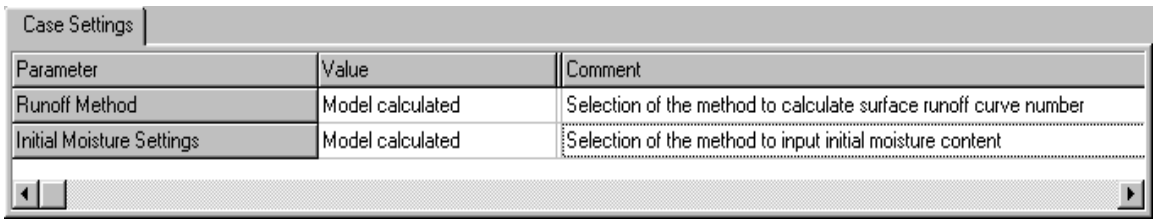
☞☞ **Case Settings** (found in the Project Tree)

☞ in the **Value** field beside **Runoff Method,**

☞ the drop-down arrow. Three options are available in the drop-down list: **User specified, User modified and Model calculated** (see the product Manual for explanation of these functions).

☞ **Model calculated.** The program will automatically pick up the slope and slope length values for the top layer and use them in the runoff calculation.

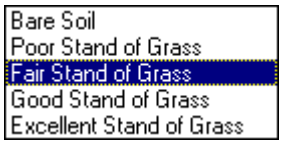
Leave the **Initial Moisture Settings** as **Model Calculated**



☞ **[OK]**

To specify additional parameters for surface runoff,

☞☞ **Surface Water Settings** from the Project Tree View.



To set the various vegetation classes, click in the **Value field**, found beside the **Vegetation Class.**

☞ the drop-down arrow. The drop down list of available types of the land cover will appear.

☞ **Fair Stand of Grass**

Note: Surface Slope and Slope Length which regulate the runoff, may be edited in the General Tab of the top profile layer.

☞ **[OK]**

The next step is to generate your site specific weather using the USDA Weather Generating program that synthetically generates daily values for precipitation, temperature and solar radiation.

NOTE: Visual HELP is also an effective tool for generating aquifer recharge values that can be used in other models such as Visual MODFLOW. To calculate accurate recharge rates for your project, simply set your model profile to represent your site specific hydrogeological conditions and run!

This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **HELP tutorial 1**, which was specifically developed for you and contains all changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

☞ ☞ project **HELP tutorial 1.**

You are now ready to generate the required weather data and, subsequently, run Visual HELP.

Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting

We will now create site specific weather data that will be considered when modeling our landfill profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill,*
- *Simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *Simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

Visual HELP includes a built-in Weather Generator for synthetic generation of daily values of precipitation, mean temperature, and solar radiation. The WHI International Weather Generator includes a global database with data from more than 3000 stations and a GIS feature for locating the nearest stations globally

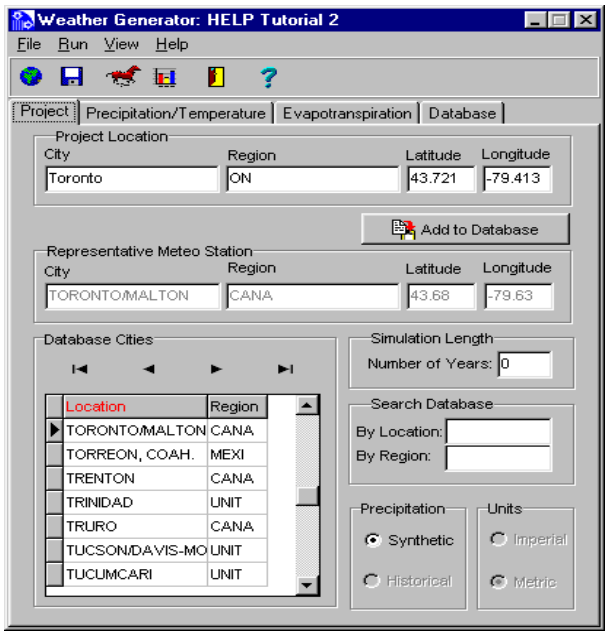
To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the top toolbar

☞ **Weather Generator**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

After an introductory splash screen, the Weather Generator dialog box will appear.



For this tutorial, you will be using data for the TORONTO/MALTON weather station. The TORONTO/MALTON station was selected among five stations located nearest to our landfill site.

Parameters for TORONTO/MALTON appear automatically in the text boxes throughout the **Weather Generator** dialog boxes after it starts (if you have time, click **Precipitation/ Temperature, Evapotranspiration and Database** tabs to see the options to customize the weather parameters).

Now you will generate weather data for your site for the next 20 years.

☞ **Number of Years** text box,

Type: 20, to represent the number of years you wish to build weather data for.

To run the Weather Generator,

☞  on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  to save the generated files

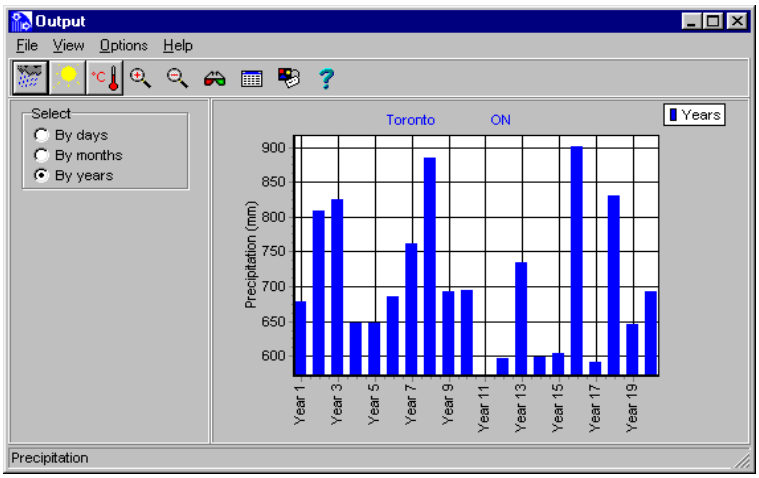
Viewing Weather Data

To view the results for the weather data you just generated,

☞  on the toolbar to view the **Output** dialog box.

☞  on the toolbar to view generated precipitation values

By default annual totals will be presented, as shown below.

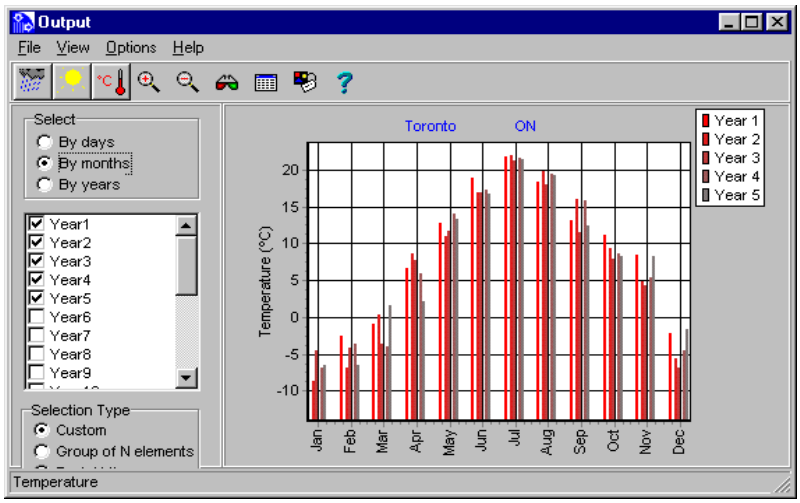


☞  to view the generated solar radiation data.

☞  to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  to view the generated raw data in table format.

☞  to close the table.

☞  to close the graphics.

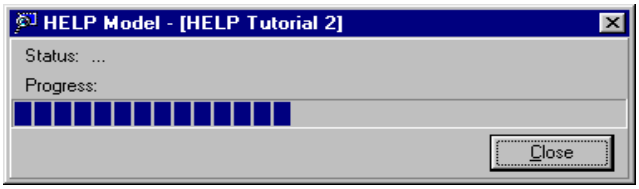
☞  to exit the Weather Generator.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

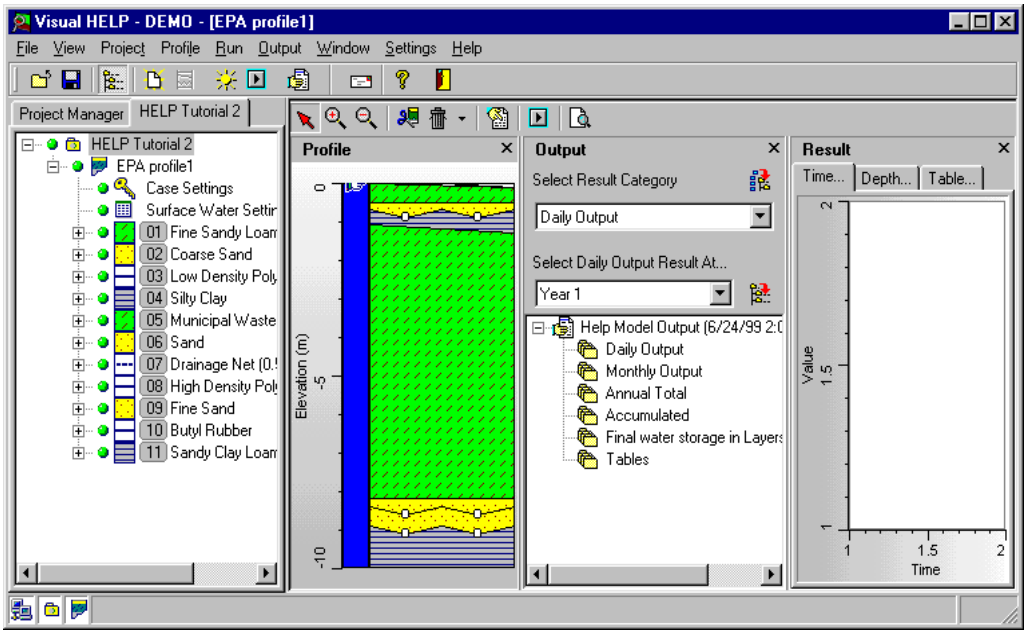


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

After the model has successfully run, Visual HELP will display the Output View and Result View windows.



☞  from the Operational Icons bar, to close the Project Manager and enlarge the graphs viewing area

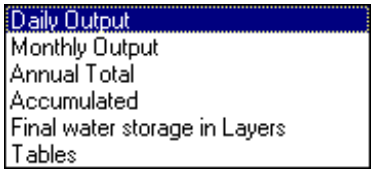
☞ **X** in the Profile View to close the window.

The **Output** View window contains two drop-down fields:

Select Result Category (used to select general result category)

Select Specified (Category Name) Result At.... (used to select specific result category)

To select the general output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



For this exercise, we recommend viewing your results with an **Accumulated** category. This plot will show you the total volumes of water drained during the simulation period. This allows you to examine the total volume of leachate that percolated through the landfill bottom during the simulated time and assess the total volumes of other water balance constituents.

☞ **Accumulated**

The results for the the **Accumulated** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox.

☞ **Select Accumulated Result at...** , to view the available balance types. ie) rate, volume

Rate means the annual rate of the balance constituent, and **volume** means the volume for the area represented by the profile.

☞ **rate**

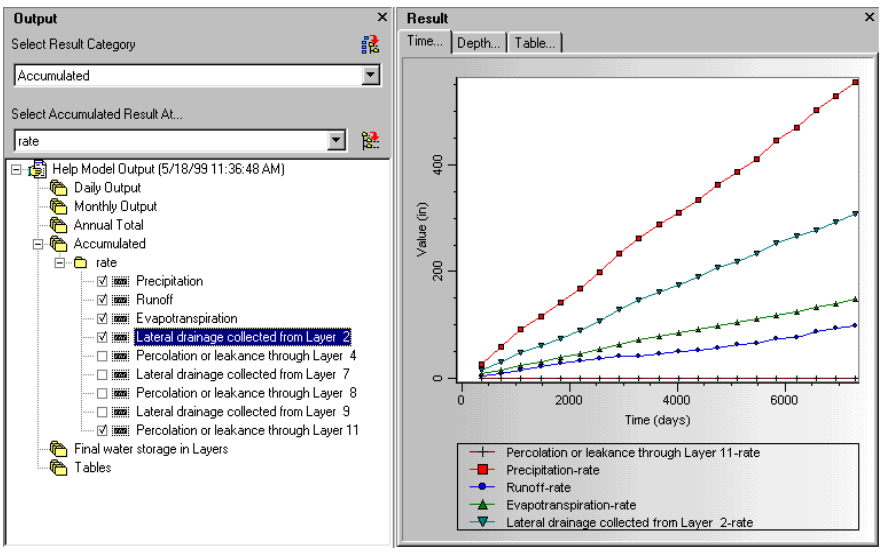


To view all the available results for this specific type of balance, click the icon to the right of the **Select Accumulated Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Lateral Drainage collected from Layer 2** (the first drainage layer), **Percolation or leakance through Layer 11** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



To erase a variable from the Result View window, deselect the corresponding check box in the Result Tree.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

From the graph, one may conclude that Runoff, Evapotranspiration, and Lateral Drainage collected from Layer 2 play an important role in the landfill balance while Percolation through Layer 11 (the landfill bottom) is equal or close to 0.

Viewing Tables

To view HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Accumulated rate**, from the **Select Table Result at..** lower listbox.

☞  **Add to Output Tree** icon to the right of the lower output listbox

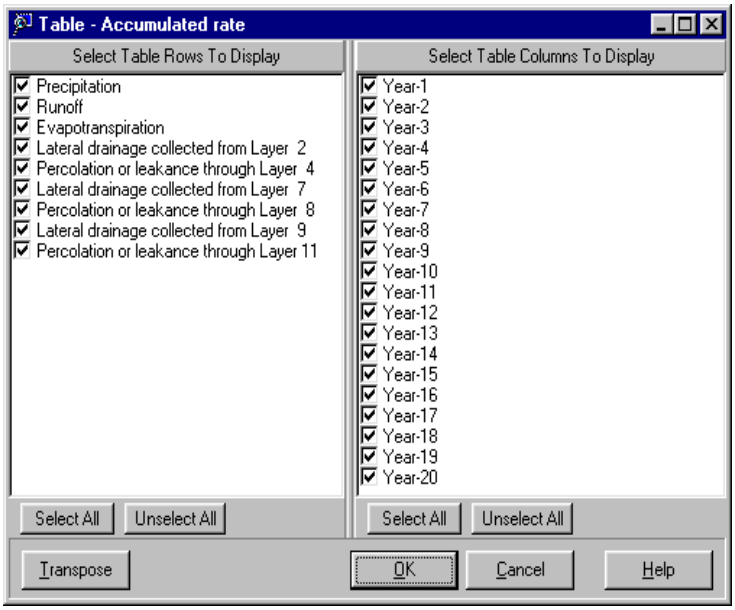
The table name **Accumulated rate** will appear in the Output Tree. For future use, let's place the table results into a new Result Window.

To do this, **<right click>** the name, **Accumulated rate** in the Output Tree. The following menu will appear:



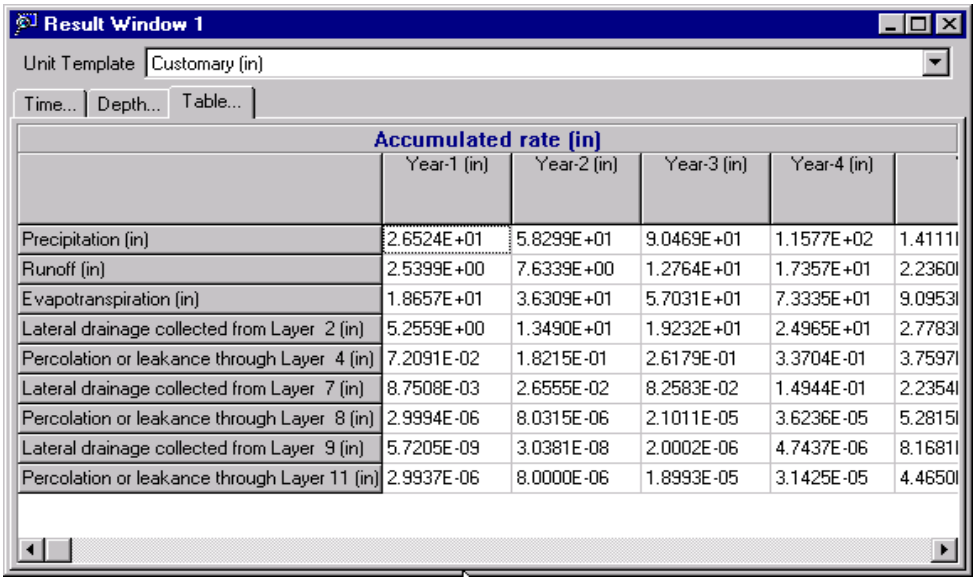
Choose **New Result Window**.

The New **Result Window 1** will open in the background and the table editing dialog box will appear.



☞ **[OK]**

The table will appear in **Result Window 1**:



	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)	
Precipitation (in)	2.6524E+01	5.8299E+01	9.0469E+01	1.1577E+02	1.4111I
Runoff (in)	2.5399E+00	7.6339E+00	1.2764E+01	1.7357E+01	2.2360I
Evapotranspiration (in)	1.8657E+01	3.6309E+01	5.7031E+01	7.3335E+01	9.0953I
Lateral drainage collected from Layer 2 (in)	5.2559E+00	1.3490E+01	1.9232E+01	2.4965E+01	2.7783I
Percolation or leakance through Layer 4 (in)	7.2091E-02	1.8215E-01	2.6179E-01	3.3704E-01	3.7597I
Lateral drainage collected from Layer 7 (in)	8.7508E-03	2.6555E-02	8.2583E-02	1.4944E-01	2.2354I
Percolation or leakance through Layer 8 (in)	2.9994E-06	8.0315E-06	2.1011E-05	3.6236E-05	5.2815I
Lateral drainage collected from Layer 9 (in)	5.7205E-09	3.0381E-08	2.0002E-06	4.7437E-06	8.1681I
Percolation or leakance through Layer 11 (in)	2.9937E-06	8.0000E-06	1.8993E-05	3.1425E-05	4.4650I

Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation	**553.98 in**
Runoff	**74.84 in**
Evapotranspiration	**359.14 in**
Lateral Drainage collected from Layer 2	**119.43 in**
Percolation or leakance through Layer 11	**0.00 in**

These values show that the default EPA landfill design (slightly modified by you during the exercise), provides good ground water protection for our project location. Having made such an optimistic conclusion, lets prepare the Visual HELP report.

☞  to close Result Window 1.

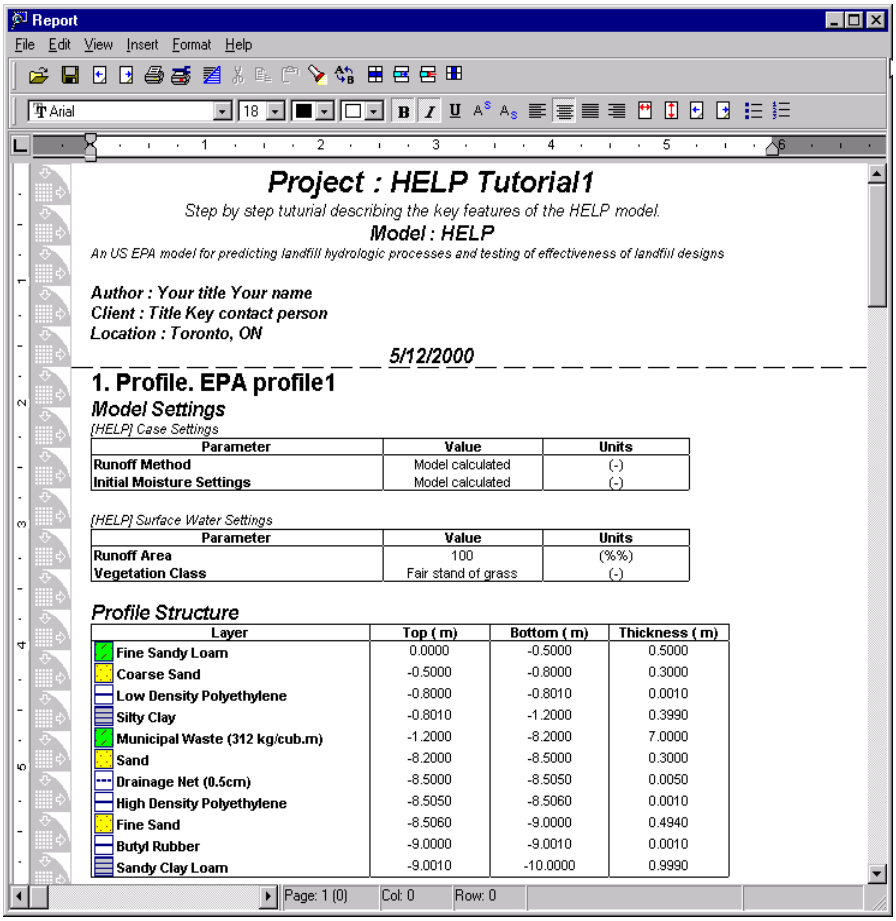
Creating a Report

As an added feature of Visual HELP, we have developed a time saving report generator that will help you prepare professional reports of the model simulation.

 To create a report and add the project input data, click the icon from the Operational Icons toolbar.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The Visual HELP Report Generator will display your report in a separate window:



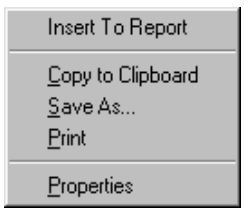
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add a graph from the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the graph in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The graph will be inserted into the report.

Go to the "Project" in the main menu and select **[view report]** to continue editing. To re-size the graph you just inserted, click the graph in the **Report** window and stretch it until it reaches the proper size.

To add a table from the **Result Window 1** to the report:

In the **Report** window place the cursor below the imported graph.

<right click> the table in the **Result Window 1**

3) ☞ **Insert To Report**. The table will appear in the report. Again, go to the "Project" in the main menu and select **[view report]** to continue editing or to view your report at any time.

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to, insert headers /footers, change fonts/ letters size, etc..

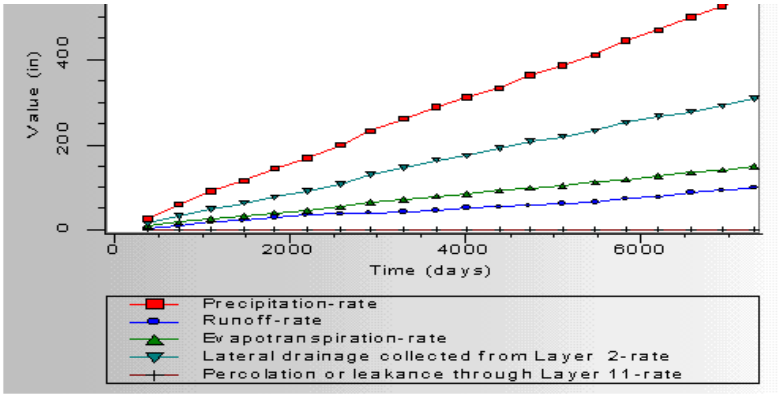


Fig.1 Accumulated balance constituents.

Table 1. Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation	2.6669E+01	5.8476E+01	9.0929E+01	1.1642E+02
Runoff	3.9219E+00	9.5287E+00	1.6193E+01	2.2387E+01
Evapotranspiration	8.2591E+00	1.5974E+01	2.4371E+01	3.1858E+01
Lateral drainage collected from Layer 2	1.4484E+01	3.1264E+01	4.8366E+01	6.2130E+01
Percolation or leakance through Layer 4	3.7569E-03	6.5516E-03	9.2846E-03	1.2289E-02
Lateral drainage collected from Layer 7	3.7532E-03	6.5417E-03	9.2717E-03	1.2271E-02
Percolation or leakance through Layer 8	3.7459E-06	8.9520E-06	1.2983E-05	1.7380E-05
Lateral drainage collected from Layer 9	3.5588E-06	8.4902E-06	1.2188E-05	1.6448E-05
Percolation or leakance through Layer 11	1.8707E-07	4.6209E-07	7.9524E-07	9.3185E-07

(continued)

	Year-5 (in)	Year-6 (in)	Year-7 (in)	Year-8 (in)

Once you are satisfied with your report, you may print the report and/or save it for future use. Let's take a look at additional output features available with Visual HELP.

Peak Daily Values

To assess the landfill hydrologic performance in extreme conditions, it is important to know when the peak hydrologic events occurred. The **Peak Daily Values** table picks up the dates of such events.

☞ **[Tables]** from the **Select Result Category** drop-down listbox.

☞ **[Peak daily values]** from the **Select Table Result at..**



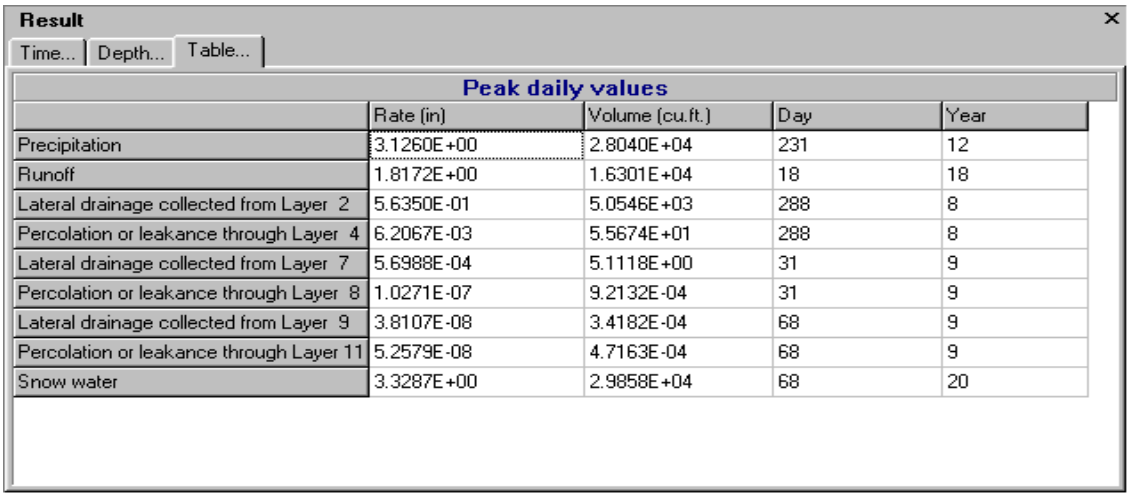
☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name "**Peak daily values**" will appear in the Output Tree. Click the check box beside it.

☞ **[OK]** in the table editing dialog box.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The following table will appear:



Peak daily values

	Rate (in)	Volume (cu.ft.)	Day	Year
Precipitation	3.1260E+00	2.8040E+04	231	12
Runoff	1.8172E+00	1.6301E+04	18	18
Lateral drainage collected from Layer 2	5.6350E-01	5.0546E+03	288	8
Percolation or leakance through Layer 4	6.2067E-03	5.5674E+01	288	8
Lateral drainage collected from Layer 7	5.6988E-04	5.1118E+00	31	9
Percolation or leakance through Layer 8	1.0271E-07	9.2132E-04	31	9
Lateral drainage collected from Layer 9	3.8107E-08	3.4182E-04	68	9
Percolation or leakance through Layer 11	5.2579E-08	4.7163E-04	68	9
Snow water	3.3287E+00	2.9858E+04	68	20

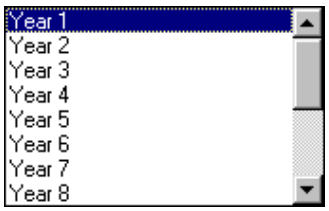
You may conclude that no leakage occurred through the bottom of the landfill (Peak daily percolation/leakage through layer 11). You may also find that the peak precipitation (3.12 in) happened on day 231 of year 12 and that peak runoff (1.81 in) happened on day 18 year 18.

To learn more about the circumstances of these events, Daily Output graphs may be studied.

Daily Output

☞ **[Daily Output]** from the **Select Result Category** drop-down listbox.

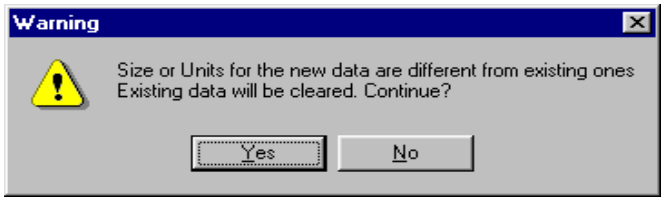
Next select the year for which you would like to have the results to be displayed from the listbox.





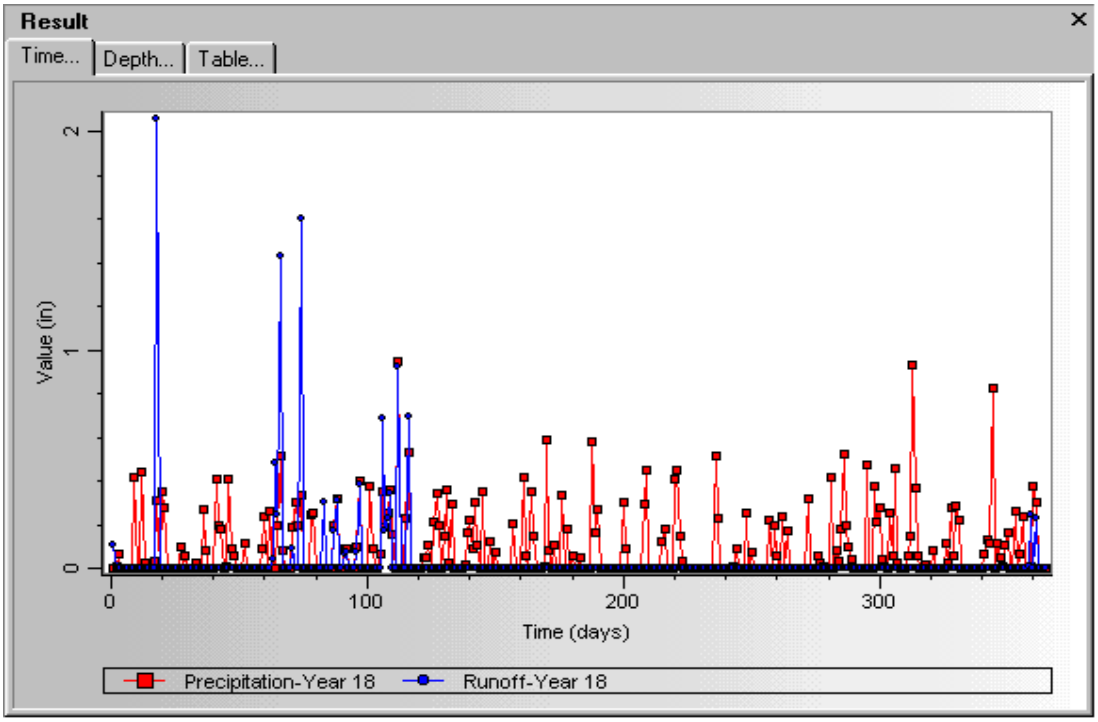
Scroll the list of years, and select the desired year: **18**. To view the available results for this specific type of balance, click the **Add to Output Tree** icon to the right of the lower output listbox.

The tree of available variables for year 18 will appear. Select **Precipitation** and **Runoff** by clicking the appropriate check boxes. This means that you are going to place these graphs to the Main Result Window. A warning will appear:



☞ **[Yes]**

The graph for the daily values of these variables will appear:



This graph shows that the maximum runoff event was not preceded by very intensive precipitation. However, after reviewing the weather data, it appears that day 18 was the first warm day with an average temperature of 5 degrees C (41 Fahrenheit) after a long span of frosty days. The peak runoff was caused by the intensive thaw of snow accumulated during the previous cold period.

Click to close the graph.

This concludes the Visual HELP demonstration exercise. Close Visual HELP by clicking in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us (519) 746-1798 or visit our webpage, www.flowpath.com.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ - Click the left mouse button where indicated

☞☞ - Double-click the left mouse button where indicated

Bold faced type indicates menu or window items to click, or values to be entered.

Text in italics explains some theoretical and methodological aspects of SESOIL simulation. These sections do not constitute the body of the exercise and are presented to provide you with additional information.

Background Information for SESOIL Exercise

The project site is a former gasoline service station located in Nashville, TN with three steel, 6000-gallon, gasoline underground storage tanks. After fifteen years of operating, an unknown quantity of gasoline was reportedly lost from the underground storage tanks. The average depth of the leakage occurrence was 10 feet, and the length of the leakage period was approximately 2 years.

Subsequently, the service station was closed and a detailed subsurface environmental investigation was performed to assess the soil contamination. The investigation identified hazardous materials primarily consisting of benzene, the main component of gasoline. Boring samples revealed a high level of benzene contamination over a 400 m^2 area. The weighted concentration of benzene in the 20-foot upper soil layer was 200 mg/kg. Additional chemicals were detected in considerable quantities within the contaminated area, including toluene, ethylbenzene, and xylene.

In this exercise, you will use SESOIL to simulate the transport and fate of benzene in solid, liquid, and gas phases through the unsaturated zone, and subsequent leaching of this contaminant to the groundwater. Benzene was chosen for simulation as the most mobile and toxic constituent with the highest concentrations at the site. Additional simulations for other waste materials may be performed in the same fashion as the benzene simulation, which spans 10 years.

The following are parameters of the profile to be simulated with the SESOIL model:

The profile consists of one layer, which has a thickness of 40 feet and is composed of loamy sand. As there is not enough data to obtain reliable site specific characteristics for all required flow and transport parameters, values from the WHI UnSat Suite database for soil and hazardous material will be used. However, some of the parameters are known and will be used to customize the input data set. The known parameters for loamy sand include:

- dry bulk density 1.38 g/cm3
- effective porosity 0.36
- fraction organic content 0.3

The aquifer underlying the unsaturated zone profile has a thickness of 10 feet, a hydraulic conductivity equal to 0.01 cm/sec, and a hydraulic gradient equal to 0.01. The initial concentration of benzene in the aquifer is equal to 0.

The SESOIL model

SESOIL is a popular US EPA model capable of simulating model water transport, sediment transport and pollutant fate.

Allowed profile structure: *homogeneous regarding flow and variable regarding transport profile with initially uneven distribution of contaminant.*

Surface: *weather boundary conditions, surface runoff, soil erosion, and pollutant washload.*

Subsurface: *variable flow, which depends on the average moisture content, and advective transport of the contaminant. Advective transport through the soil, in turn, is affected by sorption, volatilization, degradation, cation exchange, hydrolysis and metal complexation.*

SESOIL is a 1-D compartment model, as the code simulates leaching in a soil polygon. The contaminant may be present in the soil initially or may be introduced or removed from any layer at any time during the simulation.

The total mass of contaminant within each cell is partitioned among three phases: liquid (dissolved in water), vapour, and sorbed to solid surfaces.

The SESOIL model has proven to be a very efficient tool for the screening assessment of potential groundwater contamination. The ability of the model to account for the following processes makes it a unique tool that can be applied to many practical cases:

- *contaminant washload*
- *volatilization and air diffusion of the volatile organic contaminants*
- *sorption*
- *volatilization*
- *degradation*
- *cation exchange*
- *hydrolysis, and*
- *metal complexation.*

This model also has a significant number of advantages that are useful for risk assessment studies that require multi-variant simulations.

UnSat Suite allows you to quickly prepare, run and interpret SESOIL simulation via graphical output tools. SESOIL is also equipped with a database of soil materials and agricultural and industrial contaminants.

In SESOIL, complex surficial and unsaturated zone processes are grouped into three cycles:

- *Hydrologic cycle*
- *Washload cycle, and*
- *Pollutant fate cycle.*

The hydrologic cycle is simulated first, followed by the washload cycle, and subsequently the results are used in the pollutant fate cycle. The hydrologic cycle is based on a statistical dynamic theory of a vertical soil water budget formulated by Eagleston (1978), that was adapted to account for variations in soil moisture. This hydrologic cycle controls the sediment and washload cycles. The pollutant cycle simulates contaminant transport and transformation in three soil phases: soil-air or gas phase, soil-moisture phase and soil-solid phase.

SESOIL requires time-variable boundary conditions and sources, and performs calculations using constant time intervals of 30.4 days (average month). The minimum simulation length for SESOIL is one year, which starts in October and ends in September.

Starting WHI UnSat Suite

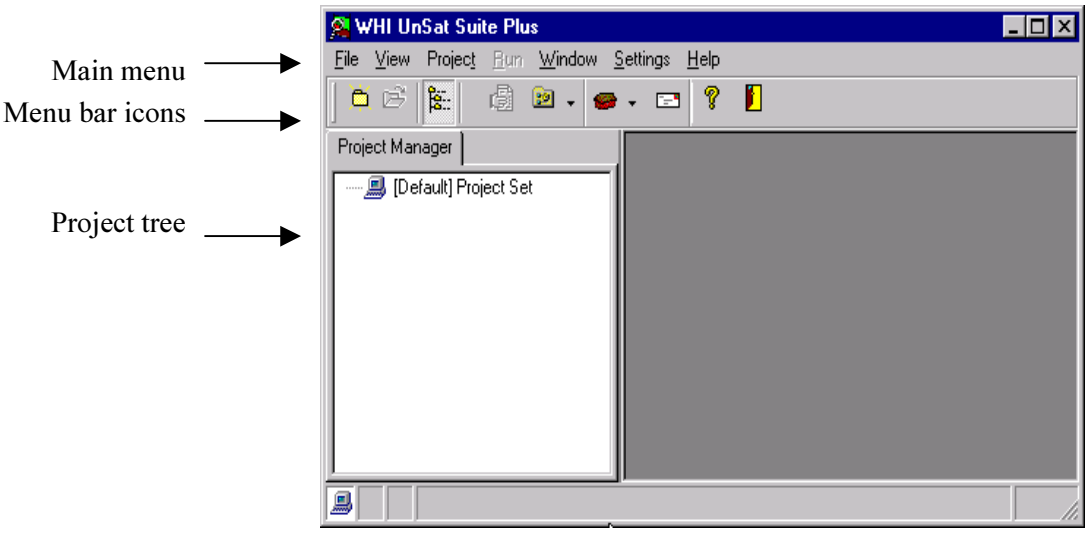
To start WHI UnSat Suite,

☞☞  (WHI UnSat Suite desktop icon)

An introductory screen will be displayed when you open WHI UnSat Suite:



After the program is loaded, the main WHI UnSat Suite window will appear on your display:



Maximize the application by **clicking** the ☐ icon from the upper-right of the window.

The WHI UnSat Suite interface has been designed to help you navigate through the program with ease.

The **Main menu** is located at the top, which contains **Menu bar icons** that provide you with quick access to many features of the program.

The **Project Tree View**, located in the left side of the screen, displays the ongoing projects and their specific structure.

Module I: Creating a New Project

To create a new project,

☞ **File** (from the top menu bar)

☞ **New Project** (from the drop-down menu)

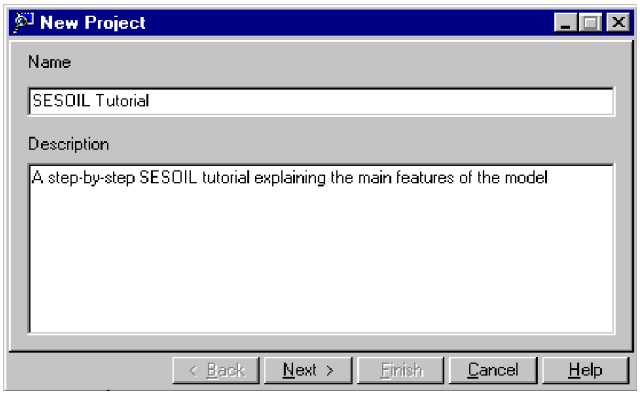
The **New Project** dialog window will open. This is the first step in the **New Project** wizard, which will guide you through the steps required to setup a new WHI UnSat Suite project.

Type **SESOIL Tutorial** (In the **Name** field)

In the Description field,

Type **A step-by-step SESOIL tutorial explaining the main features of the model**

Your dialog window should appear as seen in the figure below:



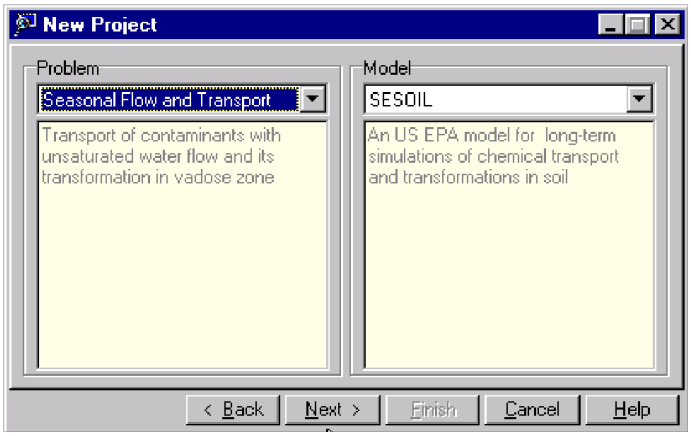
☞ **Next** (to continue with your project setup)

A **Problem/Model** dialog window will appear that allows you to select the problem type.

☞ **Problem** drop-down menu (to view problems that can be examined)

☞ **Seasonal Flow and Transport** (from the list that appears, to activate **SESOIL**)

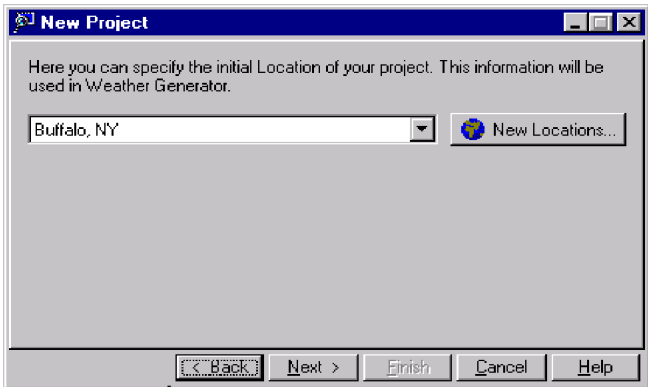
Notice that WHI UnSat Suite automatically recognizes the appropriate model that should be used with the selected **Problem** type.



☞ **Next**

Selecting the Location

We will now select the location of our project using the GIS-WHI Locator. The following dialog window should appear on your screen:

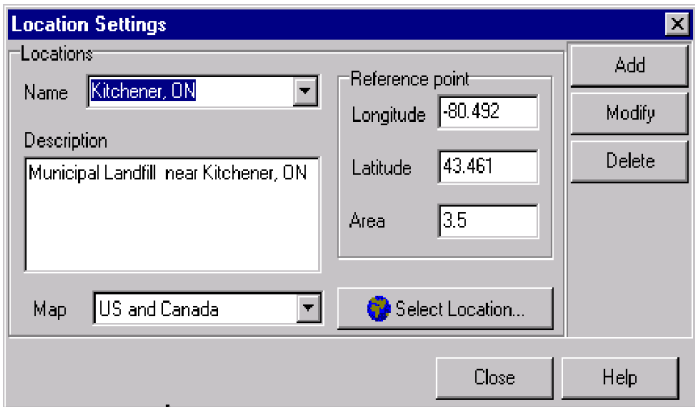


At this stage, the location list will only contain Buffalo, NY and Kitchener, ON by default.

To add a new location,

☞ **New Locations…**

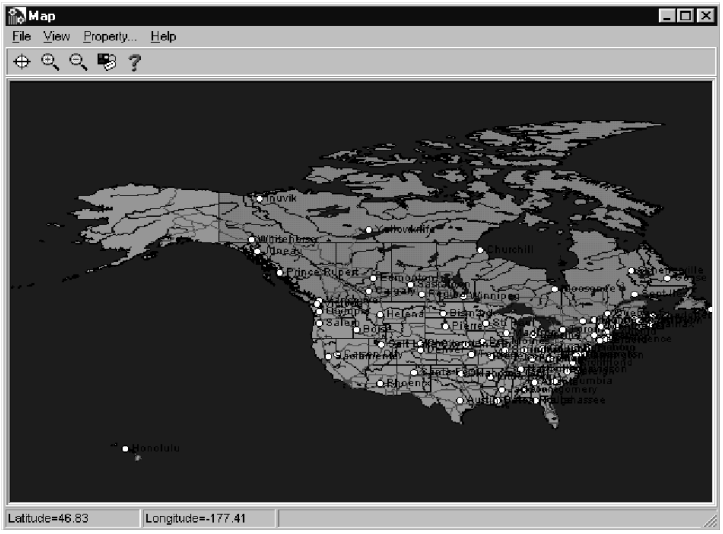
The following Location Settings dialog window will appear:



To enable the Site Locator,

☞ **Select Location…**

As shown below, a map of USA and Canada appears on your display:



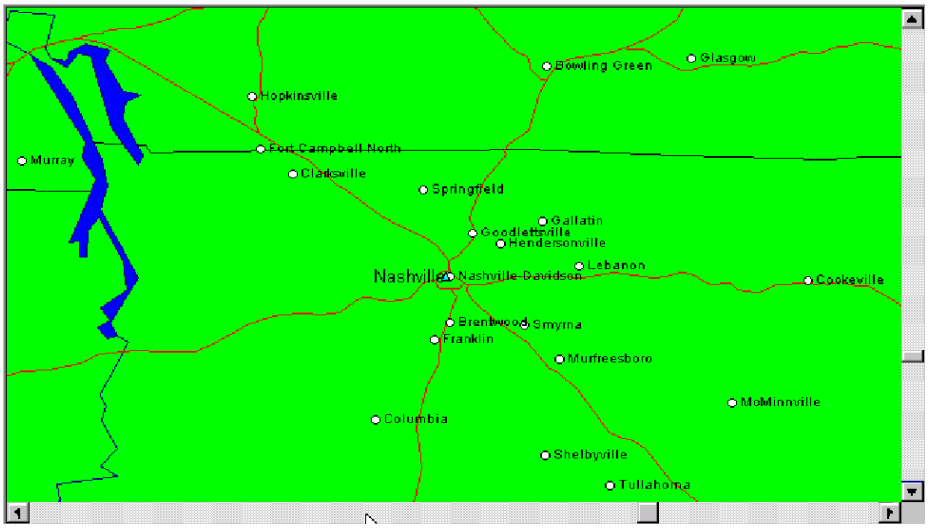
Now let's select Nashville as your site location. To zoom into our location,

☞  (**Zoom in** icon from the top menu bar)

Move the mouse to the right of the region where the project site is located (to the area between the Lake Michigan and the Gulf of Mexico). The coordinates, seen in the bottom left of the screen, should read approximately [Latitude: 38, Longitude: - 90].

Click the left mouse button and stretch a zoom rectangle to the right and down to encompass Nashville, TN, and **release** the button [~ Latitude: 34, Longitude: -84].

If necessary, perform the same operation again to zoom in further; subsequently, the figure below should resemble your screen.

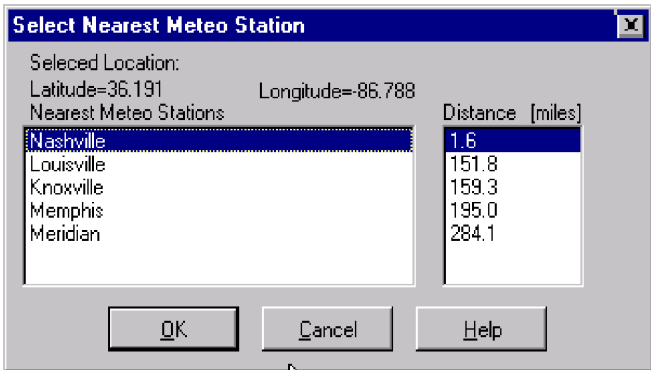


Next, you will select **Nashville, TN** to set the longitude and latitude of the project location.

To activate the crosshairs,

☞  (**Select Location** icon from the top menu bar)

Move the crosshair to the spot on the map where Nashville is located and click to the north of the city with the left mouse button. A dialog window showing the five nearest weather stations, including distance, will appear:



Note: If you click directly on the city, its location will be forwarded to the database automatically. No nearest stations will show up.

In the **Select Nearest Meteo Station** dialog window,

☞ **Nashville** (as the nearest weather station)

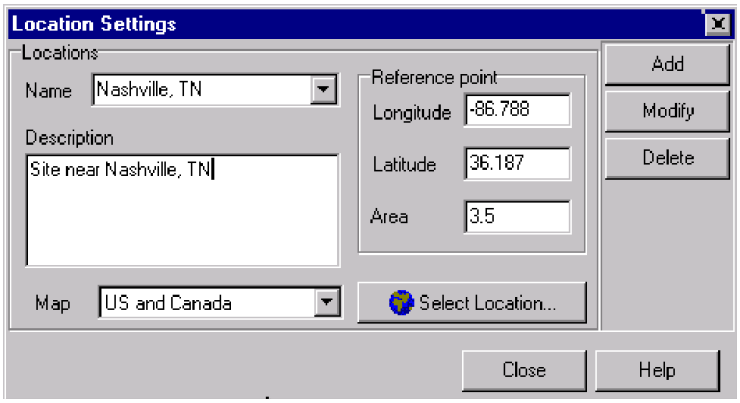
☞ **OK**

The Location Settings dialog window will show the selected weather stations and coordinates of the location you have selected.

Type **Nashville, TN** (in the **Name** field)

Type **Site near Nashville, TN** (in the **Description** field)

Finally, the **Location Settings** dialog window should appear similar to the figure below:



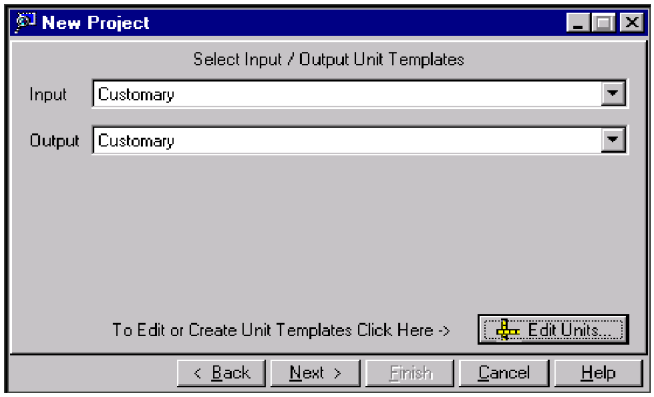
☞ **Add** (to add the new location to the database)

☞ **Close**

While back in the **New Project**, select **Nashville, TN** from the drop-down list.

☞ **Next**

A **New Project** dialog window will appear, as seen below:



Selecting Units

The next step is to define the units, which can be completed in the **Select Input/Output Unit Templates** dialog window.

We will use the **Customary** unit template, which is displayed by default. This unit template utilizes the common units used in the US.

This window also contains an **Edit Units…** button that allows you to create unit templates to fit the format of your data. The WHI UnSat Suite Unit Converter was developed to maximize the flexibility for data input and output unit conversion.

Note: To apply units which are used in the DOS version of the SESOIL program, one should select SESOIL unit template.

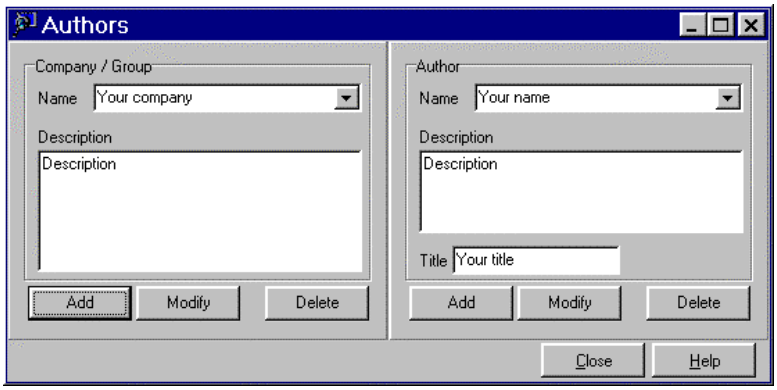
☞ **Next**

Assigning Authors and Clients

The WHI UnSat Suite allows you to create and maintain your own database of project authors and clients for use in future projects. The information that you input/specify will appear automatically in your project report. To add a new author to the database,

☞ **Authors…** (button from the **New Project** window)

The **Authors** dialog window will appear:



Each author is associated with a company.

Company/Group

In this area, you can add, modify, or delete companies and groups. To add your company to the database, in the **Name** field:

Type **Your company name**

In the **Description** field:

Type **Your address and phone number**

☞ **Add**

Author

In this area, you can add, modify, or delete authors from companies. To add a new author, input your information in the **Name**, **Title**, and **Description** fields.

☞ **Add**

☞ **Close** (to return to the **New Project** dialog window)

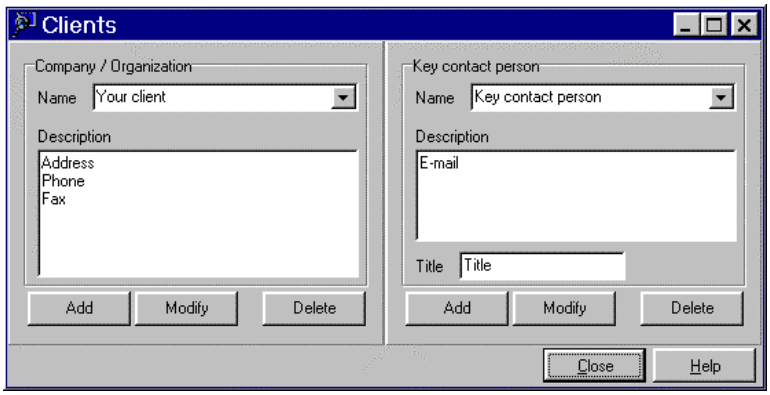
In the **New Project** dialog window, click the drop-down arrow and select your name from the list of Authors.

Clients

To add a new client to the database:

☞ **Clients…** (button from the **New Project** window)

The Clients dialog window will appear, as seen below:



Each contact is associated with a company or organization.

Company/Organization

In this section you can add, modify, or delete companies and organizations. To add a new company, edit the information in the **Name** and **Description** fields and,

☞ **Add**

To update an old company with current information, edit the information and,

☞ **Modify**

To delete a company, select the company from the list and,

☞ **Delete**

Key Contact Person

In this area you can add, modify, or delete contacts from companies. To add a new contact, edit the information in the **Name**, **Title**, and **Description** fields and,

☞ **Add**

To update the information about a client, edit the information and,

☞ **Modify**

To delete a client, select the client from the list and,

☞ **Delete**

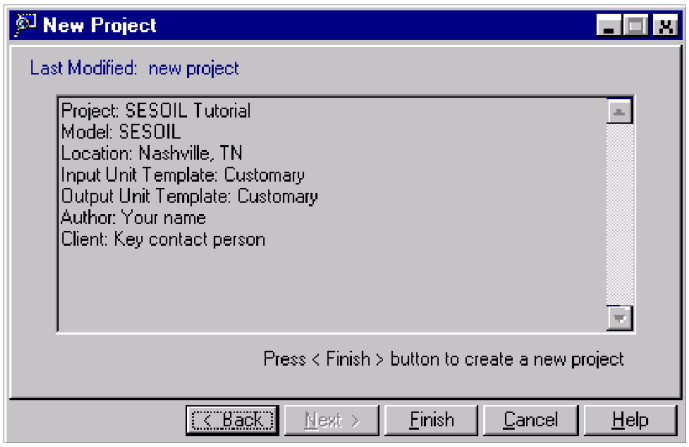
☞ **Close** (to return to the **New Project** dialog window)

Now, select an existing client from the list, or create a new client,

☞ **Clients…**

☞ **Next**

All the project information will be presented, as seen in the figure below:



To save the information as a project in the database and add it to the Project Tree,

☞ **Finish**

After the project settings are specified, the **New Profile Wizard** will open.

Profiles in SESOIL

The SESOIL model can simulate an impressive list of soil processes using easily obtainable data, which subsequently allows for site characterization. This easy-to-use, comprehensive package became available as a result of the schematization method developed by the program authors.

The SESOIL profile represents the unsaturated zone consisting of one material with homogeneous water flow. In the hydrologic cycle, the whole column is treated as a single compartment extending from the soil surface to the groundwater table.

The profile is separated into 2, 3 or 4 ***layers****, which can vary in thickness. Layers are used to setup different transport parameters and pollutant loads within the profile. As designed by the code developers, the user has to specify flow and transport parameters for the uppermost layer only. Individual transport parameter settings for the lower layers are determined by parameter value ratios for each specific layer.*

Each layer may be separated into 10 ***sub-layers*** *of equal thickness within a layer. Initial pollutant concentration may be set for each sub-layer individually. In SESOIL, flow calculations between sub-layers represents the highest detail level. In addition, these sub-layers are used to output simulation results.*

Creating a Profile

The **New Profile Wizard** that appears after you finish creating a project is shown below:



In this dialog window, you can select an already existing profile template or you can build a new profile 'from scratch'. In this demonstration exercise, you will build a new profile.

☞ **create new profile**

☞ **Next**

A dialog window with default profile parameters will appear, which we will modify.

To set the bottom elevation of the profile,

☞ In the **Bottom** text window

Type **- 40** (thickness of profile)

The value of **Thickness** will change accordingly.

☞ **Material Category** drop-down arrow (to produce a drop-down list)

☞ **SESOIL Loamy Sand group**

The default material **Loamy Sand** will appear in the **Material** drop-down list window.

Note: You may add as many materials to the specific Material Category group as you need once inside the model , in Material Designer

As a result of editing the **New Profile Wizard** window, it will appear as follows:



☞ **Next**

All the profile information will be presented:

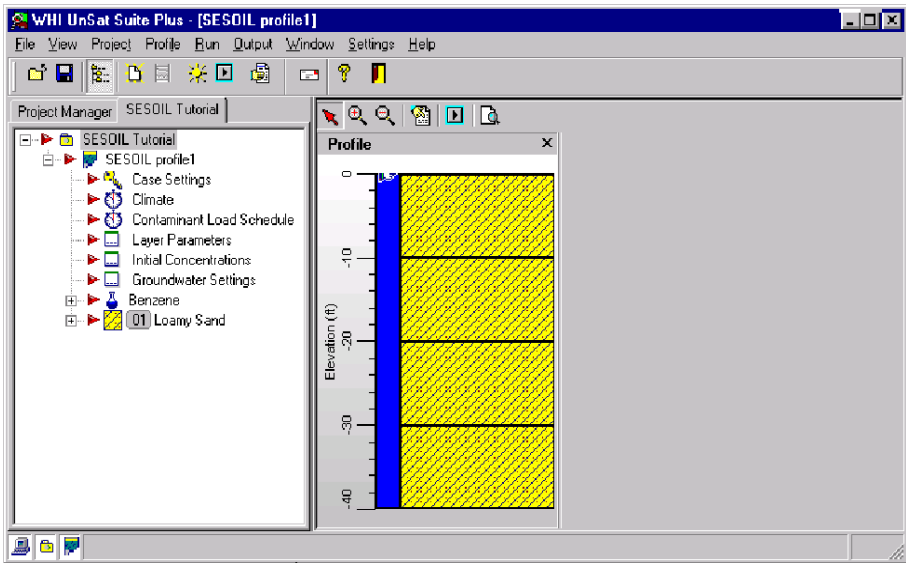


☞ **Finish** (to load the profile)

The new project details will be added to the Project Tree View, the project will open and the profile picture will appear in the Profile View. This concludes creating a new project using WHI UnSat Suite. We will now continue with designing the profile and setting specific simulation details.

Module II: Designing a Profile and Setting Case Specifications

The following figure will appear:



In the Profile View, located in the center of the WHI UnSat Suite window, you will see the loaded profile with a depth scale on the left.

A description of a layer will appear when the mouse pointer is placed over a layer.

The Project Tree View in the left part of the WHI UnSat Suite window provides access to parameters for your simulation scenario. The icon and the name of the profile layer **Loamy Sand** are located in the bottom of the list.

In this part of the exercise you will:

- learn how to set general parameters of the profile
- learn how to edit parameters of a layer
- learn how to set boundary conditions (Climate, Pollutant Load) and initial conditions for the profile
- learn how to specify groundwater conditions.

Editing General Profile Properties

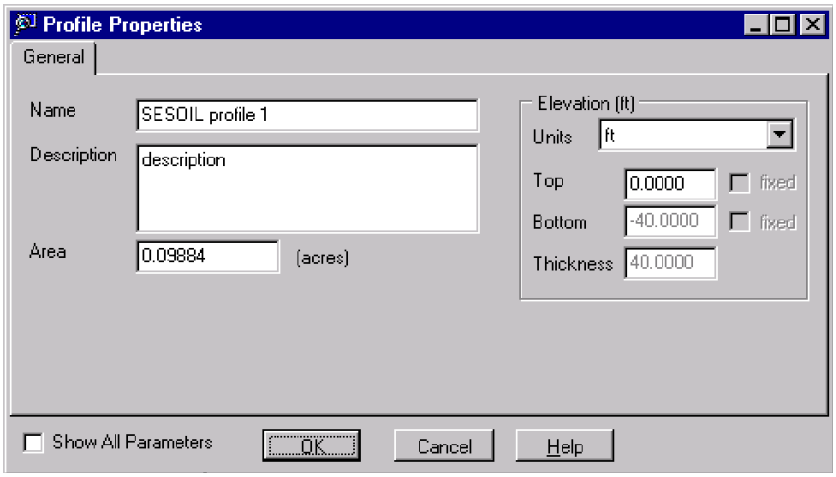
☞☞ **SESOIL Profile 1** (in the Project Tree View)

The **Profile Properties** dialog window will appear. In this window you may change units for elevation, profile name or add a profile description.

☞ **Area** text window

Type **0.09884** (the area of the contaminated spot in acres)

The **Profile Properties** dialog window will appear as seen below:



☞ **OK** (to save changes and exit)

Editing Layer Parameters

☞ '+' sign to the left of **SESOIL profile 1** (to expand the tree)

☞ **'+'** sign to the left of **Loamy Sand** (to expand the tree)

☞☞ **Soil Parameters**.

The **Edit Parameters** dialog window will appear. This dialog window contains default soil hydrologic parameters for the generic **Loamy Sand** soil type. In this window, you can view and edit general layer parameters.

Note: The WHI UnSat Suite database contains generic parameters for all 12 major soil types recognized by the USDA ARS soil classification system.

☞ **Value** field (beside **Dry Bulk Density**)

Type **1.38**

☞ **Value** field (beside **Effective Porosity**)

Type **0.36**

☞ **Value** field (beside **Fraction Organic Content**)

Type **0.3**

The parameter, **Intrinsic Permeability**, has the dimensions for area. Within the selected Customary unit template, area is measured in acres. However, **Intrinsic Permeability** is traditionally presented in sq. cm.

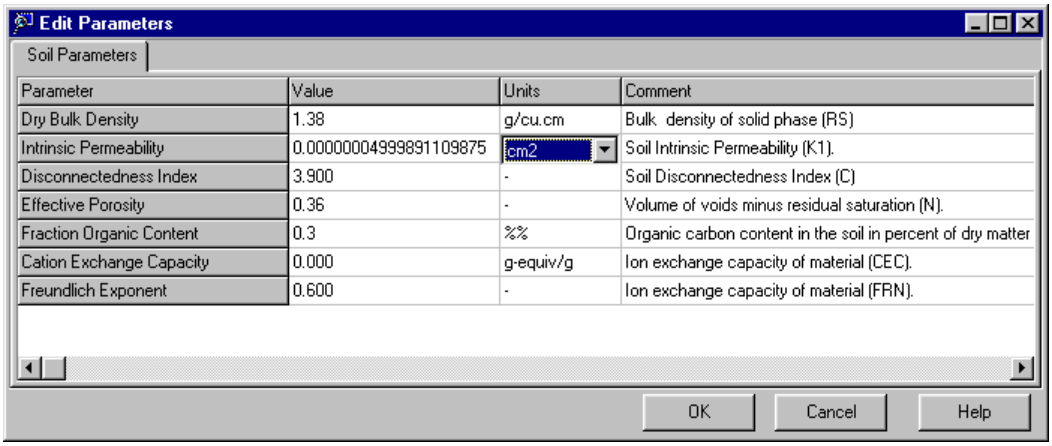
To change the units for **Intrinsic Permeability**:

☞ **Units** field (beside **Intrinsic Permeability**)

☞ **cm^2** (from the list that appears)

The value for **Intrinsic Permeability** will change to: 0.00000004999 **OR** 5.0 x 10^{-8}.

Eventually, the new **Edit Parameters** tab will appear as seen below:



☞ **OK**

☞ '-' sign (beside **Loamy Sand** in the Project Tree)

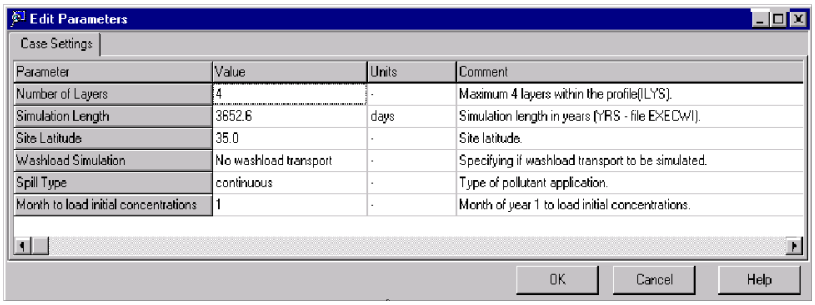
Next, you will specify boundary and initial conditions and set computational parameters for the project case.

Case Settings

Constructing the unique simulation case requires you to input several simulation parameters, principal settings and initial conditions of the profile. These parameters may be edited in the project parameter group labeled **Case Settings**.

☞☞ **Case Settings** (in the **Project Tree View**)

The **Edit Parameters/Case Settings** dialog window will appear:



The following parameters can be edited in the **Case Settings** parameter group:

Number of Layers *Total number of layers with different transport parameters and initial contaminant concentrations. The minimum number of layers is 2 and the maximum number is 4.*

Simulation Length *Total simulation length in years.*

Site Latitude *Site latitude.*

Washload Simulation *Toggles the subroutine simulating soil erosion and contaminant washload On/Off.*

Spill Type *Specify contaminant loading as either instantaneous spill or continuous loading.*

Month to load initial Concentrations *Month during year 1 when initial contaminant concentrations are introduced.*

☞ **Units** field (for **Simulation Length**)

☞ **years** (from the list that appears)

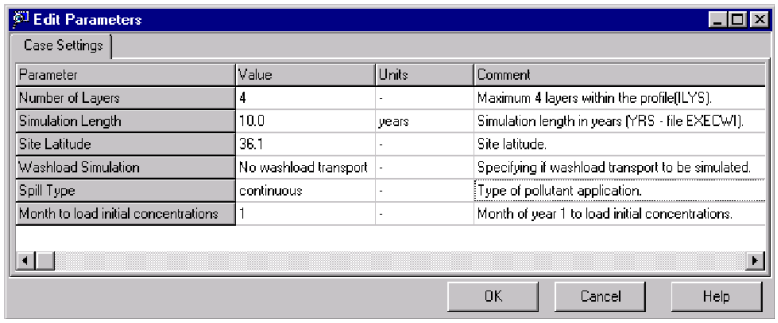
The number of years to simulate will display the simulation length, 10.0 years.

☞ **Value** (field beside **Site Latitude**)

Type **36.1** (latitude of Nashville)

The remaining parameters will remain unchanged, as we are not going to simulate pollutant washload.

Finally, the **Case Settings** parameter group will appear as seen below:



☞ **OK**

Specifying Climate

SESOIL requires you to specify monthly climate parameter values for the entire simulation length. In the original SESOIL model, the user had to input the required data manually. However, WHI UnSat Suite allows site specific data to be imported for nearly any location on the globe. This opportunity becomes available when the WHI Global Weather Generator is used in conjunction with SESOIL.

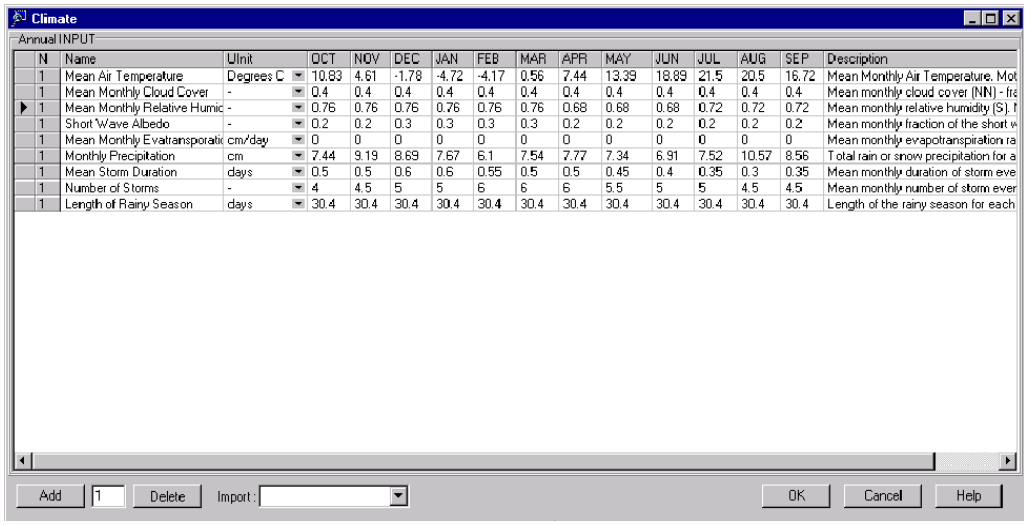
To set climate conditions, ***right-click*** on **Climate** in the Project Tree,

☞ **Edit** (from the pop-up window that appears)

OR

☞☞ **Climate** (in the Project Tree)

The following dialog window will appear:



This dialog window allows you to easily input and edit the appropriate number of annual climatic data sets.

Each annual set must contain the following parameters specified for each month:

Mean Air Temperature *Mean monthly air temperature (Degrees C).*

Mean Monthly Cloud Cover *Mean monthly fraction of cloud cover, ranging from 0.0 to 1.0.*

Mean Monthly Relative Humidity *Mean monthly fraction of relative humidity, ranging from 0.0 to 1.0.*

Short Wave Albedo *Mean monthly fraction of short wave albedo ranging from 0.0 to 1.0.*

Mean Monthly Evapotranspiration Rate *Mean monthly fraction of the evapotranspiration rate (cm/day).*

Monthly Precipitation *Total monthly precipitation (cm).*

Mean Storm Duration *Mean monthly duration of storm event (days).*

Number of Storms *Mean monthly number of storm events.*

Length of Rainy Season *Length of rainy season for each month of the year (days). For most regions this parameter should be set to 30.4, which means that rain may occur any day within the month.*

The SESOIL model applies certain relationships between climate parameters. The parameters, ***Mean Air Temperature****,* ***Mean Monthly Cloud Cover****,* ***Mean Monthly Relative Humidity*** *and* ***Short Wave Albedo,*** *are used only to simulate* ***Mean Monthly Evapotranspiration Rate.*** *This calculated rate is used by the model only if the* ***Mean Monthly Evapotranspiration Rate*** *is equal to zero.*

Based on the SESOIL code requirements, WHI has developed a technology that allows the user to easily set site specific climate data. There are four options to input climate data in SESOIL within the WHI UnSat Suite:

1. *input data manually*
2. *input data from the Weather Generator database*
3. *input weather data synthetically generated in SESOIL*
4. *import synthetically generated weather data and evapotranspiration from HELP.*

Each option requires a series of different steps that ultimately produces a set of climate data. Option 1 is the most time consuming, however it may produce the most reliable input data set. To achieve this option, the user must know all the required climate parameter values.

With options 2, 3, and 4 the user utilizes the advantages of the Global Weather Generator. In this exercise you will use option 3 - ***input weather data synthetically generated in SESOIL.*** *This option allows you to synthetically generate site-specific statistically reliable data for the entire simulation length with the Weather Generator. As well, this option allows you to assess values of* ***Mean Air Temperature, Total Monthly Precipitation*** *and* ***Number of Storms.*** *The remaining five parameters (except* ***Mean Monthly Evapotranspiration Rate****) have to be input manually.*

To use option 3 (input weather data synthetically generated in SESOIL):

☞ **Cancel** (to close the Climate parameter group)

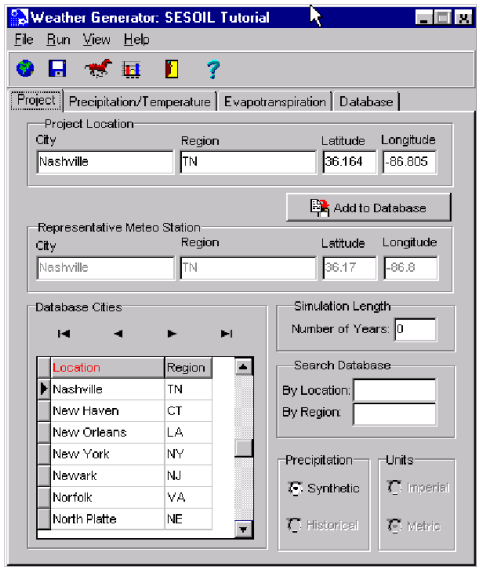
☞ **Run** (from the top menu bar)

☞ **Weather Generator** (from the pull-down menu)

OR

☞  (**Weather Generator** icon from the top menu bar)

After an introductory screen, the Weather Generator main dialog window will appear:



Let's run the Weather Generator for **N+1** years, where **N** is the specified simulation length.

☞ **Number of Years** (under **Simulation Length**)

Type **11** (the number of weather data years you require)

To run the **Weather Generator**,

☞  (to run the **Weather Generator**)

The **Weather Generator** will begin computations.

☞  (once completed to save the generated files)

☞  (to exit the **Weather Generator**)

To open the **Climate** Annual Input table,

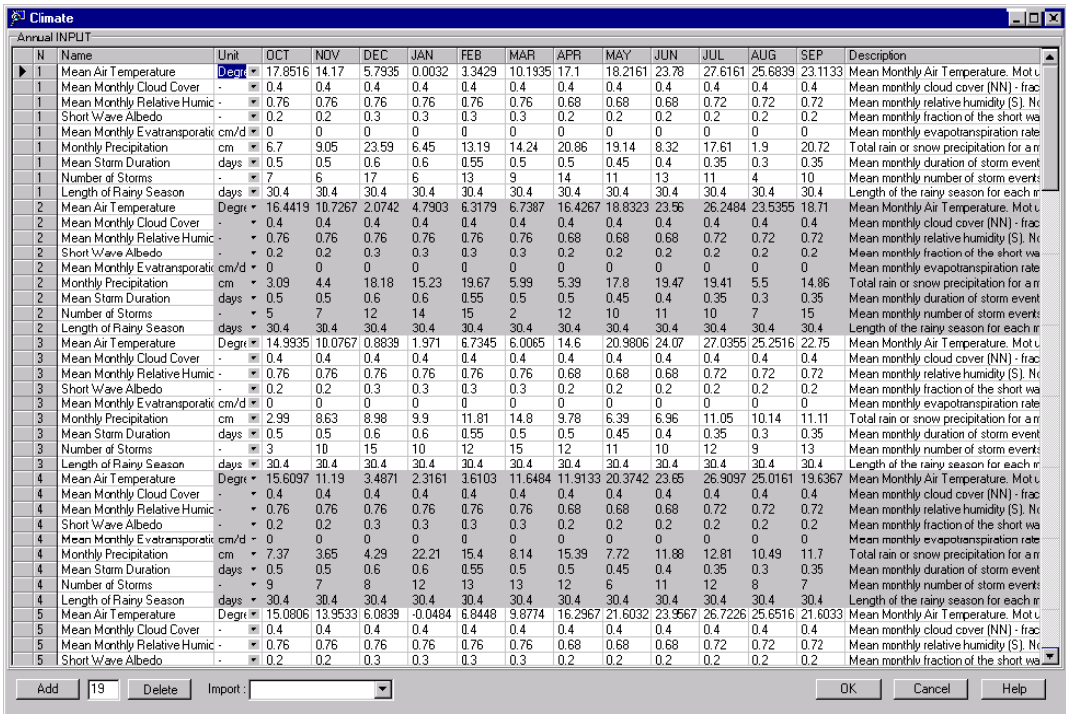
☞☞ **Climate** (from the Project Tree)

☞ the drop-down arrow in the **Import** field (bottom middle of window)

☞ **Use simulated values** (from the list that appears)

Maximize the dialog window.

Data for years 2 - 10 will appear in the table. Values of the generated monthly **Mean Air Temperature, Total Monthly Precipitation** and **Number of Storms** for the specified weather station (Nashville) will appear in the appropriate rows of the **Climate** Annual Input table, the rest of the parameters will be copied from the default annual set.



N	Name	Unit	OCT	NOV	DEC	JAN	FEB	MAR	APR	MAY	JUN	JUL	AUG	SEP	Description
1	Mean Air Temperature	Degr	17.8516	14.17	5.7935	0.0032	3.3429	10.1935	17.1	18.2161	23.78	27.6161	25.6839	23.1133	Mean Monthly Air Temperature. Mot
1	Mean Monthly Cloud Cover	-	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	Mean monthly cloud cover (NN) - frac
1	Mean Monthly Relative Humic	-	0.76	0.76	0.76	0.76	0.76	0.76	0.68	0.68	0.68	0.72	0.72	0.72	Mean monthly relative humidity (S). N
1	Short Wave Albedo	-	0.2	0.2	0.3	0.3	0.3	0.3	0.2	0.2	0.2	0.2	0.2	0.2	Mean monthly fraction of the short wa
1	Mean Monthly Evatransporatic	cm/d	0	0	0	0	0	0	0	0	0	0	0	0	Mean monthly evapotranspiration rate
1	Monthly Precipitation	cm	6.7	9.05	23.59	6.45	13.19	14.24	20.86	19.14	8.32	17.61	1.9	20.72	Total rain or snow precipitation for a m
1	Mean Storm Duration	days	0.5	0.5	0.6	0.6	0.55	0.5	0.5	0.45	0.4	0.35	0.3	0.35	Mean monthly duration of storm event
1	Number of Storms	-	7	6	17	6	13	9	14	11	13	11	4	10	Mean monthly number of storm event
1	Length of Rainy Season	days	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	Length of the rainy season for each m
2	Mean Air Temperature	Degr	16.4419	10.7267	2.0742	4.7903	6.3179	6.7387	16.4267	18.8323	23.56	26.2484	23.5355	18.71	Mean Monthly Air Temperature. Mot
2	Mean Monthly Cloud Cover	-	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	Mean monthly cloud cover (NN) - frac
2	Mean Monthly Relative Humic	-	0.76	0.76	0.76	0.76	0.76	0.76	0.68	0.68	0.68	0.72	0.72	0.72	Mean monthly relative humidity (S). N
2	Short Wave Albedo	-	0.2	0.2	0.3	0.3	0.3	0.3	0.2	0.2	0.2	0.2	0.2	0.2	Mean monthly fraction of the short wa
2	Mean Monthly Evatransporatic	cm/d	0	0	0	0	0	0	0	0	0	0	0	0	Mean monthly evapotranspiration rate
2	Monthly Precipitation	cm	3.09	4.4	18.18	15.23	19.67	5.99	5.39	17.8	19.47	19.41	5.5	14.86	Total rain or snow precipitation for a m
2	Mean Storm Duration	days	0.5	0.5	0.6	0.6	0.55	0.5	0.5	0.45	0.4	0.35	0.3	0.35	Mean monthly duration of storm event
2	Number of Storms	-	5	7	12	14	15	2	12	10	11	10	7	15	Mean monthly number of storm event
2	Length of Rainy Season	days	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	Length of the rainy season for each m
3	Mean Air Temperature	Degr	14.9935	10.0767	0.8839	1.971	6.7345	6.0065	14.6	20.9806	24.07	27.0355	25.2516	22.75	Mean Monthly Air Temperature. Mot
3	Mean Monthly Cloud Cover	-	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	Mean monthly cloud cover (NN) - frac
3	Mean Monthly Relative Humic	-	0.76	0.76	0.76	0.76	0.76	0.76	0.68	0.68	0.68	0.72	0.72	0.72	Mean monthly relative humidity (S). N
3	Short Wave Albedo	-	0.2	0.2	0.3	0.3	0.3	0.3	0.2	0.2	0.2	0.2	0.2	0.2	Mean monthly fraction of the short wa
3	Mean Monthly Evatransporatic	cm/d	0	0	0	0	0	0	0	0	0	0	0	0	Mean monthly evapotranspiration rate
3	Monthly Precipitation	cm	2.99	8.63	8.98	9.9	11.81	14.8	9.78	6.39	6.96	11.05	10.14	11.11	Total rain or snow precipitation for a m
3	Mean Storm Duration	days	0.5	0.5	0.6	0.6	0.55	0.5	0.5	0.45	0.4	0.35	0.3	0.35	Mean monthly duration of storm event
3	Number of Storms	-	3	10	15	10	12	15	12	11	10	12	9	13	Mean monthly number of storm event
3	Length of Rainy Season	days	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	Length of the rainy season for each m
4	Mean Air Temperature	Degr	15.6097	11.19	3.4871	2.3161	3.6103	11.6484	11.9133	20.3742	23.65	26.9097	25.0161	19.6367	Mean Monthly Air Temperature. Mot
4	Mean Monthly Cloud Cover	-	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	Mean monthly cloud cover (NN) - frac
4	Mean Monthly Relative Humic	-	0.76	0.76	0.76	0.76	0.76	0.76	0.68	0.68	0.68	0.72	0.72	0.72	Mean monthly relative humidity (S). N
4	Short Wave Albedo	-	0.2	0.2	0.3	0.3	0.3	0.3	0.2	0.2	0.2	0.2	0.2	0.2	Mean monthly fraction of the short wa
4	Mean Monthly Evatransporatic	cm/d	0	0	0	0	0	0	0	0	0	0	0	0	Mean monthly evapotranspiration rate
4	Monthly Precipitation	cm	7.37	3.65	4.29	22.21	15.4	8.14	15.39	7.72	11.88	12.81	10.49	11.7	Total rain or snow precipitation for a m
4	Mean Storm Duration	days	0.5	0.5	0.6	0.6	0.55	0.5	0.5	0.45	0.4	0.35	0.3	0.35	Mean monthly duration of storm event
4	Number of Storms	-	9	7	8	12	13	13	12	6	11	12	8	7	Mean monthly number of storm event
4	Length of Rainy Season	days	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	30.4	Length of the rainy season for each m
5	Mean Air Temperature	Degr	15.0806	13.9533	6.0839	-0.0484	6.8448	9.8774	16.2967	21.6032	23.9567	26.7226	25.6516	21.6033	Mean Monthly Air Temperature. Mot
5	Mean Monthly Cloud Cover	-	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	0.4	Mean monthly cloud cover (NN) - frac
5	Mean Monthly Relative Humic	-	0.76	0.76	0.76	0.76	0.76	0.76	0.68	0.68	0.68	0.72	0.72	0.72	Mean monthly relative humidity (S). N
5	Short Wave Albedo	-	0.2	0.2	0.3	0.3	0.3	0.3	0.2	0.2	0.2	0.2	0.2	0.2	Mean monthly fraction of the short wa

☞ **OK** (from the bottom of **Climate** window)

Specifying the Contaminant

To specify contaminants in SESOIL, you must select the proper contaminant, initial concentrations, and a contaminant application schedule.

To select the contaminant, use the WHI UnSat Suite database that contains parameters for 23 commonly used industrial and agricultural contaminants. As well, you may add a new contaminant to the database. However, in this exercise the requested chemical (benzene) is already a default contaminant.

To view and edit contaminant properties:

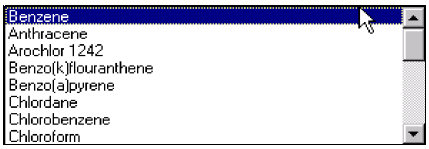
☞☞ **benzene** (from the Project Tree)

☞ **Chemical Parameters** tab

In the dialog window that appears, you may edit contaminant properties or select another contaminant from the database.

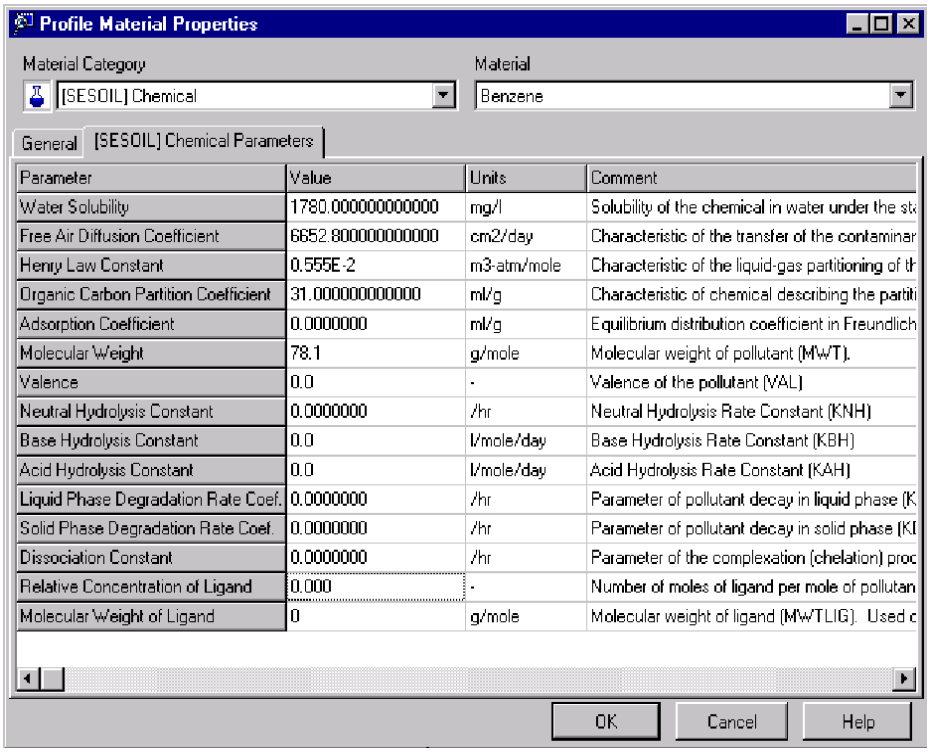
☞ the drop-down arrow of the **Material** list

A list of available chemicals will appear:



☞ the drop-down arrow of the **Material** list field

Your dialog window should appear as seen below:



☞ **Cancel** (to close the **Chemical Parameters** tab)

Contaminant Application Schedule

In this exercise, you are going to simulate a gasoline leak from the underground storage tank. As stated previously, the average depth at which leakage occurred was 10 feet. As well, the duration of leakage is 2 years.

The weighted concentration of benzene in the 20-feet upper soil layer was 200 mg/kg. To calculate the amount of the benzene that has been lost, we have to assess the amount of contaminant in the vertical prism stretching from the bottom to the top of the contaminated layer with a unit area 1 cm^2.

The total volume of soil in this prism is: 600.96 cm * 1.0 cm^2 = 600.96 cm^3.

The total mass of soil in this prism is: 600.96 cm^3 * 1.38 g/cm^3 = 829 g.

The total mass of pollutant in the prism: 200mg/1000 g * 829 g = 165 mg = 165,000 µg.

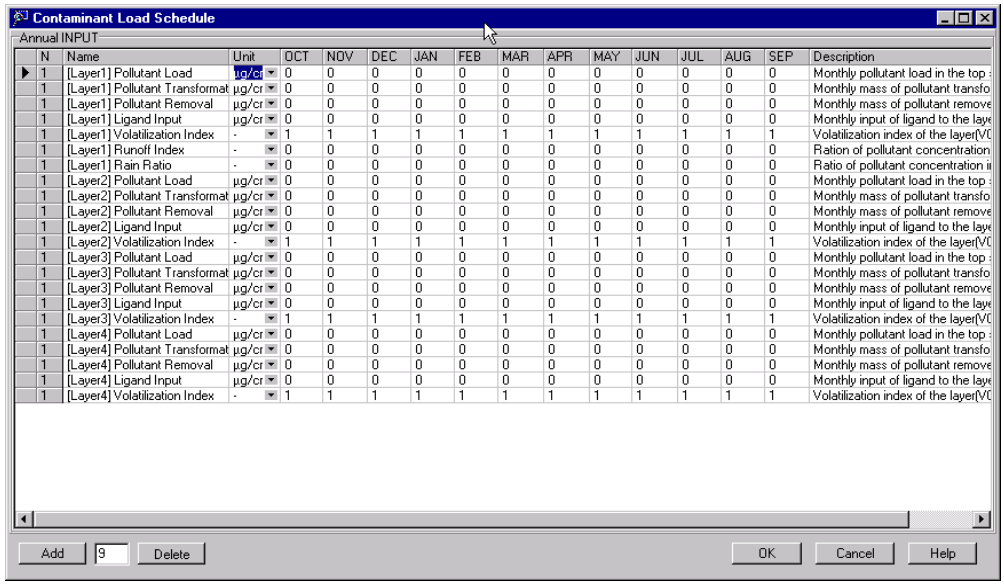
This amount has to be introduced evenly over a 24-month period with intensity 6875 µg/month.

To set the Contaminant Load Schedule,

☞☞ **Contaminant Load Schedule** (in the Project Tree View)

The following dialog window will appear:

In this table, you can specify contaminant sources/sinks for any layer for each month of the simulation. By default, the input template for Year 1 is opened.



The following contaminant sources/sinks may be specified:

Pollutant Load	*Monthly pollutant load in the top sub-layer.*
Pollutant Transformation	*Monthly mass of pollutant transformed in the layer (beside decay).*
Pollutant Removal	*Monthly mass of pollutant removed from the layer.*
Ligand Input	*Monthly input of ligand to the layer.*
Volatilization Index	*Volatilization index of the layer. VOLF=0.0 means no volatilization/ diffusion from the layer, while VOLF=1.0 means full volatilization/diffusion from the layer.*
Runoff Index	*Ratio of pollutant concentration in surface runoff to concentration in top sub-layer.*
Rain Ratio	*Ratio of pollutant concentration in rain to pollutant solubility in water.*

In this exercise, we will specify only values for **Pollutant Load.** Pollutant load has to be applied to Layer 2 which has an upper boundary located at depth - 10 feet.

Because the layer top and depth at which the contaminant was released coincide in this exercise, there is no need to adjust the layer thickness. However, if the layer thickness and other layer parameters have to be edited, this can be easily done in the **Layer Parameters** group.

To set the pollutant load units in accordance to the exercise settings,

☞ the field to the right of the **Unit** field (beside **Layer 2 Pollutant Load**)

Type **6875** (to set the pollutant load for the first month of **Year 1 - October**)

Type the identical number in the rest of the row. By doing so, you will input pollutant load for the first year of simulation.

To input the same schedule for the Year 2, you should copy the annual data set for Year 1.

☞ in the text-window beside the **Add** button (bottom left of window)

Type **1** (the number of annual data set copies)

☞ **Add**

The requested number of annual set copies will appear appended to the bottom of the table.

In SESOIL, if the number of years specified for annual input of climate, pollutant load or washload parameters is less than the length of simulation, the data set for the last year will be repeated until the end of simulation.

To stop simulating the pollutant load after Year 2 ends, you have to add another year with zero pollutant load.

☞ in the leftmost field beside **Layer 2 Pollutant Load** for Year 2

The entire annual set will become highlighted.

☞ in the text-window beside **Add**

Type **1** (the number of copies)

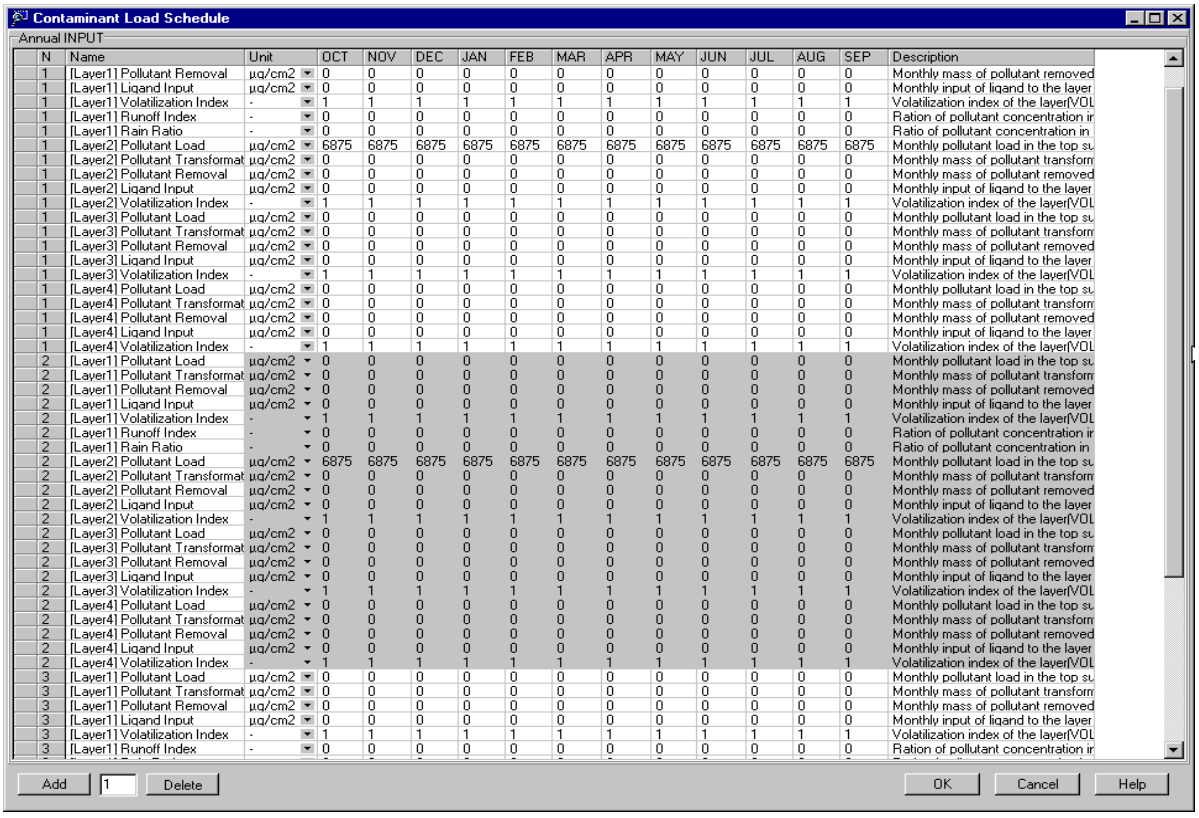
☞ **Add**

The copy of annual set for Year 2 will appear appended to the bottom of the table.

☞ in the field to the right of **Unit** in the **Layer 2 Pollutant Load**

Type **0** (to set the pollutant load to zero for the first month of **Year 3 - October**)

Type the same number in the rest of the row, to produce a figure similar to the one below:



N	Name	Unit	OCT	NOV	DEC	JAN	FEB	MAR	APR	MAY	JUN	JUL	AUG	SEP	Description
1	[Layer1] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
1	[Layer1] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
1	[Layer1] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
1	[Layer1] Runoff Index	-	0	0	0	0	0	0	0	0	0	0	0	0	Ration of pollutant concentration ir
1	[Layer1] Rain Ratio	-	0	0	0	0	0	0	0	0	0	0	0	0	Ratio of pollutant concentration in
1	[Layer2] Pollutant Load	µg/cm2	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	Monthly pollutant load in the top su
1	[Layer2] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
1	[Layer2] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
1	[Layer2] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
1	[Layer2] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
1	[Layer3] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
1	[Layer3] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
1	[Layer3] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
1	[Layer3] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
1	[Layer3] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
1	[Layer4] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
1	[Layer4] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
1	[Layer4] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
1	[Layer4] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
1	[Layer4] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
2	[Layer1] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
2	[Layer1] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
2	[Layer1] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
2	[Layer1] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
2	[Layer1] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
2	[Layer1] Runoff Index	-	0	0	0	0	0	0	0	0	0	0	0	0	Ration of pollutant concentration ir
2	[Layer1] Rain Ratio	-	0	0	0	0	0	0	0	0	0	0	0	0	Ratio of pollutant concentration in
2	[Layer2] Pollutant Load	µg/cm2	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	6875	Monthly pollutant load in the top su
2	[Layer2] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
2	[Layer2] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
2	[Layer2] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
2	[Layer2] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
2	[Layer3] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
2	[Layer3] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
2	[Layer3] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
2	[Layer3] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
2	[Layer3] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
2	[Layer4] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
2	[Layer4] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
2	[Layer4] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
2	[Layer4] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
2	[Layer4] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
3	[Layer1] Pollutant Load	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top su
3	[Layer1] Pollutant Transformat	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transform
3	[Layer1] Pollutant Removal	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed
3	[Layer1] Ligand Input	µg/cm2	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the layer
3	[Layer1] Volatilization Index	-	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer[VOL
3	[Layer1] Runoff Index	-	0	0	0	0	0	0	0	0	0	0	0	0	Ration of pollutant concentration ir

☞ **OK** (to save data and exit)

Specifying Groundwater Settings

The SESOIL model allows you to set parameters of the aquifer and assess the potential concentration of the contaminant concentration in the groundwater under the contaminated site. A modified Summer's model equation is used for computing the contaminant concentration in the saturated zone. This balanced groundwater mix model does not account for the dynamics of dissolution or stratigraphy of concentration within the aquifer. However, it may be useful for quick estimates of the contaminant concentration in the groundwater.

☞☞ **Groundwater Settings** (in the Project Tree)

The **Edit Parameters** dialog window will appear,

☞ **Units** field (beside **Saturated Hydraulic Conductivity**)

☞ **cm/sec** (from the drop-down list)

☞ **Value** field (beside **Saturated Hydraulic Conductivity**)

Type **0.01**

☞ **Value** field (beside **Thickness of Saturated Zone**)

Type **10.0**

Next, you have to specify the width of the contaminated zone. You may estimate this parameter based on the assumption that the contaminated spot has a square shape. Taking into account a contaminated area of 400 m^2, the width of the zone equals 20 m.

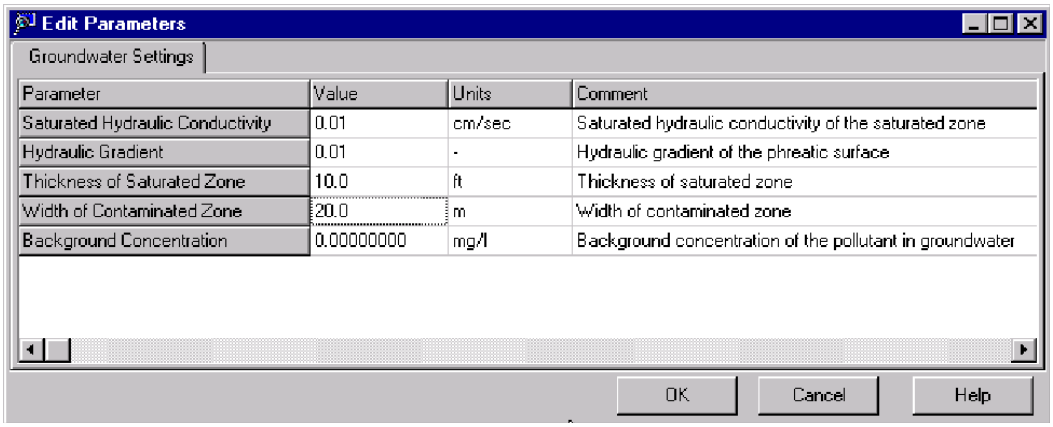
☞ **Units** field (beside **Width of Contaminated Zone**)

☞ **m** (from the drop-down list)

☞ **Value** field (beside the **Width of Contaminated Zone**)

Type **20.0**

As a result of editing, the **Groundwater Settings** parameter window will appear as seen below:



☞ **OK** (to save data and exit)

You have finished inputting data for this exercise. In addition to already specified parameters, with the WHI UnSat Suite interface you can easily specify contaminant washload settings, initial concentrations and layer parameters. However, these settings are either not used (washload process) or used with their default values (initial concentrations and layer parameters).

This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **SESOIL tutorial 1**, which was specifically developed for you and contains all changes you have done so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

☞ ☞ project **SESOIL tutorial 1.**

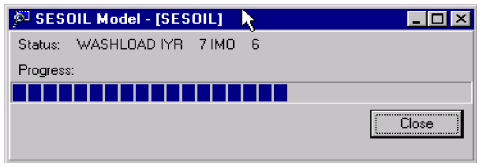
Module III: Running the Model, Viewing Output, and Reporting

Running the SESOIL Model

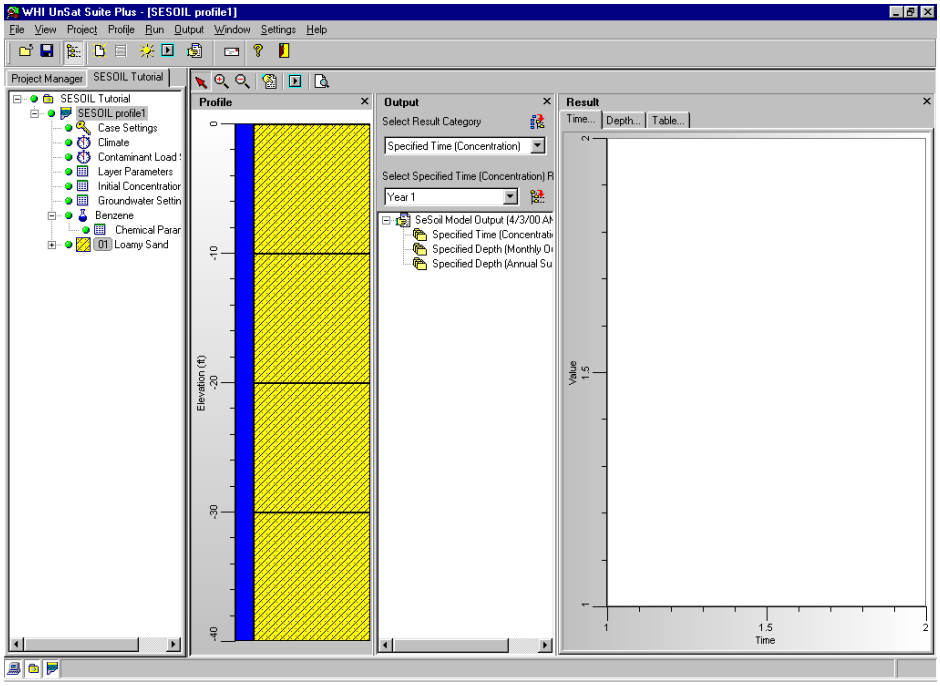
To run the model,

☞  (the **Run SESOIL** icon from the top menu bar)

A progress bar will appear as shown below:



When completed, the **Output** and **Result** windows will be added to your display automatically.



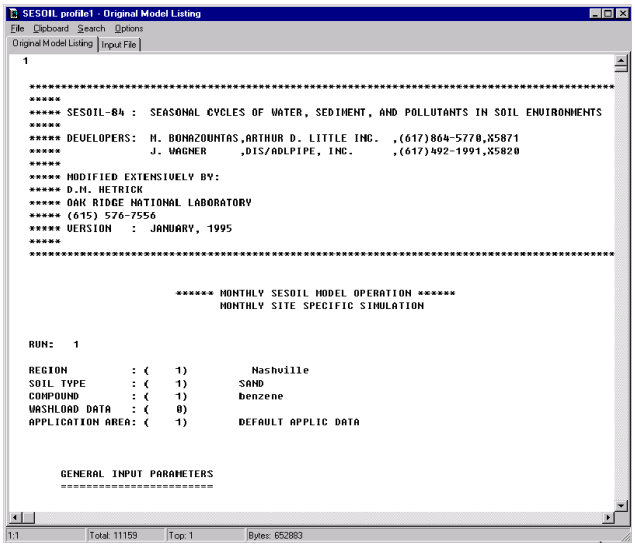
Viewing Original DOS SESOIL Output

UnSat Suite allows you to view and print the original DOS **SESOIL** output.

☞ **Output** (from the top menu bar)

☞ **View Original Listing** (from the drop-down menu that appears)

The **Original Model Listing** dialog window will appear:



You may scroll and view the original listing, find specific expressions, and print your results.

Click the [**X**] to close the window.

Viewing Output Graphs

To enlarge the graph viewing area, you may remove the **Project Tree** and **Profile view** from your display:

☞  (Project Tree icon from the top menu bar)

☞ '**X**' in the **Profile** View (to close the window)

The **Output** View window contains two drop-down windows:

1. **Select Result Category** (used to select general result category)
2. **Select Specified (Category Name) Result At....** (used to select specific result category)

To select the general output category, click the arrow in the **Select Result Category** drop-down list window. The following list will appear:



If you select the category you wish to view from the list, the first available result group for this category will appear in the window **Select 'Name of Category' Result at...** below.

Click the arrow to close the drop-down list.

Now, you will view various categories to present simulation results from different points of view.

Viewing Specified Depth (Annual Summary) Output

This output category allows you to view annual balances of water and contaminant, as well as breakthrough curves at fixed depths (in specific sub-layer) within the simulation period.

As this output category allows you to make a general overview of simulation results, we recommend you always start your output examination with this option.

You may view the following sub-categories and variables as annual totals and averages:

Hydrologic Cycle Components *Components of the profile hydrologic balance*

Washload Cycle Components *Components of the profile washload cycle (if simulated)*

Pollutant Mass Input *Contaminant input into the profile presented for each layer*

Pollutant Mass Distribution *Contaminant balance presented for each layer and sub-layer*

Average Pollutant Concentration *Contaminant concentration in liquid, solid and gas phases of the soil presented for each layer and sub-layer*

Maximum Pollutant Depth *Contaminant downward propagation within the profile*

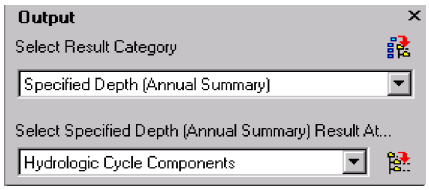
Average Concentration in Groundwater *Contaminant concentration in groundwater*

You will examine the general characteristics of contaminant flow and transport, but first we must begin by examining the hydrologic balance of the profile.

Hydrologic Cycle Components

☞ **Specified Depth (Annual Summary)** (from the **Select Result Category** list)

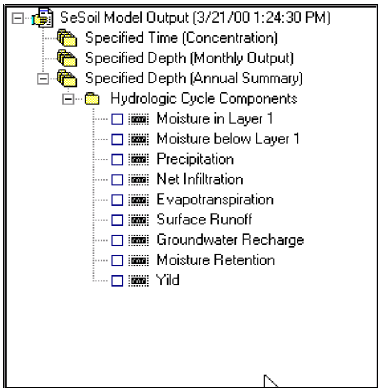
The following window will appear, displaying the first sub-category **Hydrologic Cycle Components**:



To view results for this sub-category,

☞  (icon to the right of **Hydrologic Cycle Components**)

A list of available hydrologic variables will appear:



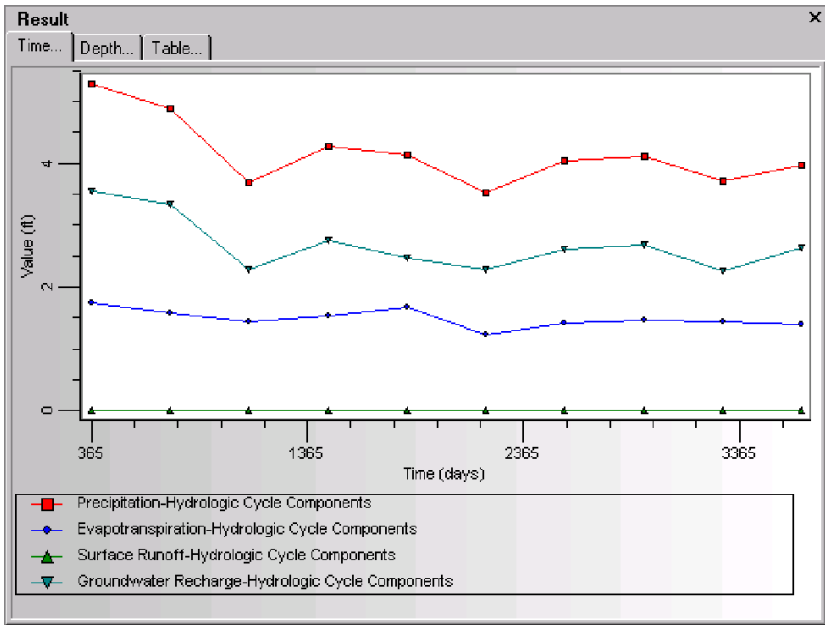
☞ the check-boxes beside the following variables: **Precipitation**

Evapotranspiration

Surface Runoff

Groundwater Recharge

As you select check-boxes, graphs for specific variables will appear in the Result View window. Once completed, your display will appear as seen below:



The graph illustrates that precipitation ranges from 3.5 to 5.5 feet per year, and that water entering the unsaturated profile discharges mainly to the groundwater surface. The annual amount of groundwater recharge varies between 2.25 and 3.5 feet. In addition, the second process causing water discharge from the unsaturated profile is evapotranspiration, which accounts for 1.2 - 1.7 feet of water loss annually. Finally, surface runoff produces no water loss, as this feature is simulated through the washload cycle, which was not used in this exercise.

☞ '-' sign to the left of **Hydrologic Cycle Components** to close the **Output Tree**.

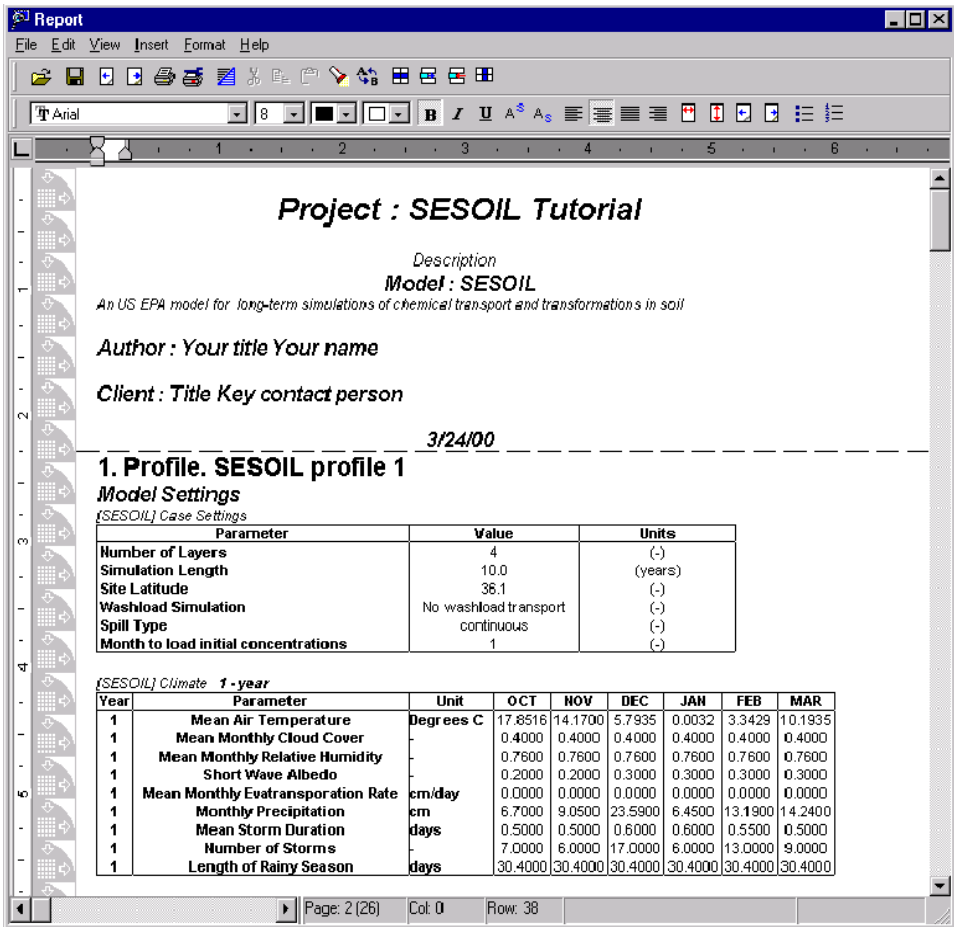
At this time, you may wish to create a project report and add the above graph to it.

Creating a Report

To present SESOIL simulation results to your clients, you may use the UnSat Suite Report Generator.

☞  (the **Report Generator** icon from the top menu bar)

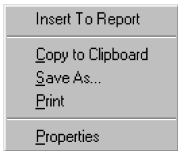
The report will appear in a separate window:



In this window, you may edit the report, input your own text, and add graphics produced by WHI UnSat Suite. The graphs and tables will be placed at the insertion point.

To add a graph to the report,

1. Scroll the bar in the **Report** window to the bottom and place the cursor below the last table in the report
2. Return to the main window of WHI UnSat Suite and ***right-click*** on the graph or table in the Result View
3. The following menu will appear:



☞ **Insert To Report**

The graph will appear in the report. To view the graph of desired size, click in the **Report** window and stretch the graph until it reaches the proper size.

Now we have the report started. Further analysis will produce additional graphs and tables that can be added at a later date.

Next, you will view the constituents of the contaminant mass balance at different depths. This information is presented in the sub-category **Mass Distribution.**

Contaminant Mass Distribution

☞ the arrow in the **Select** Specified Depth (Annual Summary) Result at... window

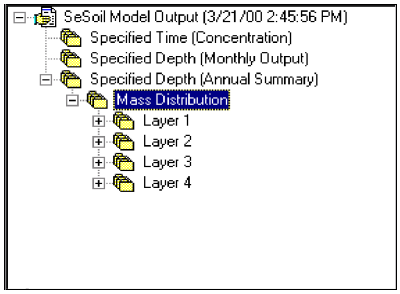
The list of available sub-categories will appear:



☞ **Mass Distribution**

☞  (the icon to the right of **Mass distribution**)

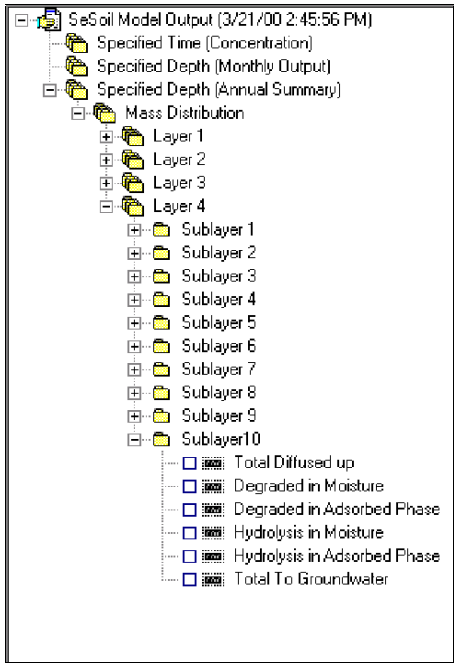
The list of four profile layers will appear as seen below:



First, you will examine the contaminant distribution close to the groundwater level (Layer 4, Sub-layer 10).

☞ '+' sign to the left of **Layer 4** (Ten sub-layers will appear in the tree)

☞ '+' sign to the left of **Sub-layer 10** (available variables will appear in the tree)



☞ the check-boxes beside **Total To Groundwater**

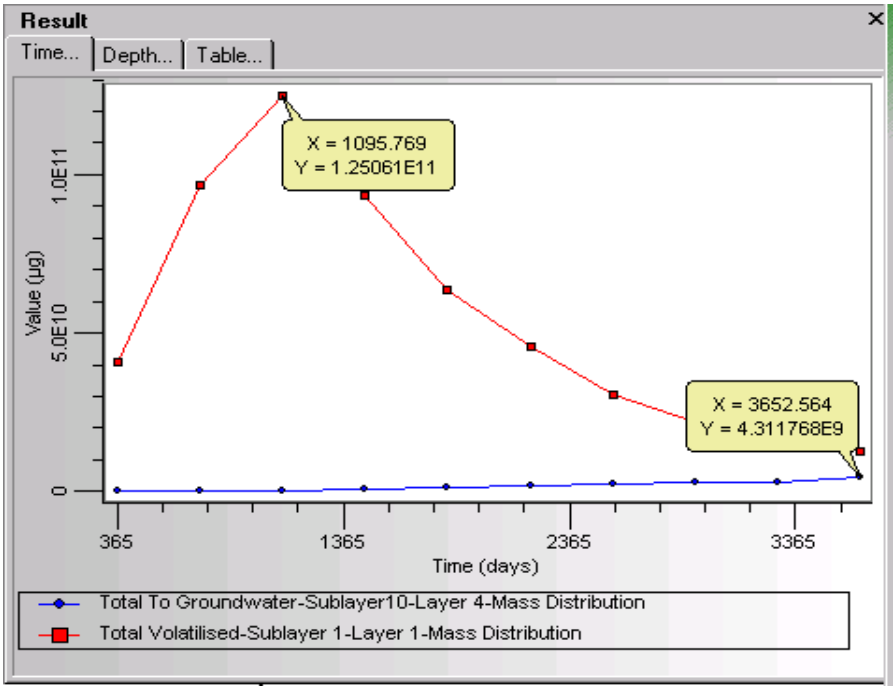
☞ '+' sign to the left of **Layer 1** (Four sub-layers will appear in the tree)

☞ '+' sign to the left of **Sub-layer 1** (available variables will appear in the tree)

☞ the check-boxes beside **Total Volatilised**

A graph containing two data sets will appear in the **Result** view window. Move your mouse over the graph, place it on the peak values for each data set (a bubble sign will appear) and click. Values of the variable and the argument will appear in the bubble. This feature of the WHI UnSat Suite allows you to highlight the most important points in your graphs.

Your graph should appear similar to the figure below:



The graph shows that emission to the atmosphere (volatilization) is close to 4.1E+10 µg during Year 1, and reaches a maximum of 1.25E +11 µg during Year 3. Emission to groundwater grows gradually and reaches a maximum of 4.31E+9 µg during Year 10.

Note: Numbers presented here relate to the entire contaminated site with the area of 400 m^2 where the total amount of leaked benzene was 6.6 E+11µg.

Right-click on the graph in the **Result** View window,

☞ **Insert To Report** (from the window that appears)

The graph will appear in the report, which you may change in size to fit your needs.

☞ **Output** (from the top menu bar)

☞ **Clear Display Results** (the Result View window will clear)

Next, you will view the contaminant distribution between three soil phases at different depths. This information is presented in the sub-category, **Pollutant Concentration.**

Pollutant Concentration

☞ the arrow in the **Select Specified Depth (Annual Summary) Result at...** window

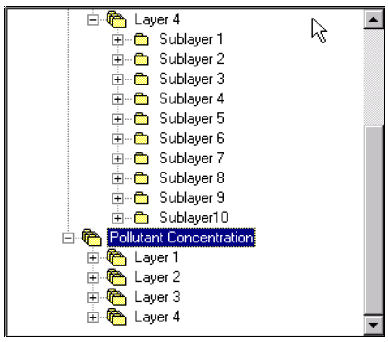
The list of available sub-categories will appear:



☞ **Pollutant Concentration**

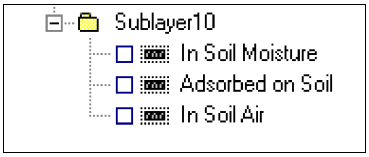
☞  (the icon to the right of the **Pollutant Concentration**)

A list of four profile layers will appear in the Output Tree. You may need to use the side slider in the Output Tree to display this sub-category.



☞ '+' sign to the left of **Layer 4** (10 sub-layers will appear in the tree)

☞ '+' sign to the left of **Sub-layer 10** (available variables will appear in the tree)



☞ the check box beside **In Soil Moisture**

The graph for concentration at Sub-layer 10, Layer 4, will appear in the **Output** View window. This is the point closest to the groundwater surface.

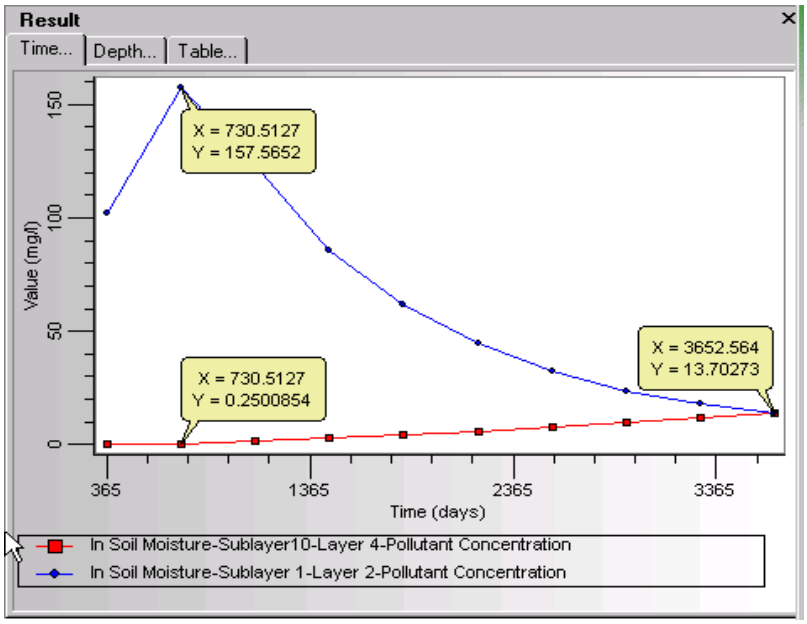
Next, you will view the graph for the depth at which the leak occurred. Using the side slider (if necessary), move the Output Tree to the node labeled **Layer 2**.

☞ '+' sign to the left of **Layer 2** (10 sub-layers will appear in the tree)

☞ '+' sign to the left of **Sub-layer 1**

☞ the check box beside **In Soil Moisture**

The graph for concentration at Sub-layer 1, Layer 2, will appear in the **Output** View window. This is the point closest to the depth of 10 feet at which the benzene leaked from the storage tanks. Move your mouse over the graph, place it on the peak values for each data set and Year 2 for the lower curve (a bubble sign will appear) and click:



The graph shows that the average annual benzene concentration at the release depth reached a maximum of 157 mg/l, at Year 2. Subsequently, the concentration level quickly dropped after the leak was liquidated at the release depth. The concentration near the groundwater surface was 0 at Year 1, grew to 0.25 mg/l for Year 2 reached a peak of 13.7 mg/l during Year 10.

Right-click on the graph in the **Result** View window,

☞ **Insert To Report**

The graph will appear in the report, which you may alter in size if necessary.

☞ **Output** (from the top menu bar)

☞ **Clear Display Results** (the Result View window will clear)

Next, you will view the average benzene concentration in the groundwater.

☞ the arrow in the **Select** Specified Depth (Annual Summary) Result at...

The list of available sub-categories will appear:

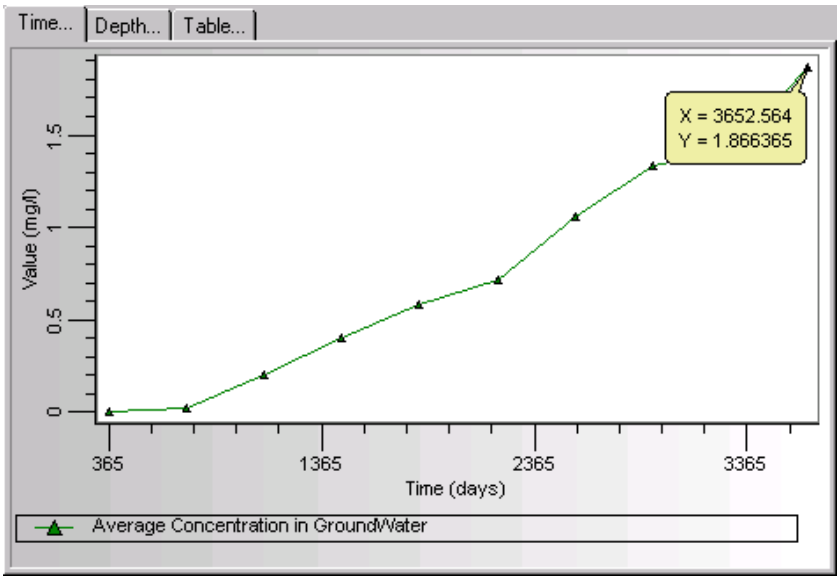
☞ **Average Concentration in Groundwater**

☞  (icon to the right of Average Concentration in groundwater)

The node for **Average Concentration in Groundwater** will appear in the bottom of the **Output Tree.**

☞ the check-box beside the node (you may need to scroll the slider down to reach it)

The following graph will appear:



The graph shows the concentration of benzene in the groundwater reached a peak of 1.87 mg/l during Year 10.

By analyzing the results, one may notice that the maximum benzene concentrations in both soil water and groundwater exceed the EPA drinking water limit of 0.005 mg/l.

The above graphs present annual concentration averages. To get a detailed picture of the process, monthly values of concentration have to be examined.

Right-click on the graph in the **Result** View window,

☞ **Insert To Report**

The graph will appear in the report, which you may alter in size if necessary.

☞ **Output** (from the top menu bar)

☞ **Clear Display Results** (the Result View window will clear)

☞ '-' sign to the left of **Specified Depth (Annual Summary)** in the **Output Tree** (to close the Output Tree)

Viewing Specified Depth (Monthly Output)

This output category allows you to view water and contaminant balances, as well as breakthrough curves at fixed depth (in specific sub-layers) within annual cycles of simulation.

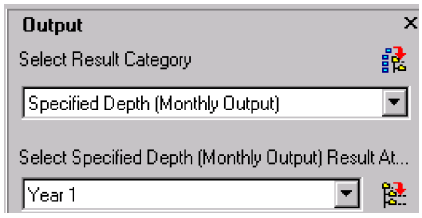
All the output methods described above for the **Specified Depth (Annual Summary)** are applicable to the **Specified Depth (Monthly Output)** category.

At the first level of output you have to select the **year** to view. within the year output branch, the structure of the output for this general category is much the same as those for **Specified Depth (Annual Summary).**

In this exercise, we will examine the monthly output for Year 3 when the highest annual average benzene concentrations both in the soil water and in the groundwater were reached.

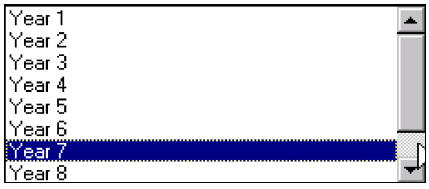
☞ **Specified Depth (Monthly Output)** (in the **Select Result Category** drop-down list)

The dialog window will display the first available annual set of results:



☞ the arrow in the **Select** 'Specified Depth (Monthly Output) Result at... list

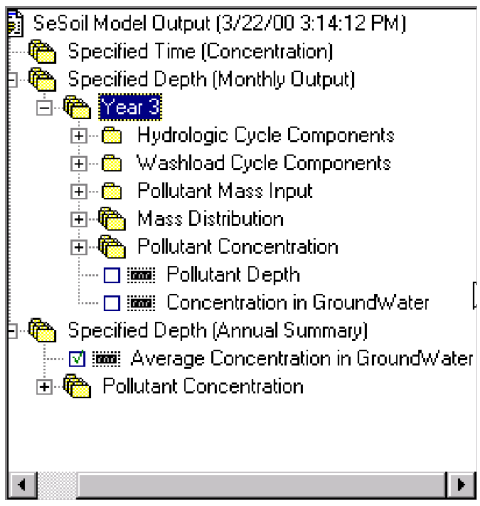
The list of output times will appear:



☞ **Year 3** from the drop-down list.

☞  (the icon to the right of **Year 3**)

The following structure will appear in the Output Tree:



☞ '+' sign to the left of **Pollutant Concentration** (below **Year 3**)

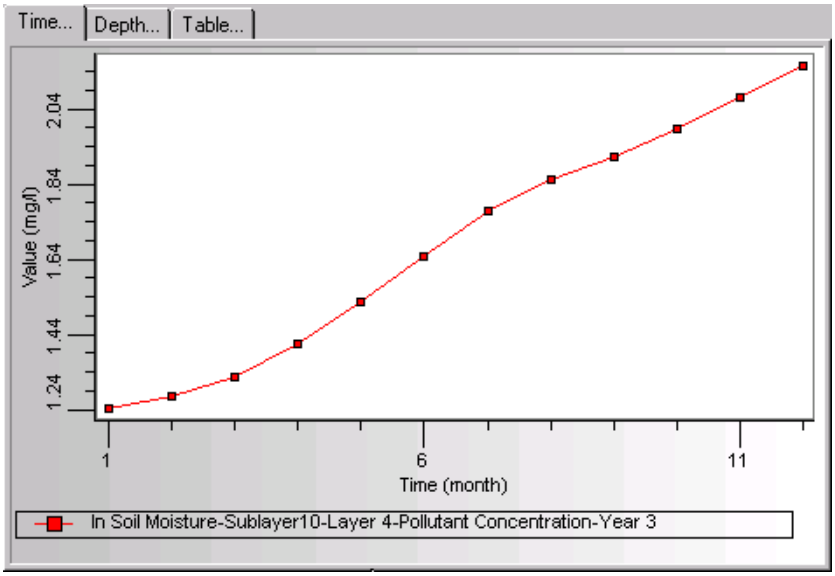
A list of four profile layers will appear in the Output Tree.

☞ '+' sign to the left of **Layer 4** (10 sub-layers will appear in the tree)

☞ '+' sign to the left of **Sub-layer 10** (available variables will appear in the tree)

☞ the check box beside **In Soil Moisture**

The graph for concentration at Sub-layer 10, Layer 4, will appear in the **Output** View window.



This is the point closest to the groundwater surface. The graph indicates that during Year 3, the concentration monotonously increased from 1.24 to 2.16 mg/l.

Right-click on the graph in the **Result** View window,

☞ **Insert To Report**

The graph will appear in the report, which you may alter in size if necessary.

☞ **Output** (from the top menu bar)

☞ **Clear Display Results** (the Result View window will clear)

☞ '-' sign to the left of **Specified Depth (Monthly Output)** in the **Output Tree** (to close the Output Tree)

Finally, you will explore the distribution of contaminant within the unsaturated profile.

Viewing Specified Time (Concentration) Output

This output category allows you to view vertical profiles of contaminant concentration in liquid, gas and solid phases at any specific time during the simulation period. You will examine the characteristics of the vertical contaminant flow and transport during the peak load at the groundwater surface.

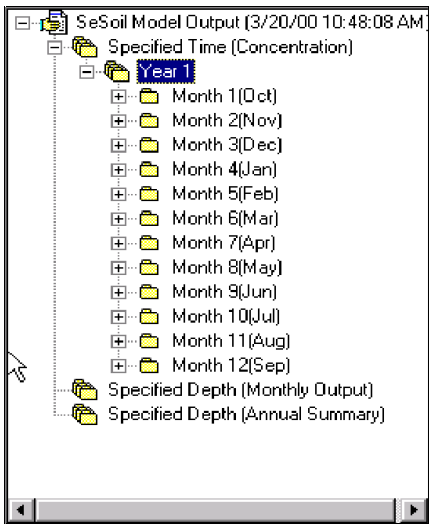
☞ **Specified Time (Concentration)** in the **Select Result Category** drop-down list

☞ **Year 1** (from the drop-down list)

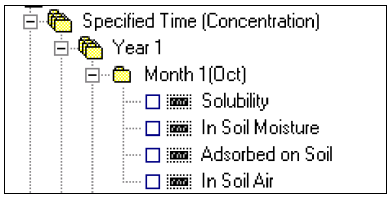
To view results for this year,

☞  (the icon to the right of **Year 1**)

A list of months for Year 1 will open in the Result Tree:



☞ **' +'** sign beside **Month 1 (Oct)**



Now, you may view the vertical profile for four variables for the specified time:

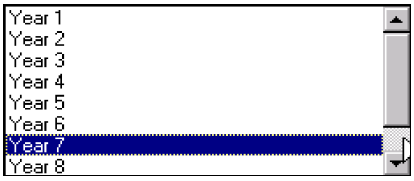
1. **Solubility** - actual concentration as percent of the maximum solubility of the chemical
2. **In Soil Moisture** - contaminant concentration in liquid phase
3. **Adsorbed on Soil** - contaminant concentration in the soil solid phase (adsorbed)
4. **In Soil Air** - contaminant concentration in gas phase.

☞ the check box beside **In Soil Moisture**

The graph of the variable will appear in the Result View window.

☞ the arrow in the **Select 'Specified Time (Concentration)' Result at..**. list

The list of output times will appear:



☞ **Year 2** (from the drop-down list)

☞  (the icon to the right of **Year 2**)

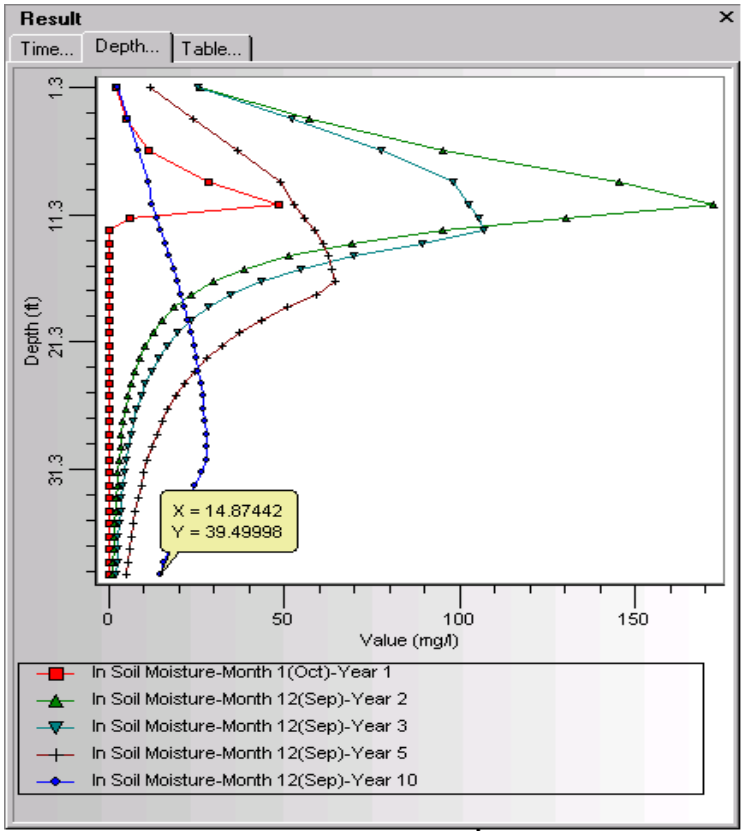
☞ '+' sign beside **Month 12 (Sept)** (you may need to scroll the list of months down)

☞ the check box beside **In Soil Moisture**

The graph for contaminant concentration for September of Year **2** will appear in the Result View window.

REPEAT the above procedure to add **In Soil Moisture** graphs for [Month **12,** Year **3**], [Month **12,** Year **5**]and [Month **12,** Year **10**] to the Result View window.

Once completed, the following figure will appear:



The graph shows that the benzene concentration at the leakage depth grew to 48.5 mg/l during the first month of the simulation and reaches its maximum of 172.4 mg/l at the end of Year 2.

During the next 8 years, the contaminant will move downwards to the groundwater surface. The concentration will decrease at the depth of contaminant release and increase at the groundwater surface. Its maximum 14.9 mg/l concentration at the groundwater surface will be reached at Month 12 Year 10.

Right-click on the graph in the **Result** View window,

☞ **Insert To Report**

The graph will appear in the report, which you may alter in size as required.

Using the same tools as when you explored the contaminant distribution in the liquid phase, you may explore distribution in gas or adsorbed phases of soil. We encourage you to do it on your own.

At this point, you have finished exploring the main output features of the WHI UnSat Suite. Finally, you will edit and print your project report.

Printing Your Report

Write your comments in the report as you would with any word processor. You may insert a header and footer in your report, and apply different fonts and styles while working in the **Report** window. To utilize these and other options, make corresponding selections from the main menu. Once you are satisfied with appearance, you may print the report or save it.

To print your report,

☞ **File** (from the top menu bar)

☞ **Print** (from the drop-down menu that appears)

This concludes the SESOIL exercise.

We hope these instructions have provided you with a good understanding of the capabilities and tools offered by the WHI UnSat Suite. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities that were not covered by this lab exercise.

WHI UnSat Suite Tutorial: VS2DT

Modeling Unsaturated Flow and Contaminant Transport in the Vadose Zone

WHI UnSat Suite for Windows 95/98/NT/2000

The WHI UnSat Suite combines four popular unsaturated zone models in a powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. These models are:

SESOIL - a popular US EPA model which is capable of simultaneously modeling water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess contaminant fate and transport for regulatory requirements.

PESTAN (PESTicide **AN**alitycal**)**- a model for assessments of groundwater contamination by pesticides used in agricultural practice.

VS2DT (Variably **S**aturated **2D** flow and **T**ransport**)** - a finite difference numerical model for determining the fate of agricultural chemicals, landfill leachate, and accidental chemical spills in the unsaturated zone.

VLEACH (**V**adose Zone **LEACH**ing model)- a model for evaluating the fate of volatile organic contaminants in the unsaturated zone.

All these popular models are seamlessly integrated with the WHI UnSat Suite and each one is compiled and optimized to run as a 32-Bit, native Windows application.

The WHI UnSat Suite shares the majority of its features with the other WHI software product Visual HELP which, besides its direct function, landfill designing, can be used as an efficient tool for assessment of soil surface water balance. The two products are connected functionally and logically and in many cases the joint use of two products in consulting projects is very beneficial. Both products are joined together in the software package called WHI UnSat Suite Plus. To learn more about Visual HELP, review the demo exercise for this product.

Learning Objectives

In this demonstration exercise, you will:

- create an UnSat Suite project
- Use VS2DT as a tool for simulating the transport of lead in the unsaturated zone;
- Prepare a project report

The instructions for this exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

Background Information for VS2DT Exercise

Some innovative companies have developed processes for recycling powerplant bottom ash, the by-product of coal combustion. The processed ash product is used as construction material to underlay asphalt for roads and parking lots. Unfortunately, some ash contains lead and cadmium which can potentially leach from the ash and contaminate groundwater.

In this exercise you will use UnSat Suite to simulate the transport and fate of lead leaching from a layer of ash to the groundwater. According to the WHI Unsaturated Zone modeling technology, the HELP and VS2DT models are used in conjunction to assess the concentration of lead in soil water reaching the groundwater at the site. HELP is used to provide accurate values of water percolating through the ash layer. VS2DT is used to simulate the unsaturated flow and transport of lead in the major part of the vadose zone below the bottom of the ash layer. The Visual HELP simulation showed that, being equal to 14 in/year in natural conditions, the average infiltration rate into the soil profile becomes equal to 4 in/year after the ash and asphalt layers are placed. The concentration of lead in solution infiltrating into the soil profile is taken from lysimeter tests.

The following are parameters of the profile simulated with the VS2DT model:

The profile consists of two layers, an upper layer which has a thickness of 20 feet and is composed of silt. A lower layer which has a thickness of 10 feet is composed of sandy clay. The parameters of the profile materials are considered equal to the default for VS2DT material database except for hydraulic conductivity of sandy clay which is equal to 5E-5 cm/sec. For simulation of lead adsorption a conservative value of 7 ml/g is used for the silt sorption coefficient (Kd) and a value of 12 ml/g for the sandy clay Kd. The depth to groundwater is 25 feet.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type-	Type in the given word or value
select-	Click the left mouse button where indicated
α -	Press the **<Tab>** key
↵ -	Press the **<Enter>** key
☞ -	Click the left mouse button where indicated
☞ ☞ -	Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

NOTE: All theoretical parts of the demo tutorial are italicized.

Starting WHI UnSat Suite

To start the WHI UnSat Suite, you must have it installed on your hard disk. Once installed, click Windows **Start** button, choose **Programs**, navigate to the folder that contains WHI UnSat Suite, and click **UnSat Suite**.

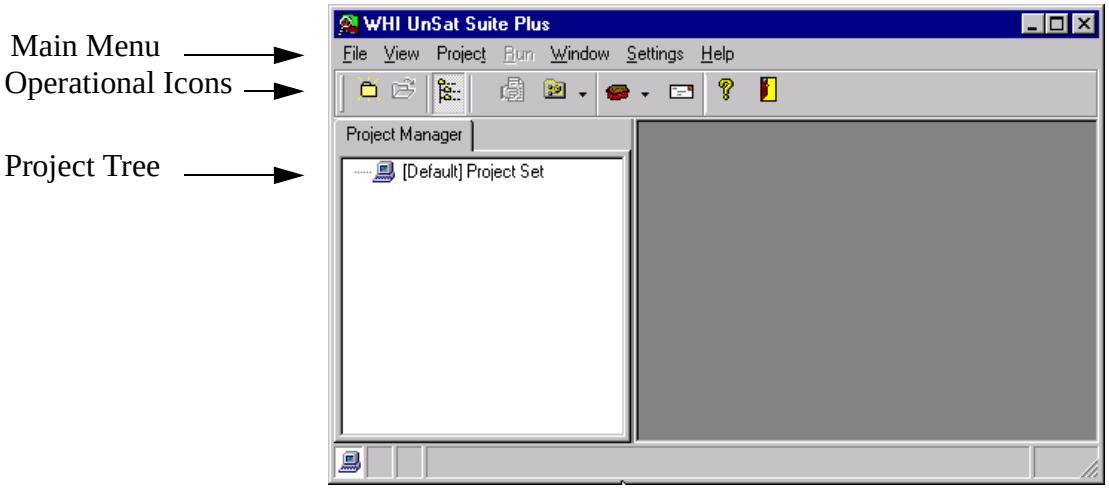
 Copyright 2000 Waterloo Hydrogeologic, Inc.

The installation program will also place an icon  on your desktop, which you can also double click to start the WHI UnSat Suite exercise.

An introductory screen will be displayed when you open WHI UnSat Suite



After the program is loaded, the main WHI UnSat Suite window will appear on your display:



The WHI UnSat Suite interface has been designed to help you navigate through the program with ease. The main menu is located at the top with operational icons located directly below the main menu items to allow quick access to many options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

Module I: Creating a New Project

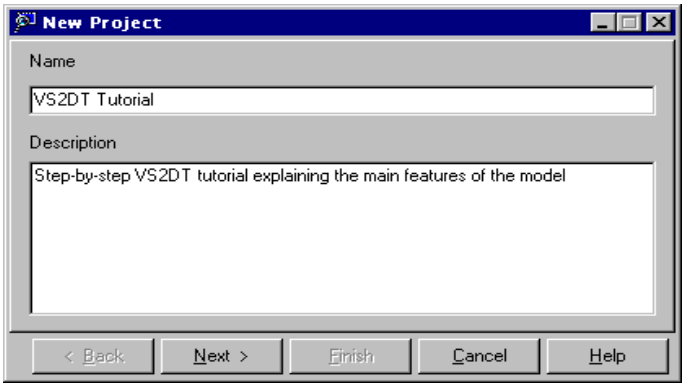
To create a new project, go to the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new WHI UnSat Suite project.

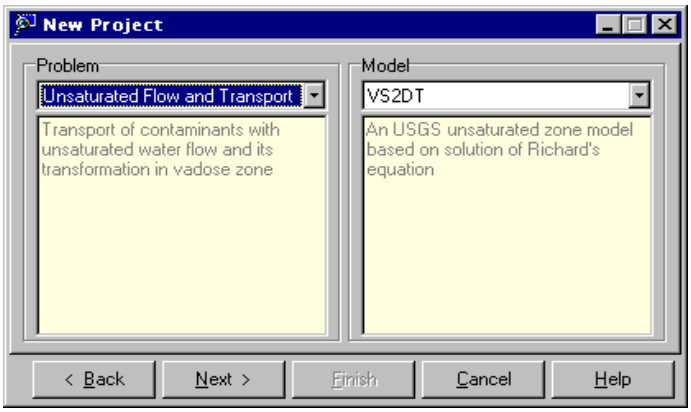
In the **Name** box type: **VS2DT Tutorial**.

In the Description box type: **Step-by-step VS2DT tutorial explaining the main features of the model.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, select **Unsaturated Flow and Transport** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that WHI UnSat Suite automatically recognizes the appropriate model that should be used with the Problem type selected.



☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

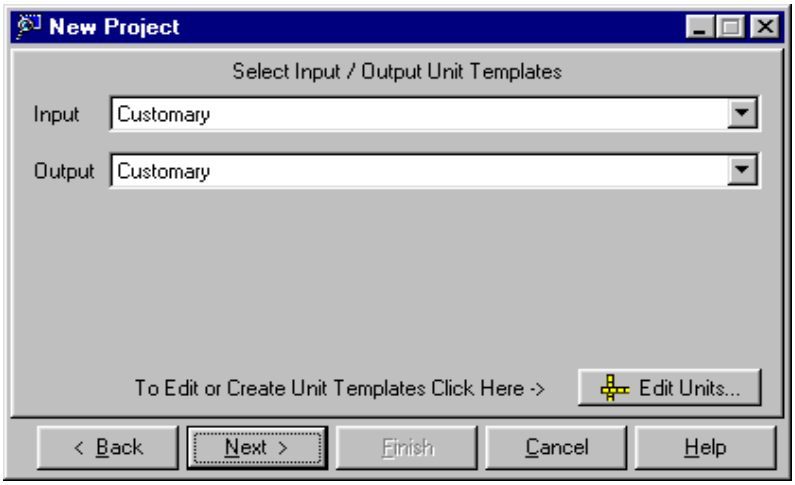
The WHI UnSat Suite Unit Converter was developed to maximize the flexibility for data input and output unit conversion. With the WHI UnSat Suite you may create your unit templates to fit the format of your data. For this exercise we will specify that all input and output will be in **Customary** (US) units.

From the **Input** drop-down list,

☞ **Customary**

From the **Output** drop-down list

☞ **Customary**



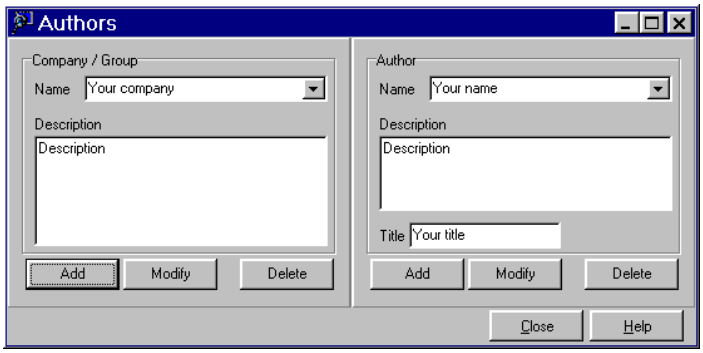
☞ **[Next]**

Assigning Authors and Clients

The WHI UnSat Suite allows you to create and maintain your own database of project Authors and Clients for use in future projects. The information which you input/specify here will appear automatically in your project report. To add a new author to the database

☞ **[Authors]**

The **Authors** dialog box will appear:



Each author is associated with a company.

Company/Group

In this area, you can add, modify, or delete companies and groups. To add your company to the database, in the **Name** box:

Type your company name

In the **Description** box:

type your address and phone number.

☞ **[Add]**

Author

In this area, you can add, modify, or delete authors from companies. To add a new author, input your information in the **Name** box, **Title** box, and **Description** box.

☞ **[Add]**

☞ **[Close]** to return to the **New Project** dialog box.

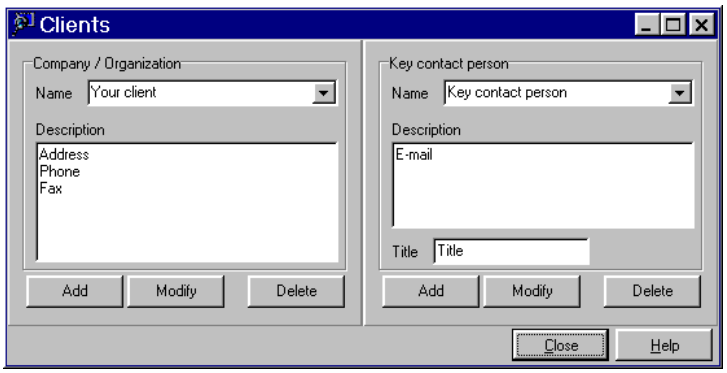
while back in the **New Project** dialog box, click the drop-down arrow and select your name from the list of the Authors.

Clients

To add a new client to the database:

☞ **[Clients]**

The **Clients** dialog box will appear



Each contact is associated with a company or organization.

Company/Organization

In this section you can add, modify, or delete companies and organizations. To add a new company, edit the information in the **Name** box and **Description** box, and click **[Add]**. To update an old company with current information, edit the information, and click **[Modify]**. To delete a company, select the company from the list, and click **[Delete]**.

Key Contact Person

In this area you can add, modify, or delete contacts from companies. To add a new contact, edit the information in the **Name** box, **Title** box, and **Description** box, and click **[Add]**. To update the

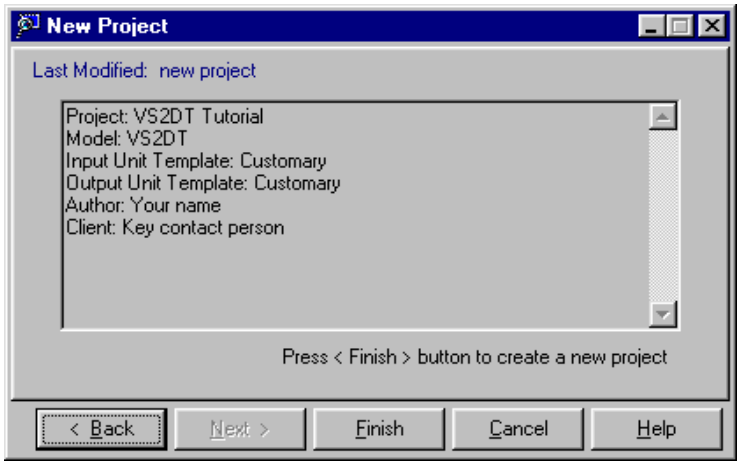
 Copyright 2000 Waterloo Hydrogeologic, Inc.

information about a client, edit the information, and click **[Modify]**. To delete a client, select the client from the list, and click **[Delete]**.

☞ **[Close]** to return to the **New Project** dialog box. Select an existing client from the list, or create a new client by clicking **[Clients].**

☞ **[Next]**

All the project information will be presented, as seen in the figure below:



☞ **[Finish]** to save the information as a project in the database and add it to the Project Tree.

After the project settings were specified, the **New Profile Wizard** will open.

Creating a Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:



In this dialog box you may select an already existing profile template or you can build a new profile 'from scratch'. In this demonstration exercise you will build a new profile.

☞ **[create new profile]**

☞ **[Next]**

The dialog box with default profile parameters will appear. In this dialog box we may assign the parameters of the simulated profile. The profile which you are going to simulate contains two layers. However, the New Profile Wizard allows you to specify only one layer for the profile at the beginning. At the later stage you will transform this initial profile into the one for the tutorial problem.

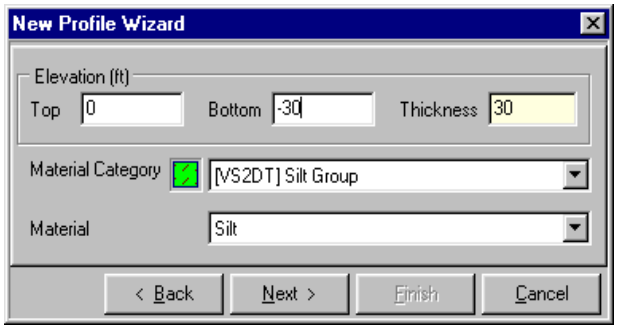
☞ in the **Bottom** text box and input **-30** for the elevation of the profile bottom. The value of **Thickness** will change accordingly.

☞ the drop-down arrow in the **Material Category** drop-down list box.

Select **[VS2DT] Silt group**. The default material **Silt** will appear in the **Material** drop-down list box.

Note: You may add as many materials to specific Material Category group as you need, once in the model, in Material Designer.

As a result of editing, the **New Profile Wizard** window will appear as below:



☞ **[Next]**

All the profile information will be presented:



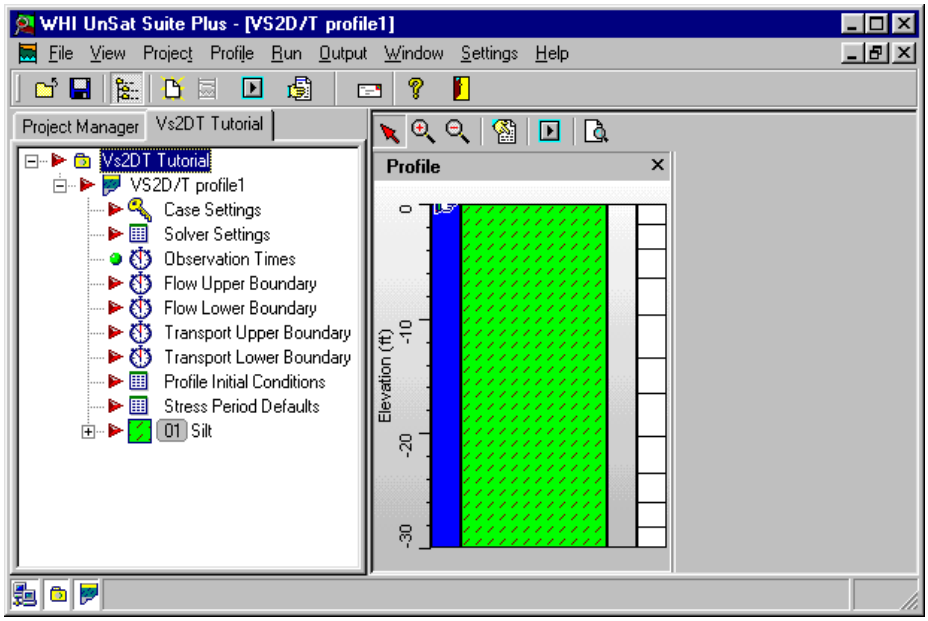
☞ **[Finish]** to load the profile.

The new project details will be added to the Project Tree View, the project will open and a profile picture will appear in the Profile View. This concludes creating a new project using the WHI UnSat Suite. We will now continue with designing the profile and setting specific simulation details.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Module II: Designing a Profile

The following picture will appear on your display:



Viewing the Profile

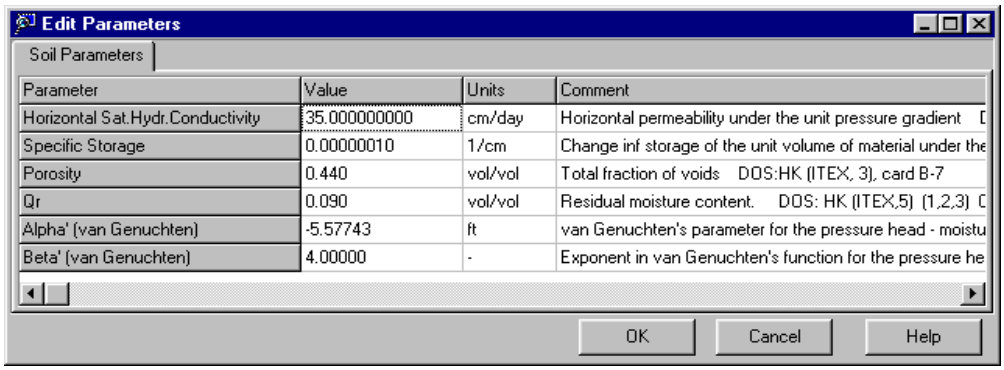
In the Profile View, located in the center of the WHI UnSat Suite window, you will see the loaded profile with a depth scale at the left and the Grid Column at the right.

A description of a layer will appear when the mouse pointer is pointed at the layer.

The Project Tree View in the left part of the WHI UnSat Suite window provides access to parameters for your simulation scenario. The icon and the name of the profile layer **Silt** are located in the bottom of the list.

☞ **'+'** sign in the box to the left of the **Silt** icon.

☞ ☞ **Soil Parameters**. The **Edit Parameters** dialog box will appear.



As you can see, the Project Tree View can be used to easily access Profile Layers, and Layer Parameters.

☞ **Cancel** to close the **Edit Parameters** dialog box

Now that you have become familiar with some viewing functions of the WHI UnSat Suite, you will learn how to set case specifications and edit the profile. In this part of the exercise you will:

- learn how to edit the layer structure of the VS2DT profile
- learn how to set type of simulated processes, boundary conditions and initial conditions of the profile
- learn how to set computational parameters of the model
- learn how to edit parameters of individual layers

Editing the Layer Structure of the Profile

To assist you in designing a profile, WHI UnSat Suite offers many convenient tools for resizing, splitting, merging, inserting and deleting layers. In this exercise, to get the required profile structure, you will split the existing layer into two layers, resize them to the thicknesses of the natural layers and, lastly, substitute the material of the lower layer with the Sandy Clay material.

Splitting a Layer

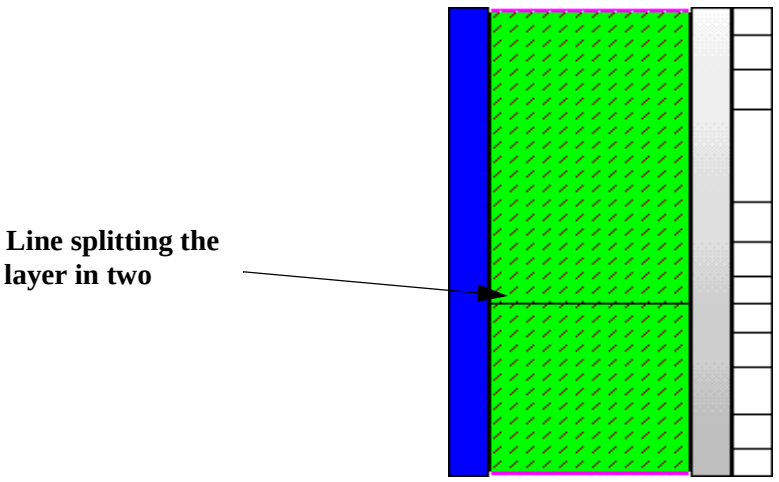
You can split a layer up into multiple sections and substitute materials for each section or assign different values of parameters for each section.

<right click> in the lower part of the **Silt** layer in the Profile View.

☞ **Layer/Split**

A line will appear through the layer at the cursor position and a new layer will appear in the project tree.

.



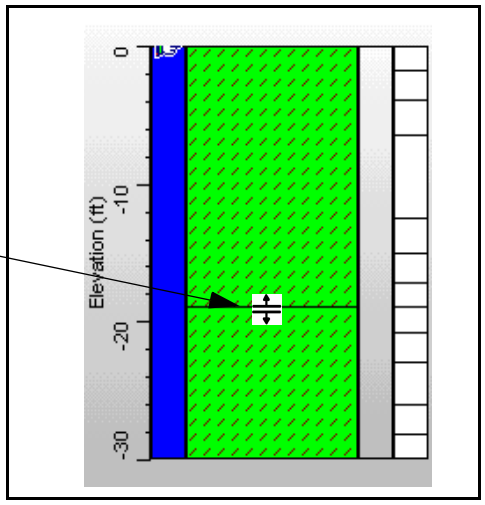
Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.

Resizing a Layer

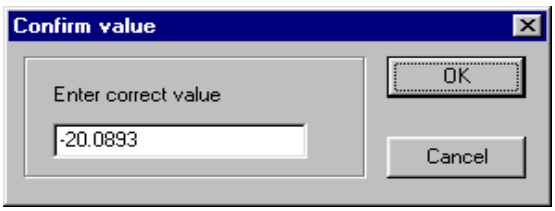
 Copyright 2000 Waterloo Hydrogeologic, Inc.

To test the resizing feature, move the mouse pointer on the **Silt** layer in the profile view and move the arrow upwards to the newly created boundary between two layers. As soon as the pointer encounters the layer's boundary, the pointer symbol changes:

New Cursor Symbol



Press the left mouse button and, holding the mouse button, drag the pointer downwards to approximately **-20** and release the mouse button. A **Confirm Value** dialog box will appear.



Type: -20.0 in the **Enter correct value** box.

☞ **[OK]**

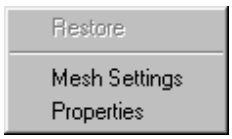
The profile will change according to the new setting specified.

Note: ***Moving the layer's boundary resizes both layers which are separated by this boundary, however it does not change the total thickness of the profile. You can also change the thickness of an individual layer.***

Substituting a Layer Material

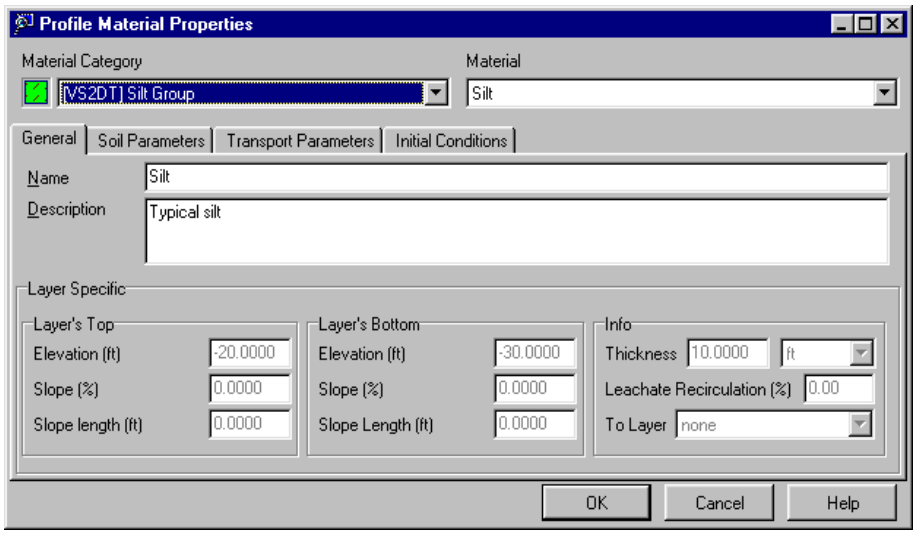
Finally, to get the desired profile, you have to substitute the silt material in the lower part of the profile with sandy clay material.

<right click> the name of the lower layer **Silt** in the Project Tree



☞ **Properties** from the drop-down menu.

A **Profile Material Properties** dialog box for the selected layer will appear:

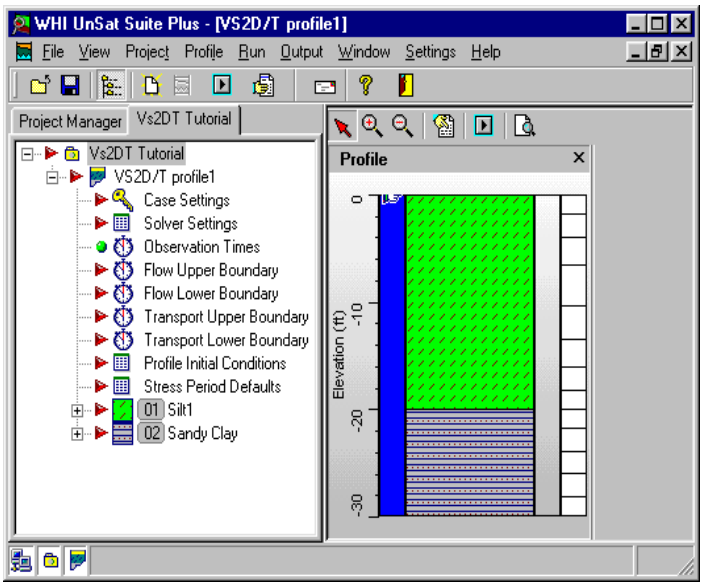


☞ the drop-down arrow in the **Material Category** drop-down list box.

☞ [VS2DT] **Sandy Clay Group** from the list. A default **Sandy Clay** material will appear in the Material drop-down list box. Parameters for this material were taken from the VS2DT manual with some additions from other literature sources.

Note: You may add as many materials with individual sets of parameters to specific Material Category group as you need using Material Designer.

☞ **OK** to confirm changes. A new layer will appear in the Profile View And Project Tree View:



- The profile structure inputted now into the model is the one which was specified in the Lab assignment. Finally, to have a profile structure completely set, we need to change the value of hydraulic conductivity for the Sandy Clay layer and set transport parameters for profile materials. However, we may not do it at this point. The list of soil hydrologic and transport parameters is dependent on the type of soil hydraulic function and transport process that is to be simulated. These parameters must therefore be set first. This can be done through the **Case Settings** param-

 Copyright 2000 Waterloo Hydrogeologic, Inc.

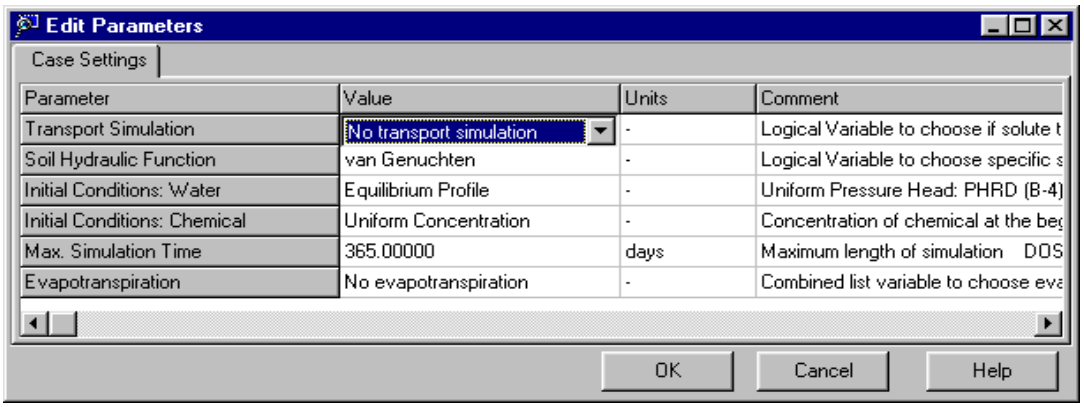
eter group. Next you will specify the type of simulated processes, boundary conditions and initial conditions of the profile and set computational parameters of the model. You will edit layer parameters at a later stage.

Setting Case Specifications

Case Settings

Constructing the unique simulation case requires you to make several decisions regarding which types of processes will be simulated and the boundary and initial conditions of the profile. These model options may be edited in the project parameter group labeled **Case Settings**. Most case settings, on selection, introduce additional parameters in the rest of the parameter groups. Thus, it is important to start entering data from the case settings parameter group.

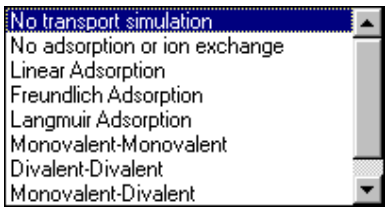
☞☞ on **Case Settings** in the **Project Tree View**.



The first item in this dialog window, **Transport Simulation,** is a logical variable that determines whether or not solute transport is simulated and if 'yes', which processes are to be taken into account.

☞ in the **Value** field beside **Transport Simulation**

☞ the drop-down arrow in the right corner that will appear. A drop-down list of allowed transport processes will appear.



Below is a short description of the different choices.

1) No transport simulation *(none of the transport processes are simulated),*

2) No Adsorption or Ion Exchange *(advection, hydrodynamic dispersion and decay are simulated),*

3) Linear Adsorption *(advection, hydrodynamic dispersion, decay and* ***linear adsorption*** *are simulated),*

4) ***Freundlich Adsorption*** *(advection, hydrodynamic dispersion, decay and* ***Freundlich*** *adsorption are simulated),*

5) ***Langmuir Adsorption*** *(advection, hydrodynamic dispersion, decay and* ***Langmuir*** *adsorption are simulated),*

6) ***Monovalent-Monovalent*** *(advection, hydrodynamic dispersion, decay and* ***Monovalent-Monovalent*** *ion exchange are simulated),*

7) ***Divalent-Divalent*** *(advection, hydrodynamic dispersion, decay and* ***Divalent-Divalent*** *ion exchange are simulated),*

8) ***Monovalent-Divalent*** *(advection, hydrodynamic dispersion, decay and* ***Monovalent-Divalent*** *ion exchange are simulated),*

9) and ***Divalent-Monovalent*** *(advection, hydrodynamic dispersion, decay and* ***Divalent-Monovalent*** *ion exchange are simulated).*

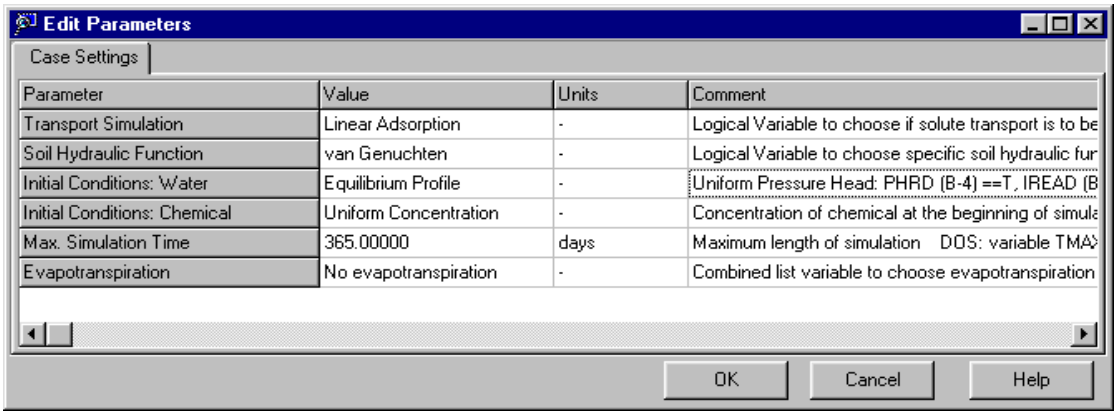
For this exercise we will specify the **Linear Adsorption** selection to be the ruling combination of processes. This selection allows suffficiently sophisticated representation of soil chemistry and utilizes widely available parameters.

☞ **Linear Adsorption** from the drop-down list.

Note: ***a group of transport parameters will be assigned to all profile layer materials if any of the transport simulation types is selected except when there is no transport simulation. The transport parameters can be edited for each layer separately.***

Soil Hydraulic Function

Soil Hydraulic Function is the second item in the Case Settings dialog window which defines the specific soil hydraulic function used for the simulation. Options include: **Brooks and Corey**, **van Genuchten**, and **Haverkamp**. In this exercise we will use the most popular **van Genuchten's** function which is set by default.



Initial Conditions

Initial water and chemical distributions may be set either as uniform through the profile when all layers have the same conditions or non-uniform where each layer may have a unique value. Depending on the selections made, the initial condition value must be entered under ***Soil Parameters*** *for each layer (for non-uniform case) or under* ***Profile Initial Conditions*** *for the whole profile (for uniform case).*

☞ in the **Value** field beside **Initial Conditions: Water**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ the drop-down arrow in the right corner that will appear. A drop-down list of allowed initial conditions for water include: **Uniform Moisture Content, Uniform Pressure Head, Equilibrium Profile, Nonuniform Moisture Content, and Nonuniform Pressure Head**.

For this exercise we will select the **Equilibrium Profile** initial condition which presents the balanced unsaturated profile existing above the groundwater surface set at specific depth.

☞ **Equilibrium Profile** from the drop-down list.

Note: Values of the groundwater depth and minimum pressure head within a profile will be set later in the Profile Initial Conditions parameter group

☞ in the **Value** field beside **Initial Conditions: Chemical**

☞ a drop-down arrow in the right corner that will appear. A drop-down list of allowed initial conditions for chemical include: **Uniform Concentration, Nonuniform Concentration.**

For this exercise we assume that the profile was free of contaminant at the beginning of the simulation.

☞ **Uniform Concentration**

Note: Values of the initial concentration within a profile (which is 0 in this exercise) will be set later in the Profile Initial Conditions parameter group.

Maximum Simulation Time

The maximum simulation time determines the length of simulation, but does not activate any additional parameters regardless of the value inputted.

In this exercise we will simulate a 10 year period. To view available time units,

☞ **Units** field beside the **Maximum Simulation Time**. A drop-down list with available units will will appear. Leave 'day' units in this field.

☞ **Value field** beside the **Maximum Simulation Time.**

Type: **3650**

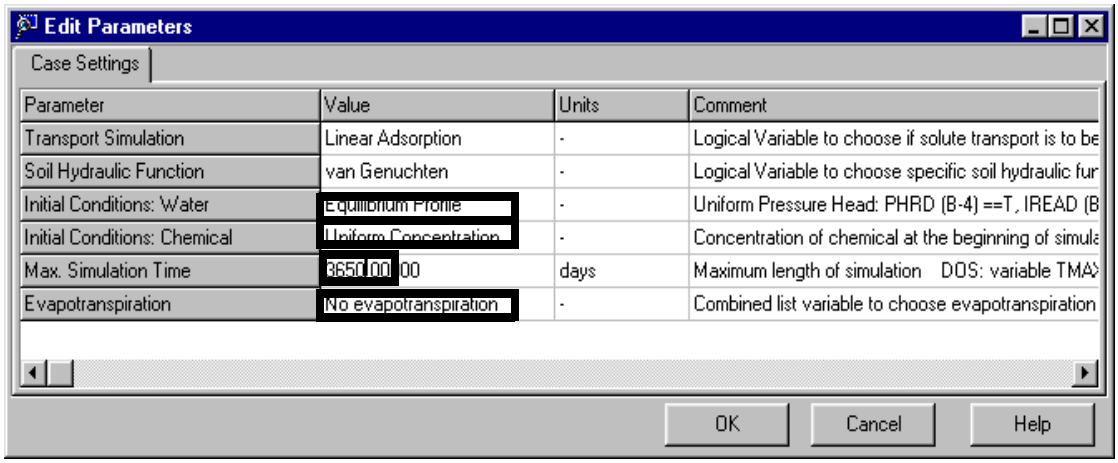
Evapotranspiration

This parameter switch is used to set Evapotranspiration at the upper boundary of the profile. In this exercise we are going to use the results of the HELP simulation to set the upper boundary condition of the soil profile for water. HELP allows the calculation of an integrated value of flux through the upper profile boundary (including evapotranspiration) in the most accurate way available.

☞ the **Value** field beside the Evapotranspiration

☞ **No evapotranspiration**

The result of editing the **Case Settings** parameter group should look like the following:



☞ **OK** to confirm changes and close the **Case Settings** parameter group

Solver Setting

Solver settings *parameter group allows the user to specify how the flow and transport equations are to be solved for each time step.*

☞☞ *on* ***Solver Settings*** *in the* ***Project Tree View***

The parameters that can be altered are:

- *Maximum and minimum number of iterations per time step;*
- *Maximum number of time steps for the whole simulation;*
- *Relaxation parameter for how aggressively the solver converges on the solution;*
- *The equation solution accuracy or closure criteria;*
- *Calculation method for achieving the weighted hydraulic conductivity between cells (for all but a few cases, the geometric mean provides the most accurate values); and*
- *Space Differencing*
- *Time Differencing*
- *Display or not display Profile Balance after Every Time Step*

These functions are pre-set to the most appropriate values and do not have to be specified for every new project: generally the default values will yield reasonable results. However, if the numeric solution becomes unstable and the output variables oscillate, correction of these functions is the only way to get a reliable result.

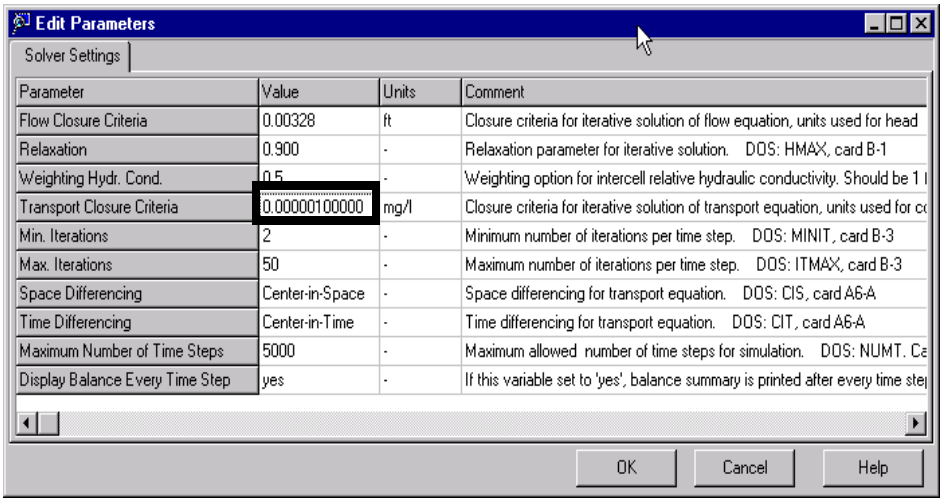
Because the inflow concentration of lead is relatively small (parts of mg/l), in this exercise we will set a finer value for **Transport Closure Criteria**.

☞ **Value** field beside the **Transport Closure Criteria**

Type: **0.000001**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

As a result of editing, the **Solver Settings** parameter group should appear as below:



☞ **OK** to confirm changes and close the **Solver Settings** parameter group.

Boundary Conditions

Simulations with the VS2DT model require boundary conditions to be specified.

In VS2DT the upper boundary is always set at the profile surface while the location of the lower boundary may be specified at any point within the profile including points below the phreatic surface.

Two sets of parameters need to be specified at the upper and lower profile boundaries:

- *flow conditions*
- *transport conditions*

*The following types of conditions are allowed at the profile **Flow Upper Boundary** which reflects interaction between the soil profile and the atmosphere or the upper soil layers:*

- *specified pressure head,*
- *specified flux,*
- *specified total head,*
- *evaporation,*
- *transpiration,*
- *evaporation and transpiration,*
- *and no specified boundary (impermeable boundary)*

*The following types of conditions are allowed for the profile **Flow Lower Boundary** which reflects interaction between the soil profile and the surrounding soil:*

- *specified pressure head,*
- *specified flux,*
- *specified total head,*

- *and no specified boundary (impermeable boundary)*

*The **Upper** and **Lower Transport Boundaries** do not differ in the type of condition they are able to simulate. For both boundaries the following types of conditions are allowed:*

- *specified concentration,*
- *specified mass flux,*
- *and no specified boundary*

In VS2DT boundary conditions are time dependent. They have to be specified for the stress periods of variable length within which your knowledge of boundary processes allows you to presume uniformity. The length of stress periods is determined by the user. By managing types of boundary conditions and length of stress periods you may approximate any situation occurring at the upper and lower boundaries of the simulation domain. For recommendations on which type of boundary condition to be used to simulate a specific practical case, look in the UnSat Suite manual.

Flow Upper Boundary

In this Lab for the Flow Upper Boundary you are using the results of a Visual HELP simulation which calculated a value of average infiltration intensity of 4 in/year (per unit area). You will apply this value through the whole simulation.

☞☞ on **Flow Upper Boundary** group in the Project Tree View

☞ in the **End Time Days** field

Type: 3650 (total length of simulation in days)

☞ in the **Boundary type** field. A drop-down arrow will appear.

☞ the drop-down arrow and

☞ **Flux** from the drop-down list

☞ in the **Units** field

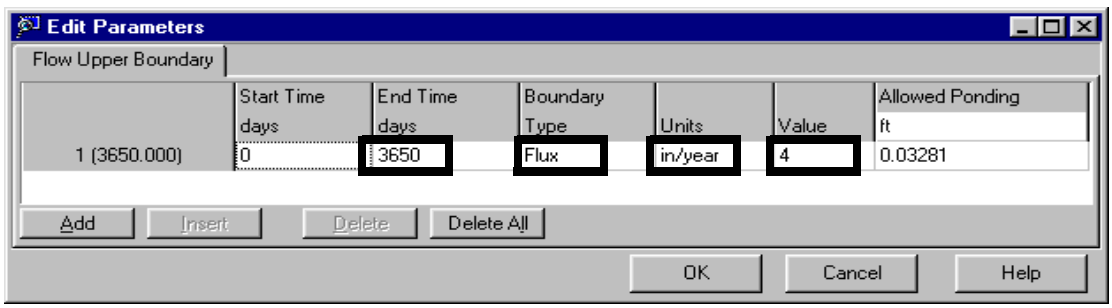
☞ the drop-down arrow

☞ **in/year** from the drop-down list

☞ in the **Value** field

Type: **4**

As a result of your editing, the Flow Upper Boundary dialog box will appear as below:



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **OK** to confirm entered values.

Flow Lower Boundary

In this Lab for the Flow Lower Boundary the groundwater depth is 25 feet. You will apply this value through the whole simulation. Because the total thickness of the profile is 30 feet, the lowest 5 feet will simulate the upper part of the aquifer. To simulate the stable groundwater depth at a certain location, the Pressure Head boundary condition is to be used. By the theory of unsaturated flow, the pressure head equals 0 at the phreatic surface and grows with a unit gradient downwards in the aquifer. In our case, the pressure head at the bottom of the profile (at elevation -30 feet) equals 5 feet.

☞☞ on **Flow Lower Boundary** group in the Project Tree View

☞ in the **End Time Days** field

Type: 3650 (total length of simulation in days)

☞ in the **Boundary type** field

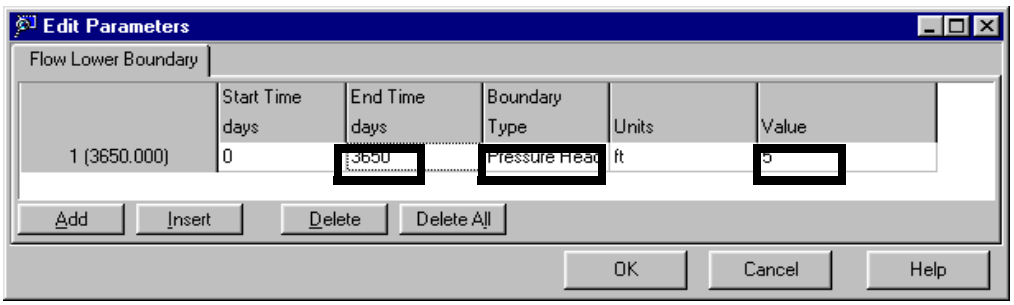
☞ the drop-down arrow

☞ **Pressure Head** from the drop-down list, the **Units** field should read **ft** (for feet)

☞ in the **Value** field

Type: 5

As a result of your editing, the **Flow Lower Boundary** dialog box will appear as below:



☞ **OK** to confirm entered values.

Transport Upper Boundary

In this Lab for the Transport Upper boundary we are using results of the lysimeter testing which are presented in the following table:

Length of a period (years)	Concentration of lead (mg/l)
1	0.66
1	0.22
7.4	0.12
Rest of time	0.1

☞☞ on **Transport Upper Boundary** group in the Project Tree View

☞ in the **End Time Days** field

Type: 365 (length in days).

☞ in the **Boundary type** field

☞ **No Specified Boundary.**

Note: You may notice the other available type of transport boundary, namely 'Concentration', in the boundary type list. In case of inflow, both types of boundary condition work nearly the same except one detail: if the 'Concentration' condition is assigned, the model will simulate an additional diffusive flux into the domain because the concentration in the uppermost boundary node will be higher than that in the adjacent domain node. This may lead to simulation of an incorrect mass balance. If 'No Specified Boundary' is selected, the Inflow concentration will be directly input into the upper node of the domain.

Skip the **Units** and **Value** fields

☞ in the **Inflow Concentration** field

Type: **0.66**. With this you have finished entering transport boundary data for the first period.

☞ **Add**. A new line will appear below the one previously entered. By default UnSat Suite reproduces the previous time step's settings.

☞ in the **Inflow Concentration** field of the second line

Type: **0.22**

☞ **Add**

For the third time interval:

Type: 3433 in the **End Time Days** field (which corresponds to the end of the period at 9.4 years)

Type: 0.12 in the **Inflow Concentration** field

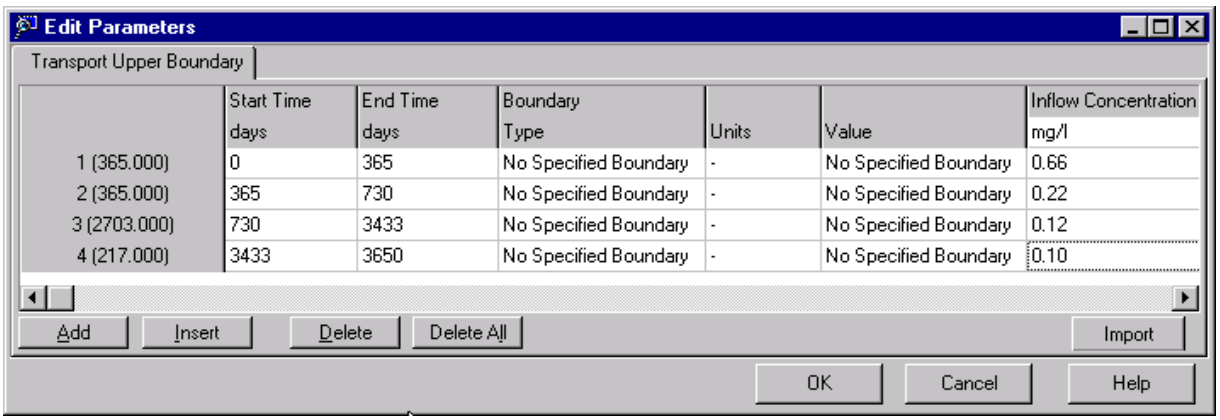
☞ **Add**

For the fourth time interval:

Type: 3650 in the **End Time Days** field (which corresponds to the end of the simulation period).

Type: 0.10 in the **Inflow Concentration** field

As a result of your editing, the Transport Upper Boundary dialog box will appear as below:



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **OK** to confirm entered values.

Transport Lower Boundary

In this Lab, for the Transport Lower boundary, you will use 'No specified Boundary' condition which is recommended for profiles that include part of saturated zone

☞☞ on **Transport Lower Boundary** group in the Project Tree View

☞ in the **End Time Days** field

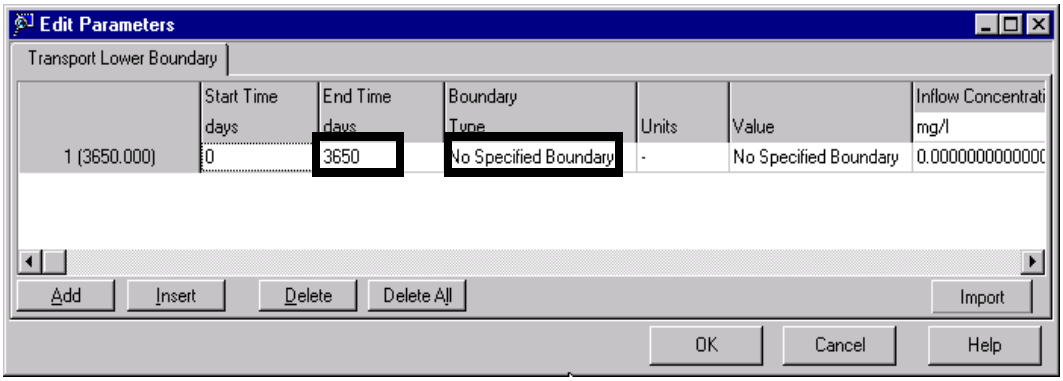
Type: 3650.

☞ in the **Boundary type** field

☞ **No Specified Boundary** from the list

The **Units** field should read '-', the **Value** field should read **No Specified Boundary**. The **Inflow Concentration** field which describes concentration of contaminant in case of inflow into simulated domain, should read **0.000**

As a result of your editing, the **Transport Lower Boundary** dialog box will appear as below**:**



☞ **OK** to confirm entered values. With this last action we have finished entering boundary conditions.

Setting Output Times

UnSat Suite allows you to view model results for all time steps. However, your desired output times may not coincide with the default time steps. In this case, you can specify your own output times for presenting simulation results.

☞ ☞**Observation Times** in the Project Tree

☞ in the **Time** field for the first period (first line)

Type: **0.1** Leave the **Balance Summary** field with default **yes** selection.

☞ **[Add],** to add another observation time. When you click **Add** for the first time, the first line will be copied.

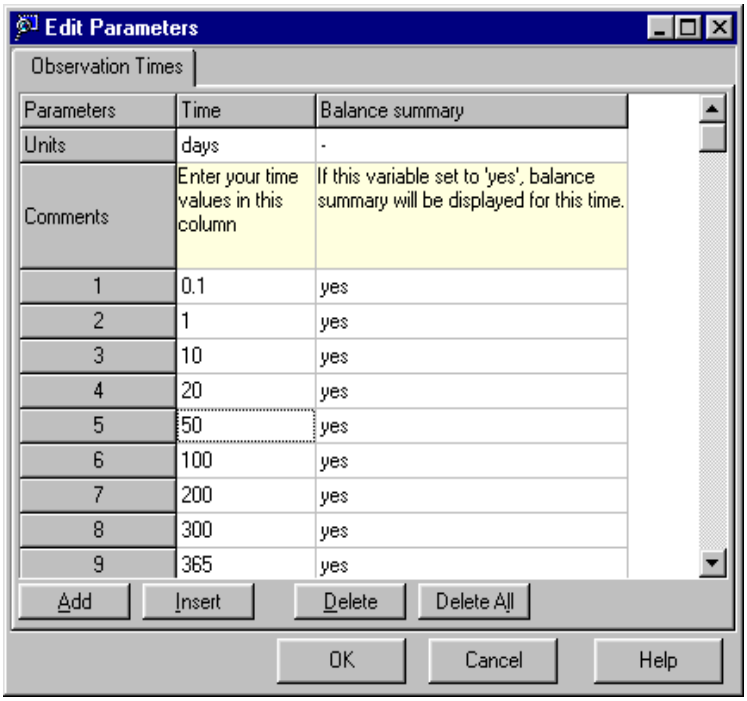
☞ in the **Time** field for the second period

Type: **1**

☞ **[Add]**

Type: **10** in line 3, **20** in line 4, **50** in line 5 and continue inputting times **100, 200, 300, 365, 730, 1095, 1460, 1825, 2190, 2555, 2920, 3285 and 3650** clicking **[Add]** after each input. While doing this you will notice a helpful feature of WHI UnSat Suite which allows you to add time steps with constant length automatically.

The **Observation Times** dialog box will appear as below:



☞ **OK** to confirm and save entered observation times.

Setting Values for Initial Conditions

To run the model, the values for the profile initial conditions have to be set. As you may remember, the types of initial conditions were selected before in the **Case Settings** parameter group. The following selections were made:

Initial Condition, Water: **Equilibrium Profile**

Initial Condition, Chemical: **Uniform Concentration**

☞☞ on **Profile Initial Conditions** group in the Project Tree View

☞ in the **Value** field beside **Groundwater Depth**

Type: **25**

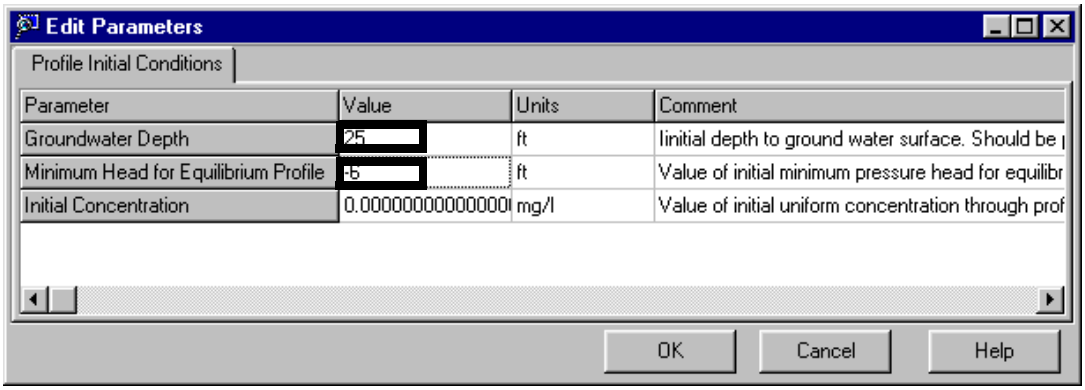
☞ in the **Value** field beside **Minimum Head for Equilibrium Profile**

Type: **-6** (which corresponds to silty soil at field capacity)

Leave the default value **0.000** in the field beside **Initial Concentration**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

As a result, the **Profile Initial Conditions** dialog box will appear as shown below:



☞ OK to confirm and save entered values.

Customizing the Finite Difference Grid

[VS2DT] uses the finite difference method to approximate the flow and transport through the profile. To utilize this method, each layer must be separated into finite difference cells. The size of these cells can affect the convergence of the set of equations being solved. Furthermore, solution convergence is more difficult at the interface between layers in the profile where there may be large differences in the properties of the individual layers.

Solution convergence is improved by using reduced grid spacing near the outer boundaries or the boundaries of the profile layers. In the middle of the layer, the finite difference grid does not need to be as fine as at the boundaries. The grid design of fine grid spacing near the layer boundaries and larger grid spacing in the middle of the layer minimizes computational time and improves solution stability and time to convergence.

The UnSat Suite interface allows the user to set the grid quickly and effectively. Built in functions allow the user to customize parameters of the finite difference grid for each individual layer.

To set the grid characteristics:

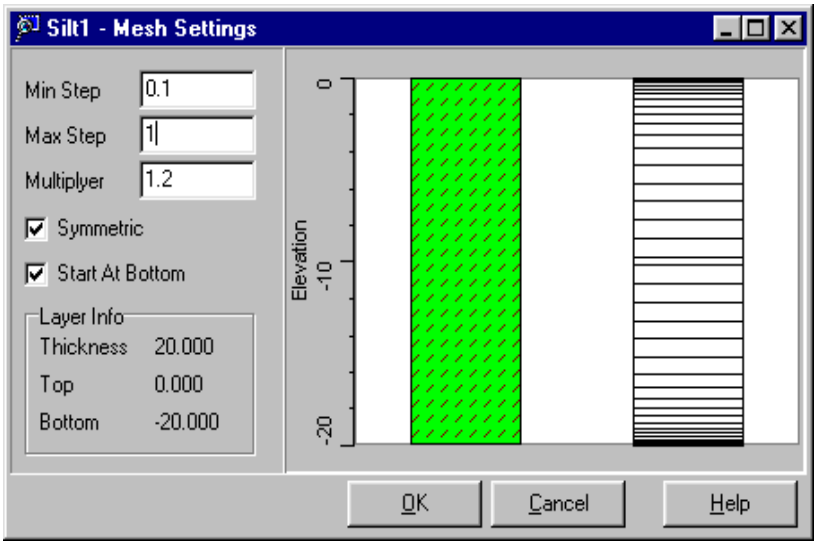
<right click> on the **Silt1** layer in the Project Tree View and select **Mesh Settings**

The **Silt - Mesh Settings** dialog box will appear.

Type: **0.1** in the **Min. Step** text box.

Type: **1** in the **Max. Step** text box.

Notice that the mesh image to the right of the layer picture is changing to show the results of editing:



☞ **[OK]** to save changes.

<right click> on the **Sandy Clay** layer in the Project Tree View and select **Mesh Settings**

In the **Sandy Clay - Mesh Settings** dialog box:

Type: **0.05** in the **Min. Step** text box.

Type:**0.5** in the **Max. Step** text box.

☞ **[OK]** to save changes.

Setting Stress Period Defaults

In the stress period defaults section, you can specify the factors associated with time steps.

The length of the initial time step can be edited along with the maximum and minimum length of time steps to follow. You can also specify a multiplier for the initial time step length. Therefore, time steps will get larger or smaller depending on the nature of the simulation.

However, complications may arise during simulation. If convergence can not be achieved during a time step, a factor for time step reduction can be modified. In this exercise you will use default settings for all parameters except for ***Head Criterion.***

☞☞ **Stress Period Defaults** parameter group in the Project Tree View

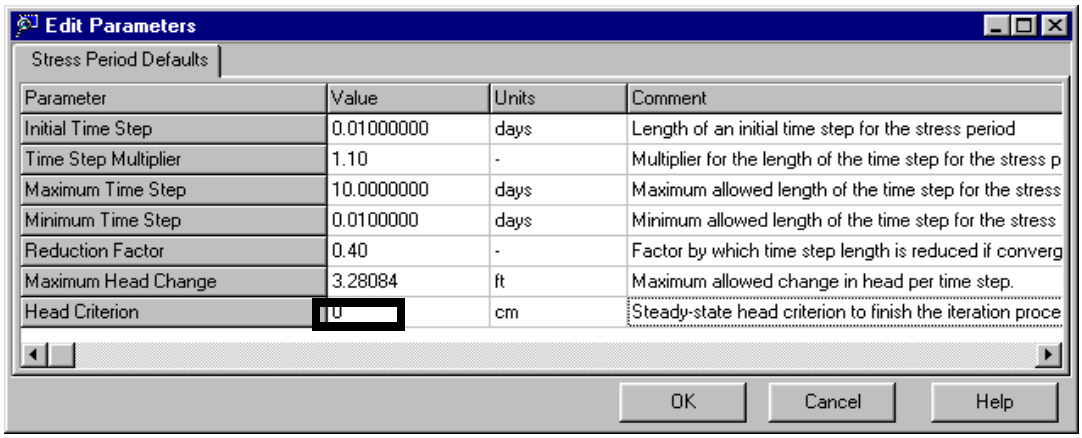
In the dialog box that appears:

☞ in the **Value** field beside **Head Criterion**

Type: **0**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The Stress Period Defaults dialog box will appear as shown below:



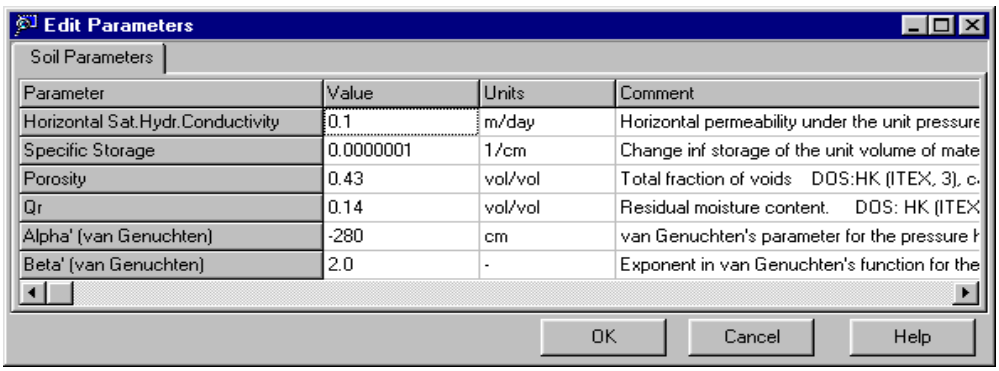
☞ [**OK**] to save changes.

You have now completed editing the parameter groups and setting the simulation scenario. Next you will edit layer parameters.

Editing Layer Parameters

☞ the '+' sign in the box to the right of the **Sandy Clay** icon.

☞ ☞ **Soil Parameters.** The **Edit Parameters** dialog box will appear:



In this dialog box, you can view and edit general parameters of the layer.

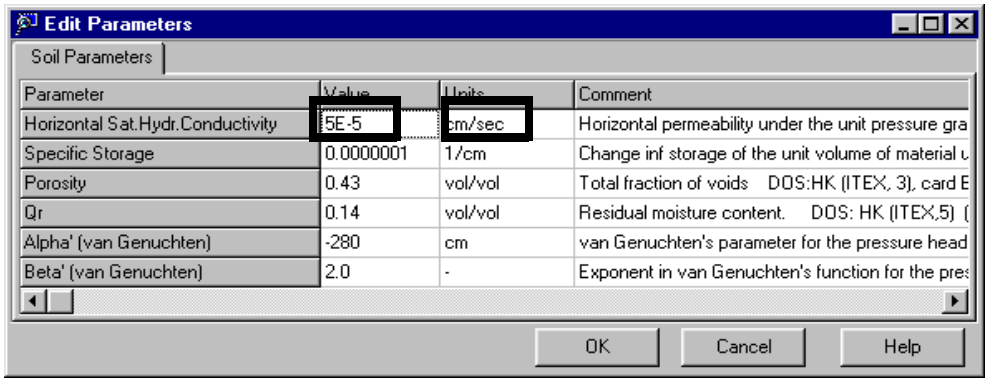
☞ **Units** field beside the **Horizontal Sat. Hydr. Conductivity**, a drop-down list with available units will appear.

☞ **cm/sec**

☞ **Value** field beside the **Horizontal Sat. Hydr. Conductivity**

Type: **5E-5**

The new **Edit Parameters** tab will look like the figure below:



☞ **[OK]**

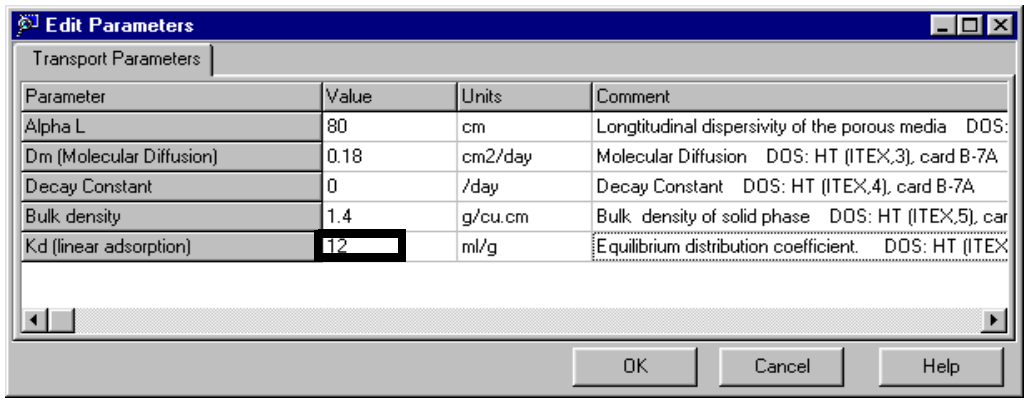
In addition to **Soil Parameters** we have to edit transport parameters of the layers.

☞ ☞ **Transport Parameters** for the **Sandy Clay** layer. The **Edit Parameters - Transport Parameters** dialog box will appear

☞ **Value** field beside **Kd [linear adsorption]**

Type: **12**

As a result of editing, the **Transport Parameters** dialog box should appear as shown below:.



☞ **[OK]** to save changes and close the dialog box.

☞ **[-],** beside **Sandy Clay** in the Project Tree.

☞ the '+' sign in the box to the right of the **Silt1** icon.

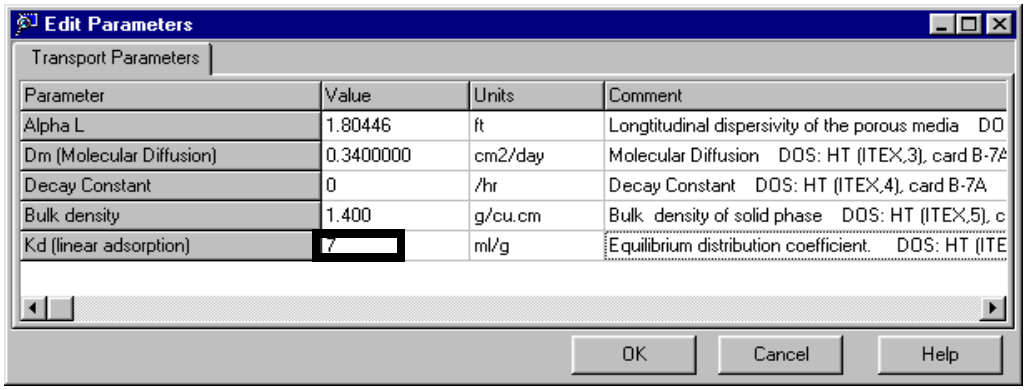
☞ ☞ **Transport Parameters** for the **Silt1** layer.

☞ **Value** field beside **Kd [linear adsorption]**

Type: **7**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

As a result of editing, the **Transport Parameters** dialog box should appear as below:



☞ **[OK]** to save changes and close the dialog box.

This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **VS2DT tutorial 1**, which was specifically developed for you and contains all changes you have done so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

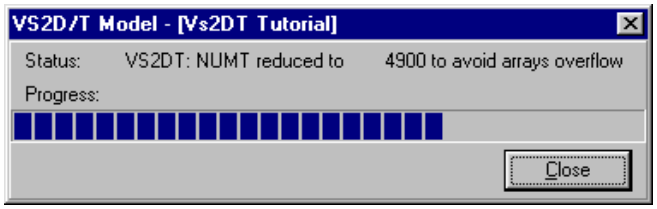
☞ ☞ project **VS2DT tutorial 1.**

Module III: Running the Model, Viewing Output, and Reporting

Running the VS2DT Model

To run the model for a single profile, click the **Run** icon  above the Profile View:

A progress bar will appear to indicate the computation progress, as shown below:



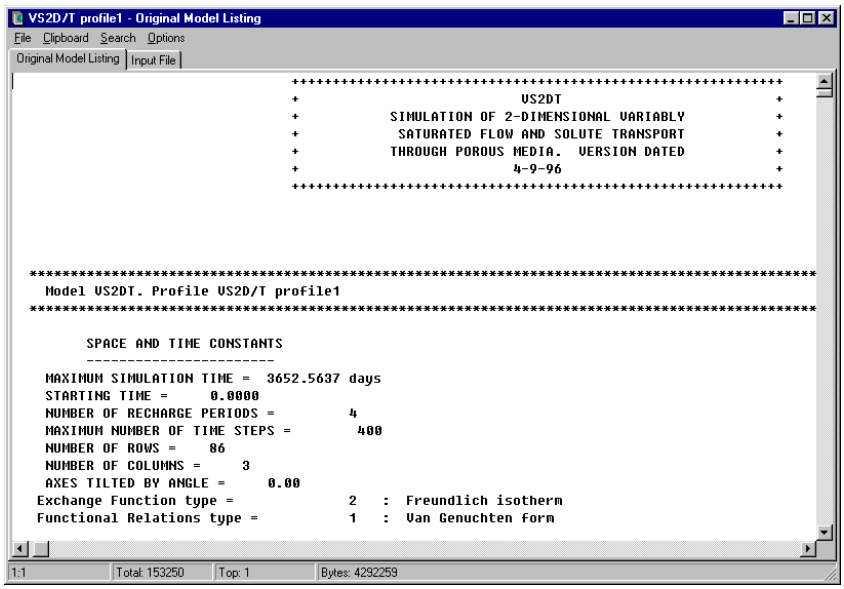
Viewing the Original VS2DT Input and Output Files

To view the original VS2DT output:

☞ **Output** in the main menu, and then

☞ **View Original Listing**

The following window will appear:



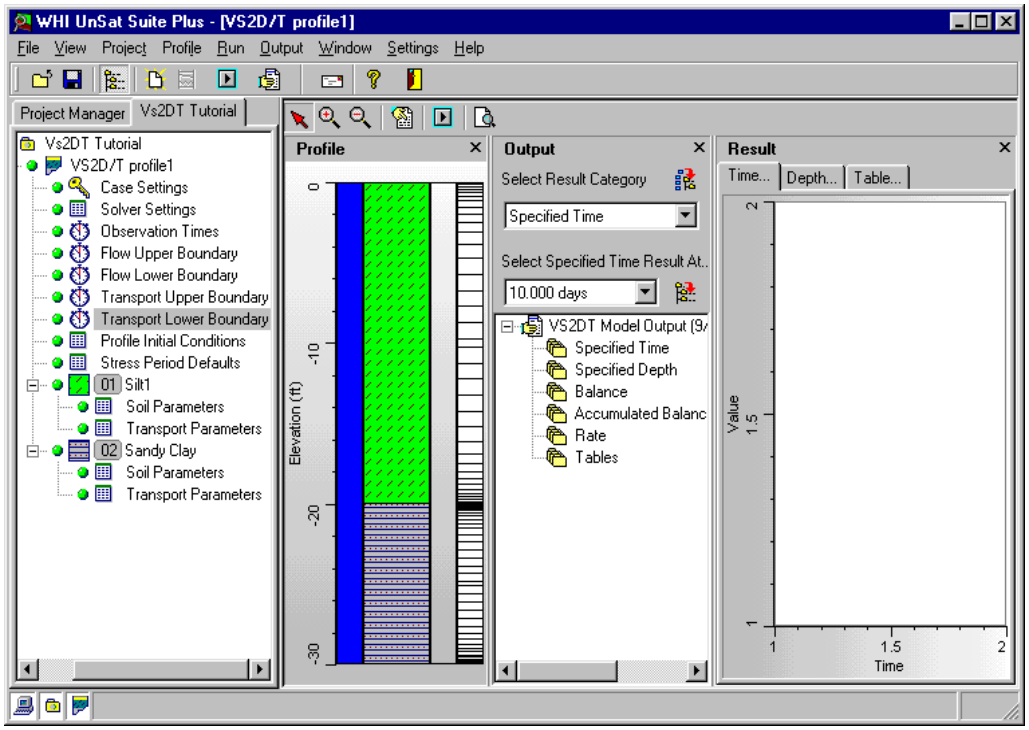
Here you can view the original listing, find specific expressions, mark and print parts or make a complete printout of the model results.

☞ the **X** in the top right corner to close the original listing.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Viewing the Output Graphs

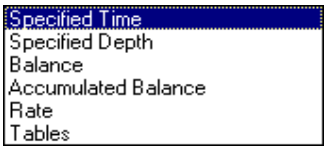
After the model has successfully ran, the Output View and Result View window will open and the UnSat Suite window will appear as below:



To enlarge the viewing area click the icon  to close the Project Tree View, or click the '**X**' in the Profile View to close it.

In the WHI UnSat Suite the output is grouped in two levels. First you have to select a general output category and then select a specific variable within the selected category. To select the output category:

☞ the drop-down arrow in the **Select Result Category** list box. The possible result groups will appear in the list box below.



*The **Result Categories** present the following:*

- ***Specified Time** - plots head, moisture content, saturation, flow velocity and contaminant concentration in soil water versus depth for specified observation times.*
- ***Specified Depth** - plots head, moisture content, saturation, flow velocity and contaminant concentration in soil water versus time for specified depths (cell centers).*
- ***Balance: Flow** - plots the amount of water that entered or left the unsaturated profile (per unit area 1 sq.cm), change in amount of stored water and balance discrepancy versus time.*
- ***Balance: Transport** - plots the amount of contaminant that entered or left the unsaturated profile (per unit area 1 sq.cm) change in amount of stored contaminant and balance discrepancy versus*

time.

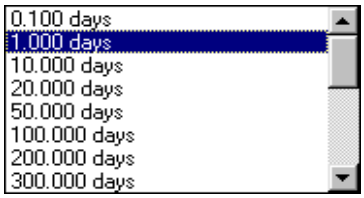
- ***Accumulated Balance: Flow*** *- plots the accumulated amounts of water that entered or left the unsaturated profile (per unit area 1 sq.cm), accumulated change in amount of stored water and balance discrepancy versus time.*
- ***Accumulated Balance: Mass*** *- plots the accumulated amounts of contaminant that entered or left the unsaturated profile (per unit area 1 sq.cm), accumulated change in amount of stored contaminant and balance discrepancy versus time.*
- ***Rate: Flow*** *- plots the rate at which water entered or left the unsaturated profile (per day and per unit area 1 sq.cm), change in amount of stored water and balance discrepancy versus time.*
- ***Rate: Mass*** *- plots rates at which the contaminant entered or left the unsaturated profile (per day and per unit area 1 sq.cm) change in amount of stored contaminant and balance discrepancy versus time.*
- ***Table: Flow Balance*** *- presents flow balance in tabular form*
- ***Table: Mass Balance*** *- presents contaminant balance in tabular form*
- ***Table: Accumulated Flow Balance*** *- presents accumulated flow balance in tabular form*
- ***Table: Accumulated Mass Balance*** *- presents accumulated contaminant balance in tabular form.*

Next you will view some types of the output and get acquainted with some of the WHI UnSat Suite viewing and reporting features.

Selecting Result Category: Specified Time and Depth

☞ **Specified Time** in the **Select Result Category**

☞ the drop-down arrow in the **Select Specified Time Result at...** list box. The list of observation times will appear:

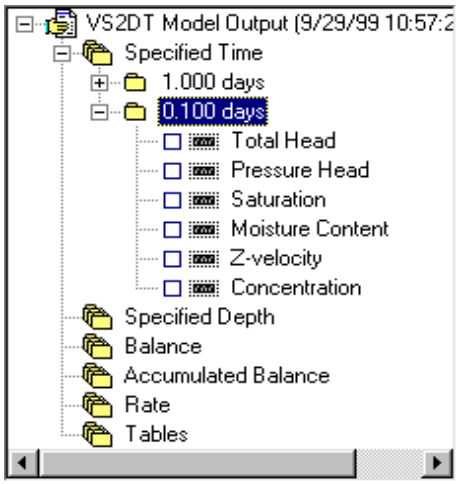


☞ **0.100 days** which is the first available output

To view all results available for this specific time, click the icon  to the right of the **Select Specified Time Result at...** box.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

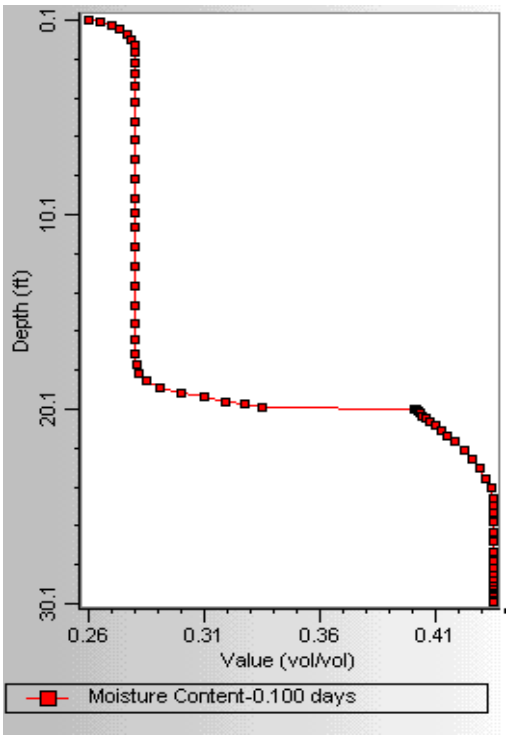
The list of available results will open in the Result Tree:



Plotting variable: Moisture Content

☞ the check box beside **Moisture Content**.

The graph of the variable will appear in the Result View window:



The graph shows full saturation in the aquifer 25 feet below the surface and a rapid change in moisture content at a depth of 20 feet at the boundary between two different soil types.

Lets watch changes of the moisture profile in time.

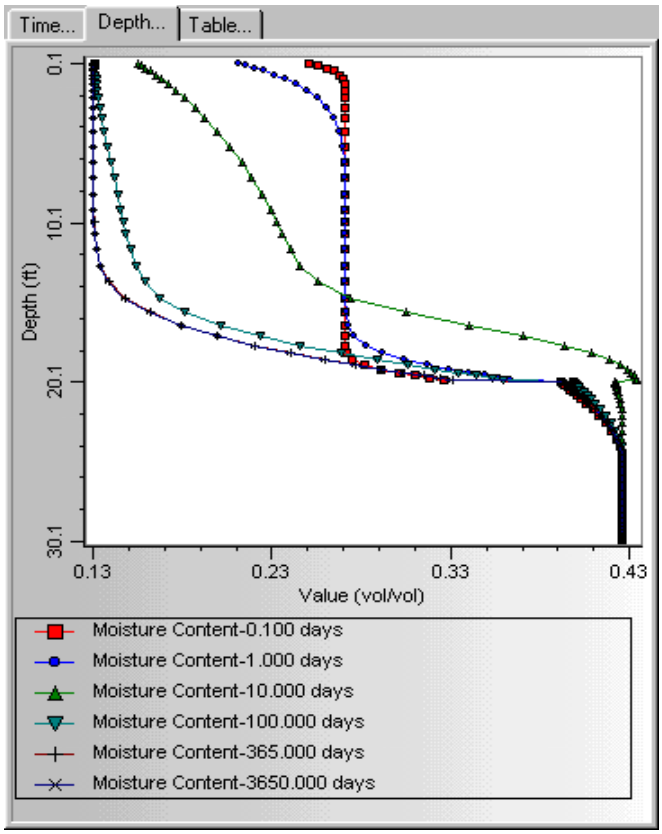
☞ the drop-down arrow in the **Select Specified Time Result at...** list box.

☞ **1 days** (you would need to scroll the bar to reach this line)

click the  icon to the right of the **Select Specified Time Result at...** box

☞ the check box beside **Moisture Content** from the Output Tree

Repeat this operation for **10 days, 100 days, 365 days** and **3650 days.** As you select the variable for specific time, it will appear in the graph. Finally the graph will look like the following:



The graph shows that considerable changes in moisture profile occurred after the asphalt pavement was placed. The moisture content which was in balance with naturally percolating rain and snow waters decreased dramatically after the top of the profile was sealed by 4 inches of asphalt. The moisture content curve stabilized at a new position after a year from the moment when asphalt was placed. The excess of water has infiltrated down to the aquifer.

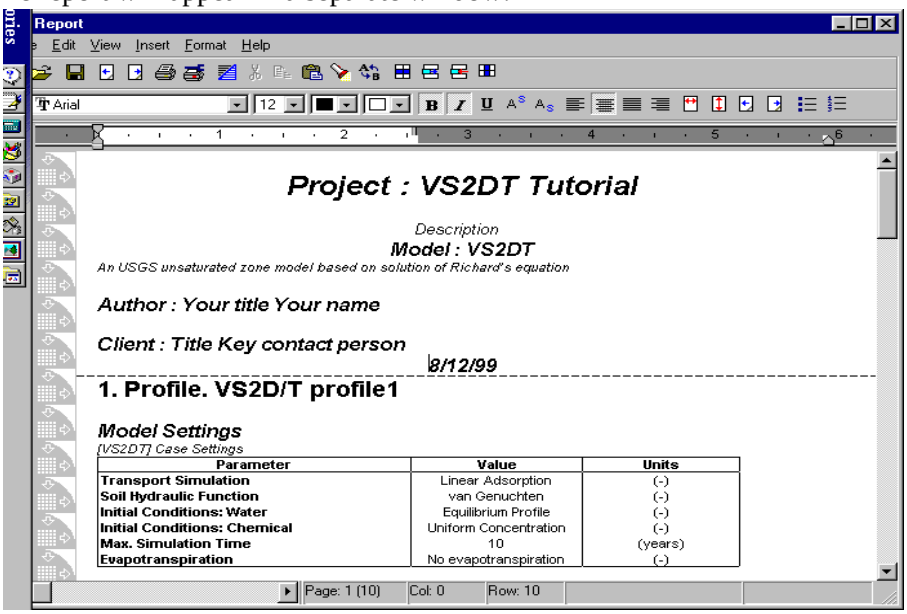
At this point you may want to start creating your project report and add the above graph to it.

Creating a Report

To present results of your VS2DT simulation to your clients you may use the UnSat Suite Report Generator.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

To create a report and add the project input data, click the icon  from the Operational Icons tool bar. The report will appear in a separate window:

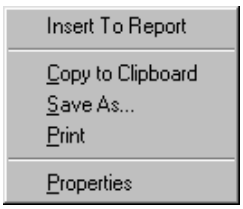


In this window you may edit the report, input your own text and add any types of graphics or table outputs produced by UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph to the report:

Scroll the bar in the **Report** window to the bottom and place the cursor to a position below the last table in the report. Return to the main window of the WHI UnSat Suite and <right click> in the Result View. The following menu will appear:



☞ **Insert To Report**. The graph or table will appear in the report.

The graph may appear smaller than the original. To view the graph of desired size, click it in the **Report** window and stretch the graph until it reaches the proper size.

Now we have the report started. Following further analysis of the results we will add other graphs and tables to it. Next lets view the concentration of the solute within the profile.

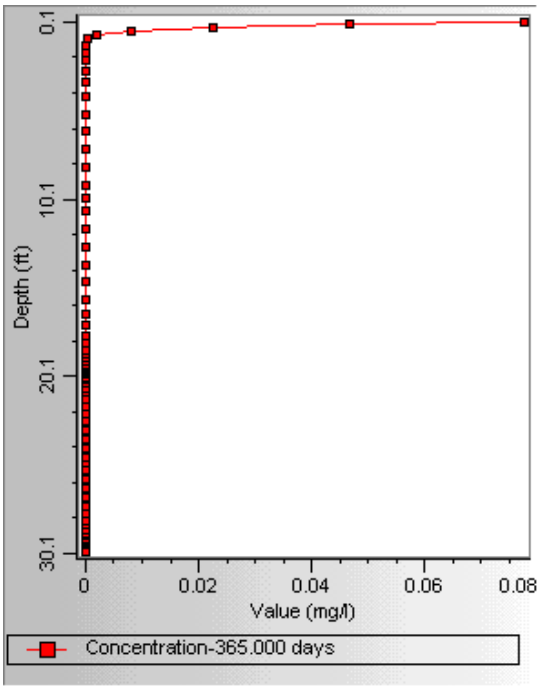
Plotting variable: Solute concentration

☞ **365 days** from the available output times in the Output Tree.

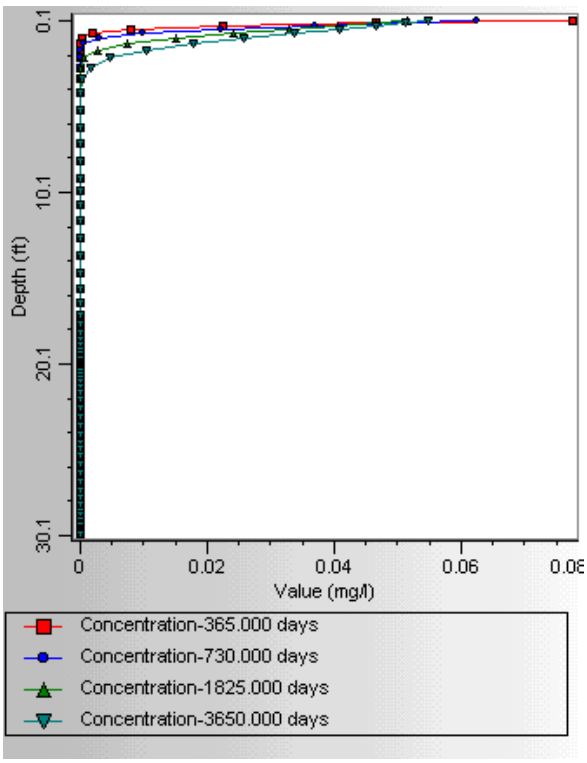
☞ the check box beside **Concentration.** A warning will appear informing you that the graph you just selected is of different type.

☞ **Yes**

A new graph will appear:



Repeat this operation for **730 days** (2 years)**, 1095 days** (3 years), **1825 days** (5 years)**,** and **3650 days** (you would need to open some of the output times in the drop-down list and scroll the bar to reach available outputs)**.** As you select the variable for a specific time, it will appear in the graph. Finally the graph will appear as shown below:



The graph shows, that the concentration of the soil water has changed considerably only in the upper part of the profile to the maximum depth of 5 feet. The concentration of lead in the bottom 5

 Copyright 2000 Waterloo Hydrogeologic, Inc.

feet of the simulated domain, which is the saturated aquifer, still remains equal to 0 after a 10 year period.

To add this graph to the report, in the **Report** window place the cursor to a position below the last graph. Return to the main window of the WHI UnSat Suite and <right click> in the Result View.

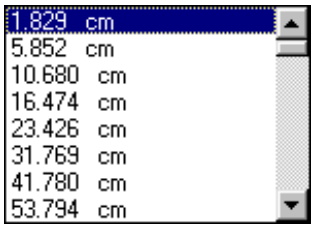
☞ **Insert To Report**. The graph will appear in the report. Stretch the graph until it reaches the proper size.

To learn more about the dynamics of the concentration, lets study the breakthrough curves at specific depths.

Plotting variable: Concentration at Specified Depth (Breakthrough Curves)

☞ **Specified Depth** in the **Select Result Category** list box

☞ the drop-down arrow in the **Select Specified Depth Result at...** list box. The list of available depth will appear:

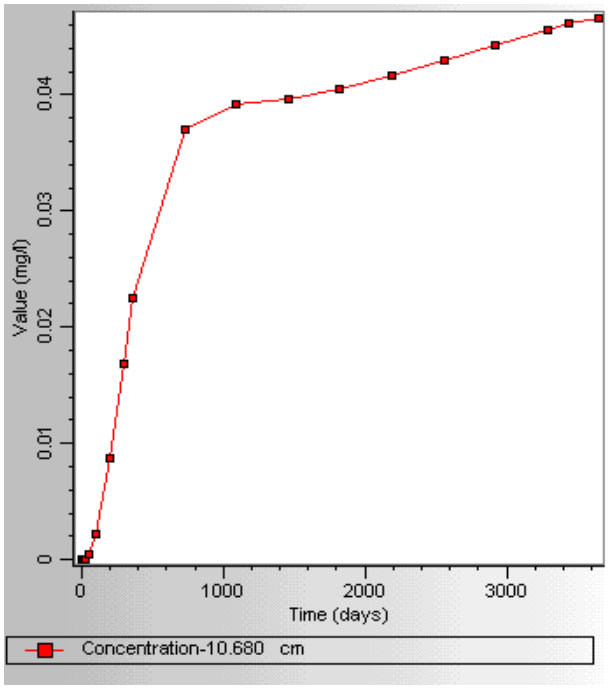


☞ **10.68 cm** (4.2 in) from the list.

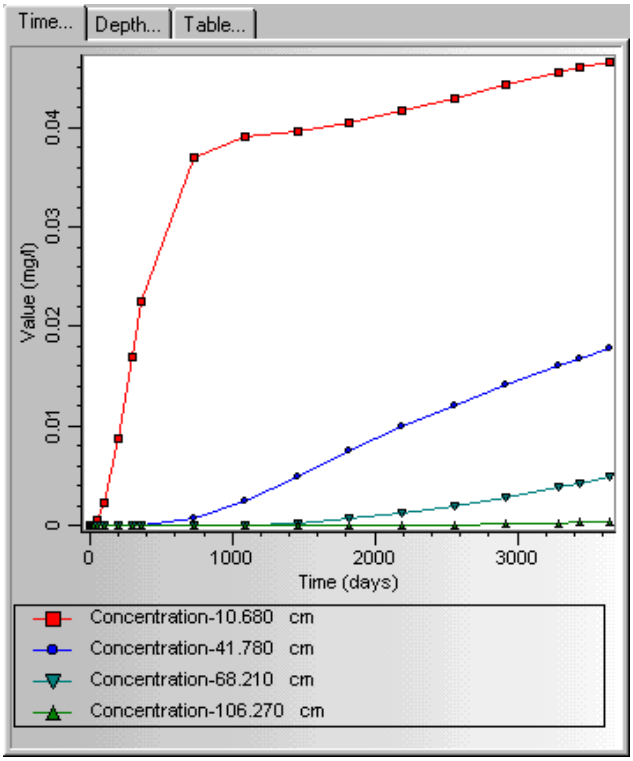
click the  icon to the right of the **Select Specified Depth Result at...** box

☞ the check box beside **Concentration** from the Output Tree

The graph of the variable will appear in the Result View window:



Repeat this operation for **23.426 cm, 41.78 cm, 68.21 cm,** and **106.270 cm.** As you select the variable for a specific depth, it will appear in the graph. Finally the graph will appear as shown below:



This graph shows in detail how the concentration of lead changes in time at specific depths. As one can see, the peak concentration of 0.05 mg/l will occur at the depth 10.68 cm (4.2 in) 10 years after the beginning of simulation. Because of adsorption and hydrodispersion, the values of maximum concentrations reached at different locations decrease abruptly with the growth of depth. Due to adsoprtion, during the whole simulation, the concentration of lead remains much less in the soil water than in the inflow. At a depth of 106.27 cm (41.8 in) the concentration of lead in soil solute remains equal to 0 for the whole 10 years of simulation. It remains unchanged in the greater part of the profile that lays below this point.

Add this graph to your report the same way as it was described above.

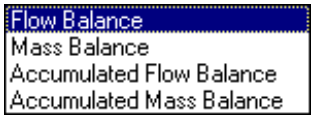
Finally, you will review data presented in table form. You will examine the accumulated flow balance of the model. Because VS2DT is a finite difference model, some combinations of the profile parameters and model settings could impact processes of solution convergence. If convergence criteria are very rough, accumulated computational errors could lead to obtaining non-physical results of simulations. Accumulated flow and mass balances allow examination of computational efficiency and validity of the model you have built.

Viewing table: Accumulated Flow Balance

☞ **Tables** in the **Select Result Category** (upper) list box

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ the drop-down arrow in the **Select Table Result at...** list box. The list of available tables will appear:



☞ **Accumulated Flow Balance**

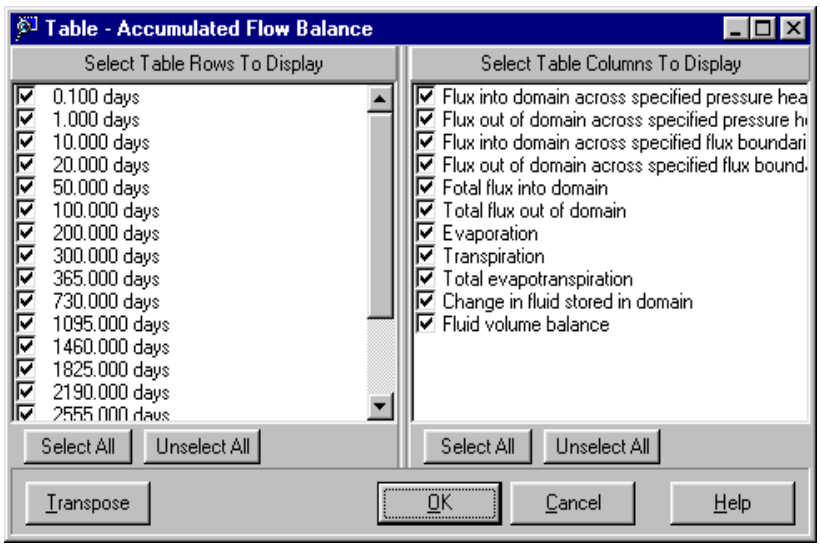
☞ the  icon to the right of the **Select Table Result at...** box. The table name **Accumulated Flow Balance** will appear in the bottom of the Output Tree. For future use, let's place the table results into a new Result Window.

To do this, **<right click>** the name, **Accumulated Flow Balance** in the Output Tree. The following menu will appear:



Choose **New Result Window**.

The new **Result Window 1** will open at the background and the table editing dialog box will appear:



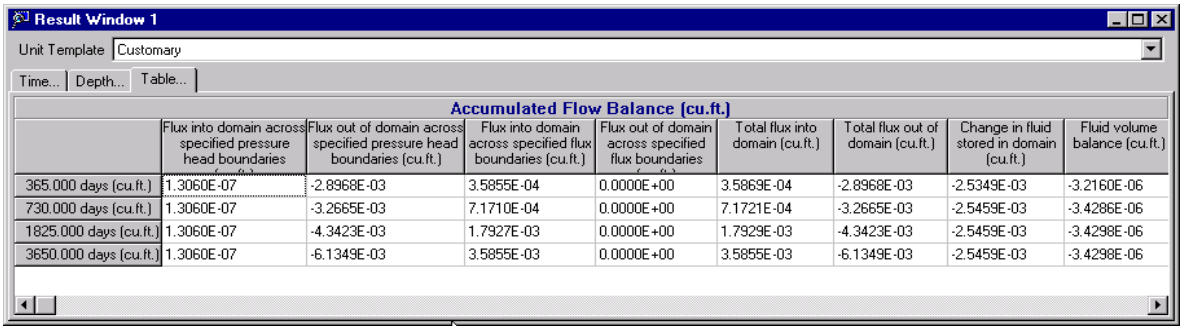
In this dialog box you may select which variables and output times you would like to have in your table.

☞ **Unselect All** button in the left part of the **Table - Accumulated Flow Balance** dialog box. The check-boxes beside output times will become clear.

☞ check-boxes beside **365 days, 730 days, 1825 days, and 3650 days** (you may need to scroll the bar in the **Select Table Rows to Display** window to reach some of the output times).

From the right panel unselect variables **Evaporation, Transpiration** and **Total evapotranspiration** by clicking check-boxes to the right of these variables.

☞ **OK**. The **Accumulated Flow Balance** table will appear in the **Result Window 1**:



Accumulated Flow Balance (cu.ft.)

	Flux into domain across specified pressure head boundaries	Flux out of domain across specified pressure head boundaries (cu.ft.)	Flux into domain across specified flux boundaries (cu.ft.)	Flux out of domain across specified flux boundaries	Total flux into domain (cu.ft.)	Total flux out of domain (cu.ft.)	Change in fluid stored in domain (cu.ft.)	Fluid volume balance (cu.ft.)
365.000 days (cu.ft.)	1.3060E-07	-2.8968E-03	3.5855E-04	0.0000E+00	3.5869E-04	-2.8968E-03	-2.5349E-03	-3.2160E-06
730.000 days (cu.ft.)	1.3060E-07	-3.2665E-03	7.1710E-04	0.0000E+00	7.1721E-04	-3.2665E-03	-2.5459E-03	-3.4286E-06
1825.000 days (cu.ft.)	1.3060E-07	-4.3423E-03	1.7927E-03	0.0000E+00	1.7929E-03	-4.3423E-03	-2.5459E-03	-3.4298E-06
3650.000 days (cu.ft.)	1.3060E-07	-6.1349E-03	3.5855E-03	0.0000E+00	3.5855E-03	-6.1349E-03	-2.5459E-03	-3.4298E-06

This table presents a detailed description of fluxes coming in/out of the domain through different types of boundaries as well as the change in the amount of stored water. However, for quality assurance of the model performance, you should compare the **Total Flux into domain**, **Total Flux out of domain** and **Change in liquid stored in domain** with the **Fluid volume balance.** From the table you may see the values of these variables (per unit area of 1 sq.cm) at the end of simulation:

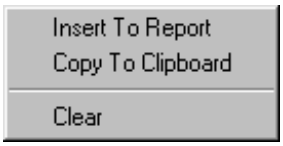
- **Total Flux into domain** **3.585 E-3 cu.ft.**
- **Total Flux out of domain** **-6.135 E-3 cu.ft.**
- **Change in liquid stored in domain** **-2.546 E-3 cu.ft.**
- **Fluid volume balance** **-3.4 E-6 cu.ft.**

From these numbers you can see that much more liquid left the domain than entered it. As we know from the moisture content plots, this difference was caused by initial drying of the profile after the asphalt was placed at the top. The comparison of the **Fluid volume balance**, which is the balance discrepancy caused by computational errors, with the **Total fluxes into/out of domain** shows that the later variable is 3 orders of magnitude higher. This is a good ratio meaning that the computational error constitutes a portion of a percent of total fluxes and simulation results are reliable. In this lab exercise you reviewed the flow balance after some results were discussed. However, while running a practical case, we advise you to check the flow and mass balances of the model prior to making any conclusions about simulation results.

Next you will add this table to your project report. This procedure is pretty much the same as adding a graph to a report.

To add a table to the report:

Scroll the bar in the **Report** window to the bottom and place the cursor to a position below the last graph or table in the report. Return to the main window of the WHI UnSat Suite and <right click> in the Result View. The following menu will appear:



☞ **Insert To Report**. The table will appear in the report.

Now you may print your report. In the main menu of the **Report** window:

☞ **File/Print**. The report will be printed by your default printer.

This concludes the VS2DT demonstration exercise. Close the WHI UnSat Suite by clicking **X** in the top right corner.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

We hope these instructions have provided you with a good understanding of the capabilities and tools the WHI UnSat Suite offers. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us at (519) 746-1798 or visit our webpage, www.flowpath.com.

WHI UnSat Suite Tutorial: VLEACH

Modeling Fate of Volatile Organic Contaminants in the Unsaturated Zone

WHI UnSat Suite for Windows 95/98/NT/2000

The WHI UnSat Suite combines three popular unsaturated zone models in a powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. These models are:

SESOIL - a popular US EPA model which is capable of simultaneously modeling water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess contaminant fate and transport for regulatory requirements.

PESTAN (PESTicide **AN**alitycal**)** - a model for assessments of groundwater contamination by pesticides used in agricultural practice.

VS2DT (Variably **S**aturated **2D** flow and **T**ransport**)** - a finite difference numerical model for determining the fate of agricultural chemicals, landfill leachate, and accidental chemical spills in the unsaturated zone.

VLEACH (**V**adose Zone **LEACH**ing model) - a model for evaluating the fate of volatile organic contaminants in the unsaturated zone.

All these popular models are seamlessly integrated with the WHI UnSat Suite and each one is compiled and optimized to run as a 32-Bit, native Windows application.

The WHI UnSat Suite shares the majority of its features with the other WHI software product Visual HELP which, besides its direct function, landfill designing, can be used as an efficient tool for the assessment of soil surface water balance. The two products are connected functionally and logically and in many cases the joint use of two products in consulting projects is very beneficial. Both products are joined together in the software package called WHI UnSat Suite Plus. To learn more about Visual HELP, review the demo exercise for this product.

Learning Objectives

In this demonstration exercise, you will use VLEACH as a tool for simulating the transport of Trichloroethylene in the unsaturated zone and prepare a project report.

The instructions for this exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

Background Information for VLEACH Exercise

The ACME Manufacturing Company has recently reported spilling a significant quantity of hazardous waste materials at their plant. A detailed subsurface environmental investigation identified the hazardous materials as primarily consisting of Trichloroethylene (TCE), a popular industrial degreasing agent. The subsurface characterization indicated the hazardous material has

been spilled over an area of 0.7 acres and has percolated into the unsaturated zone. A soil sampling program indicated that the concentration of TCE is 100 µg/kg in the upper 0.3 m of soil and 40 µg/kg in the interval 0.3 m - 1.0 m .

In this exercise you will use UnSat Suite to simulate the transport and fate of TCE in solid, liquid and gas phases through the unsaturated zone and the leaching of this contaminant to the groundwater. According to the WHI Unsaturated Zone modeling technology, the HELP and the VLEACH models are used in conjunction to assess the concentration of TCE in soil water reaching the groundwater table at the site. HELP is used to provide high accuracy in the assessment of the average groundwater recharge. VLEACH is used to simulate the unsaturated flow, transport and transformation of TCE in the unsaturated zone for a 50 year period. The Visual HELP simulation showed that the average infiltration rate into the soil profile is 9 in/year.

The following are the parameters of the profile to be simulated with the VLEACH model:

The profile consists of one layer which has a thickness of 30 feet and is composed of silt. The parameters of the silt are:

- dry bulk density 1.5 g/cm3
- effective porosity 0.47
- moisture content 0.35
- fraction organic content 0.1

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type-	Type in the given word or value
select-	Click the left mouse button where indicated
α -	Press the **<Tab>** key
↵ -	Press the **<Enter>** key
☞ -	Click the left mouse button where indicated
☞ ☞ -	Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

Starting WHI UnSat Suite

To start WHI UnSat Suite, you must have it installed on your hard disk. Once installed, click Windows **Start** button, choose **Programs**, navigate to the folder that contains WHI UnSat Suite, and click **UnSat Suite**.

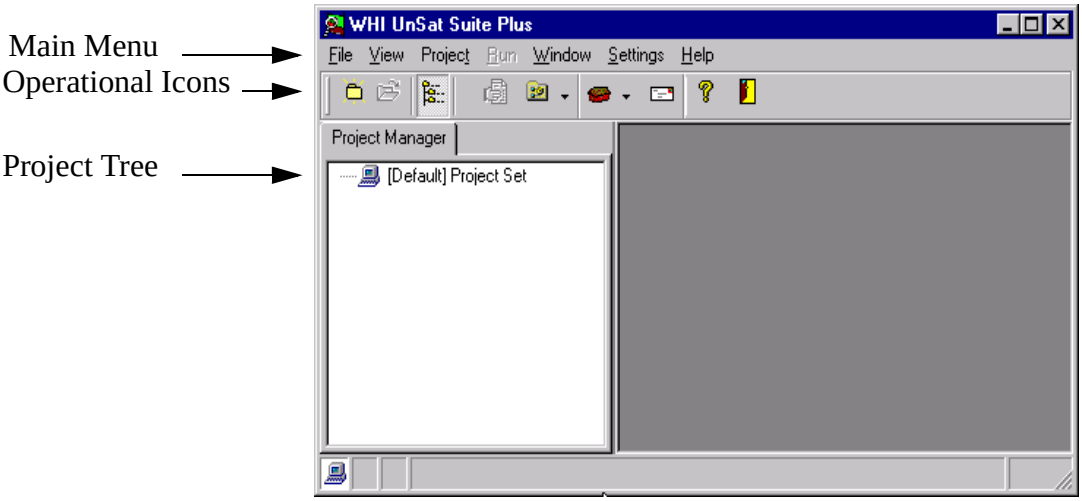
The installation program will also place an icon on your desktop, which you can also double click to start the WHI UnSat Suite exercise.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

An introductory screen will be displayed when you open WHI UnSat Suite



After the program is loaded, the main WHI UnSat Suite window will appear on your display:



The WHI UnSat Suite interface has been designed to help you navigate through the program with ease. The main menu is located at the top with operational icons located directly below the main menu items to allow quick access to many options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

Module I: Creating a New Project

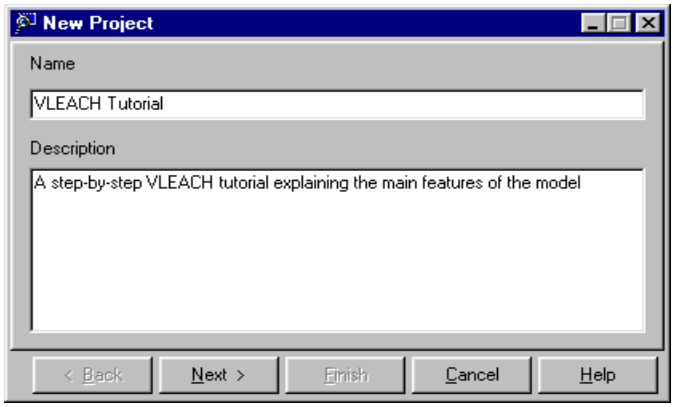
To create a new project, go the the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new WHI UnSat Suite project.

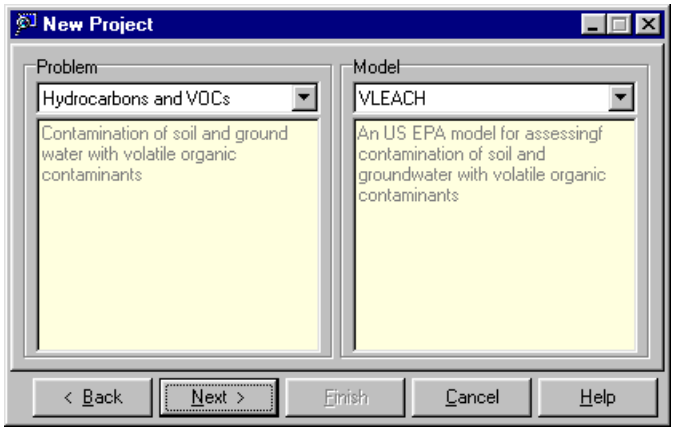
In the **Name** box type: **VLEACH Tutorial**.

In the Description box type: **Step-by-step VLEACH tutorial explaining the main features of the model.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, to view problems which can be examined with the WHI UnSat Suite Plus. Also notice that WHI UnSat Suite automatically recognizes the appropriate model that should be used with the selected Problem type.



☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The WHI UnSat Suite Unit Converter was developed to maximize the flexibility for data input and output unit conversion. With the WHI UnSat Suite you may create your unit templates to fit the format of your data.

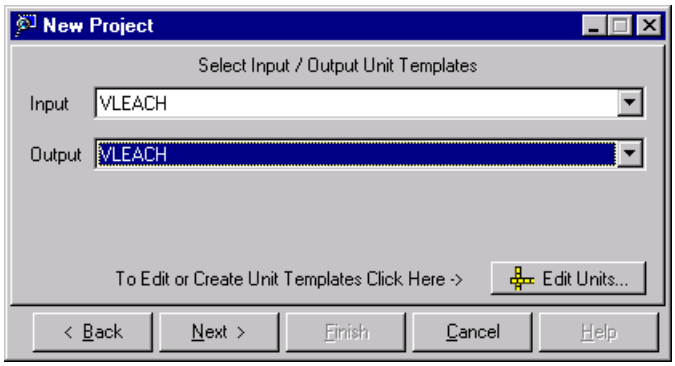
For this exercise we will specify that all input and output be set in accordance with the **VLEACH** unit template. This unit template mainly uses units of the original VLEACH model.

From the **Input** drop-down list,

☞ **VLEACH**

From the **Output** drop-down list

☞ **VLEACH**



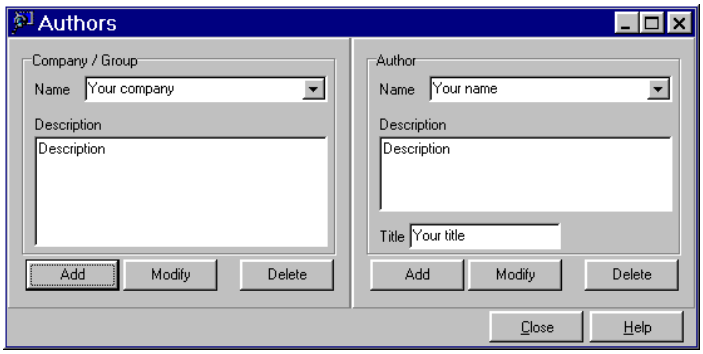
☞ **[Next]**

Assigning Authors and Clients

The WHI UnSat Suite allows you to create and maintain your own database of project Authors and Clients for use in future projects. The information which you input/specify here will appear automatically in your project report. To add a new author to the database

☞ **[Authors]**

The **Authors** dialog box will appear:



Each author is associated with a company.

Company/Group

In this area, you can add, modify, or delete companies and groups. To add your company to the database, in the **Name** box:

Type your company name

In the **Description** box:

type your address and phone number.

☞ **[Add]**

Author

In this area, you can add, modify, or delete authors from companies. To add a new author, input your information in the **Name** box, **Title** box, and **Description** box.

☞ **[Add]**

☞ **[Close]** to return to the **New Project** dialog box.

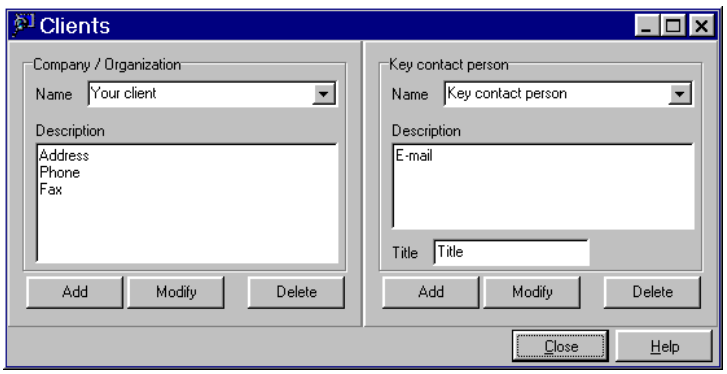
While back in the **New Project** dialog box, click the drop-down arrow and select your name from the list of the Authors.

Clients

To add a new client to the database:

☞ **[Clients]**

The **Clients** dialog box will appear



Each contact is associated with a company or organization.

Company/Organization

In this section you can add, modify, or delete companies and organizations. To add a new company, edit the information in the **Name** box and **Description** box, and click **[Add]**. To update an old company with current information, edit the information, and click **[Modify]**. To delete a company, select the company from the list, and click **[Delete]**.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

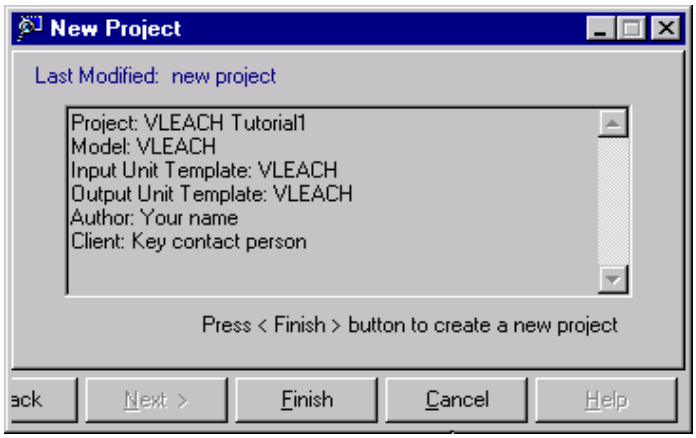
Key Contact Person

In this area you can add, modify, or delete contacts from companies. To add a new contact, edit the information in the **Name** box, **Title** box, and **Description** box, and click **[Add]**. To update the information about a client, edit the information, and click **[Modify]**. To delete a client, select the client from the list, and click **[Delete]**.

☞ **[Close]** to return to the **New Project** dialog box. Select an existing client from the list, or create a new client by clicking **[Clients].**

☞ **[Next]**

All the project information will be presented, as seen in the figure below:



☞ **[Finish]** to save the information as a project in the database and add it to the Project Tree.

After the project settings are specified, the **New Profile Wizard** will open.

Creating a Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:



In this dialog box you may select an already existing profile template or you can build a new profile 'from scratch'. In this demonstration exercise you will build a new profile.

☞ **[create new profile]**

☞ **[Next]**

The dialog box with default profile parameters will appear. In this dialog box we may assign the parameters of the simulated profile.

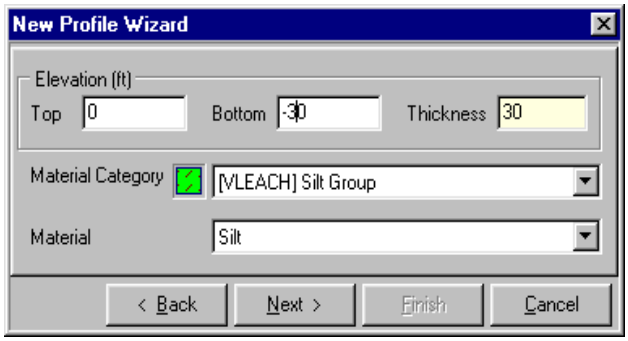
☞ in the **Bottom** text box and input **-30** for the elevation of profile bottom. The value of **Thickness** will change accordingly.

☞ the drop-down arrow in the **Material Category** drop-down list box.

Select **[VLEACH] Silt group**. The default material **Silt** will appear in the **Material** drop-down list box.

Note: You may add as many materials to specific Material Category group as you need, once inside the model, in Material Designer.

As a result of editing, the **New Profile Wizard** window will appear as shown below:



☞ **[Next]**

All the profile information will be presented:



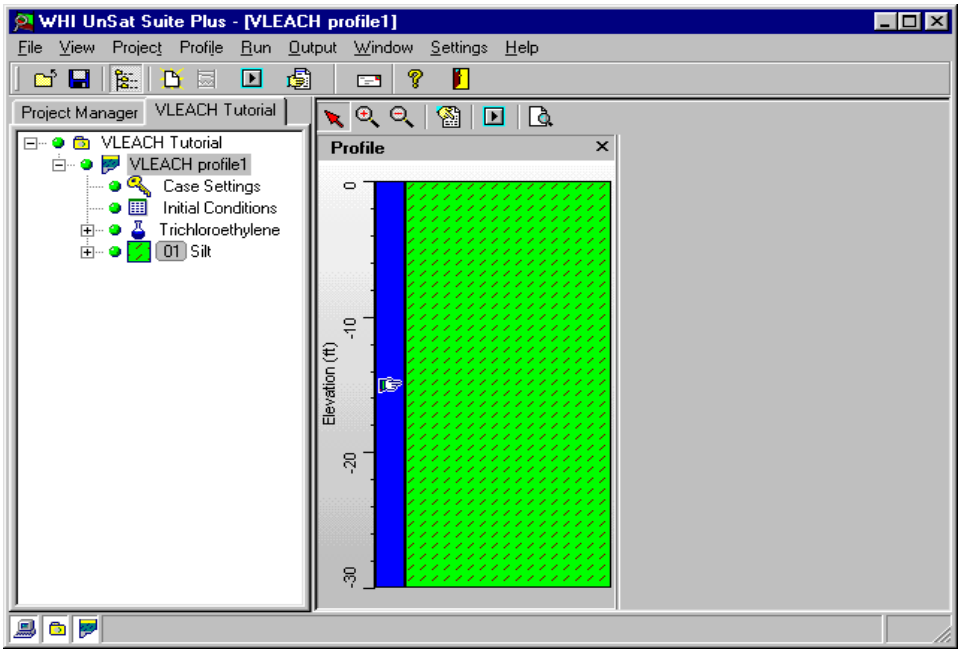
☞ **[Finish]** to load the profile.

The new project details will be added to the Project Tree View, the project will open and a profile picture will appear in the Profile View. This concludes creating a new project using the WHI UnSat Suite. We will now continue with designing the profile and setting specific simulation details.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Module II: Designing a Profile and Setting Case Specifications

The following picture will appear on your display:



In the Profile View, located in the center of the WHI UnSat Suite window, you will see the loaded profile with a depth scale at the left.

A description of a layer will appear when the mouse pointer is pointed at the layer.

The Project Tree View in the left part of the WHI UnSat Suite window provides access to parameters for your simulation scenario. The icon and the name of the profile layer **Silt** are located in the bottom of the list.

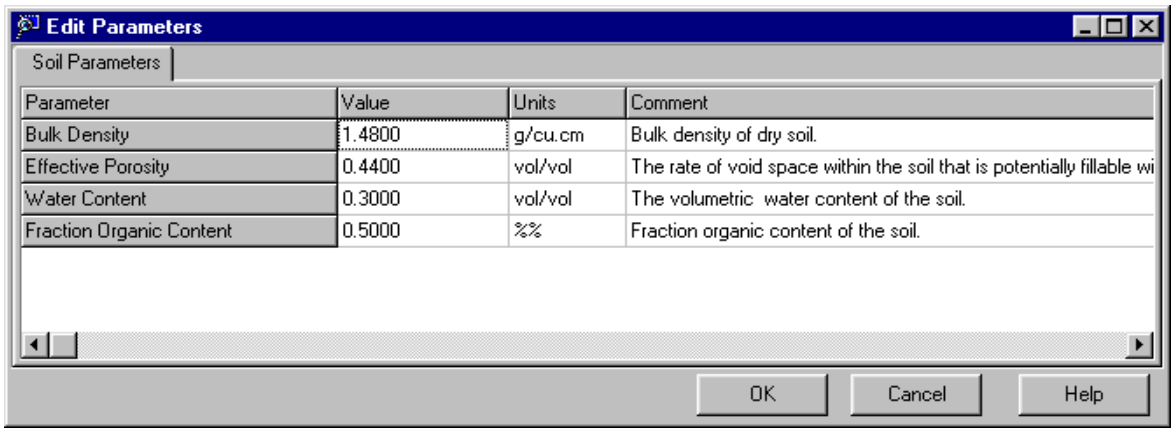
In this part of the exercise you will:

- learn how to edit the parameters of a layer
- learn how to set boundary conditions and initial conditions for the profile

Editing Layer Parameters

☞ **'+'** sign in the box to the left of the **Silt** icon.

☞ ☞ **Soil Parameters**. The **Edit Parameters** dialog box will appear.



In this dialog box, you can view and edit general parameters of the layer.

☞ **Value** field beside the **Bulk Density**

Type: **1.5**

☞ **Value** field beside the **Effective Porosity**

Type: **0.47**

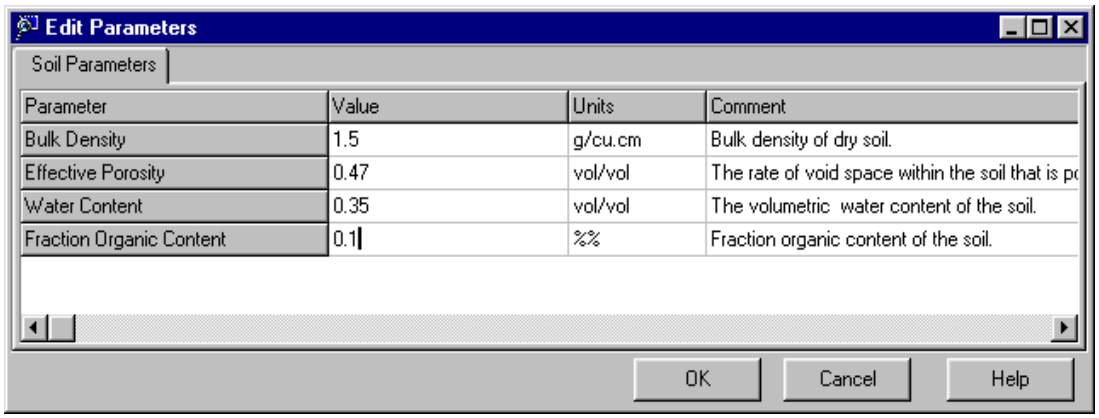
☞ **Value** field beside the **Water Content**

Type: **0.35**

☞ **Value** field beside the **Fraction Organic Content**

Type: **0.1**

The new **Edit Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-],** beside **Silt** in the Project Tree.

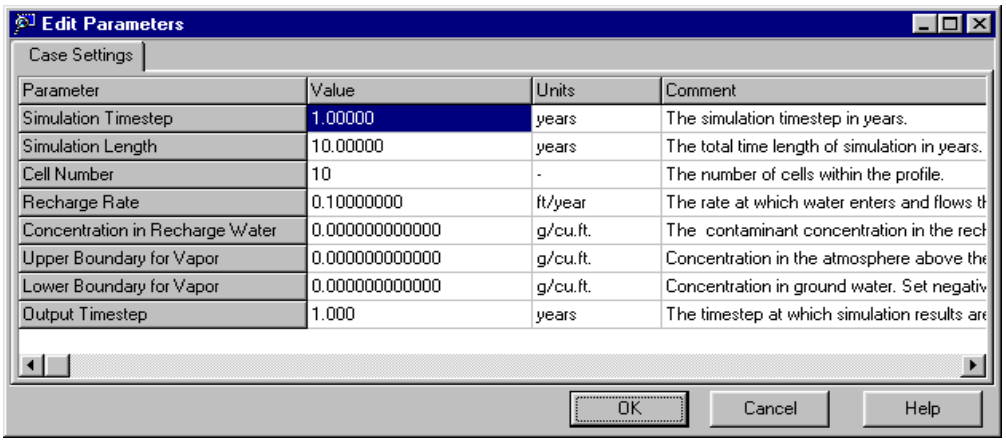
With this you have finished editing the layer properties. Next you will specify boundary and initial conditions and set computational parameters for the project case.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Case Settings

Constructing the unique simulation case requires you to input several simulation parameters and initial conditions of the profile. These parameters may be edited in the project parameter group labeled **Case Settings**.

☞☞ on **Case Settings** in the **Project Tree View**. The following dialog box will appear:



Leave the parameter **Simulation Timestep** as it is.

☞ **Value** field beside the **Simulation Length**

Type: **50**

Next you have to specify the number of cells in the vertical direction. VLEACH uses cells of constant length. The initial concentration of a pollutant is assumed to be uniform within a cell. As stated previously, the highest concentration of TCE is within the upper foot of soil. For the purpose of presentation and to input the initial condition accurately, lets set the length of a cell to 1 foot. The number of cells within the 30 foot profile would be 30.

☞ **Value** field beside the **Cell Number**

Type: **30**

Next you have to specify the recharge rate. To do this you have to first change the units for this parameter to bring them in accordance with your input data.

☞ **Units** field beside the **Recharge Rate**

☞ the drop-down arrow

☞ **in/year** from the drop-down list

☞ **Value** field beside the **Recharge Rate**

Type: **9**

Leave the parameters **Concentration in Recharge Water** and **Upper Boundary for Vapour** as they are.

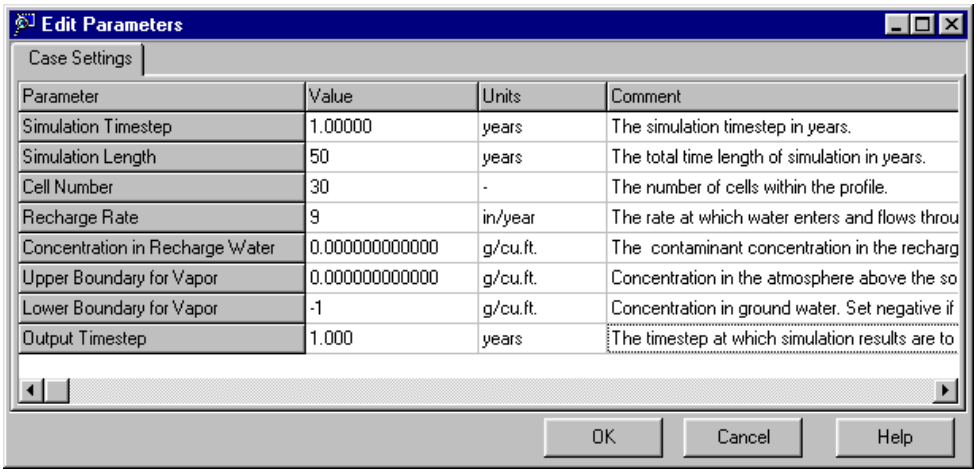
☞ **Value** field beside the **Lower Boundary for Vapour**

Type: **-1**

The negative value for the parameter **Lower Boundary for Vapour** means that no diffusion from the gas phase to the ground water is to be simulated.

Leave parameter **Output Timestep** as it is.

Finally, the Case Settings dialog window will look like the following

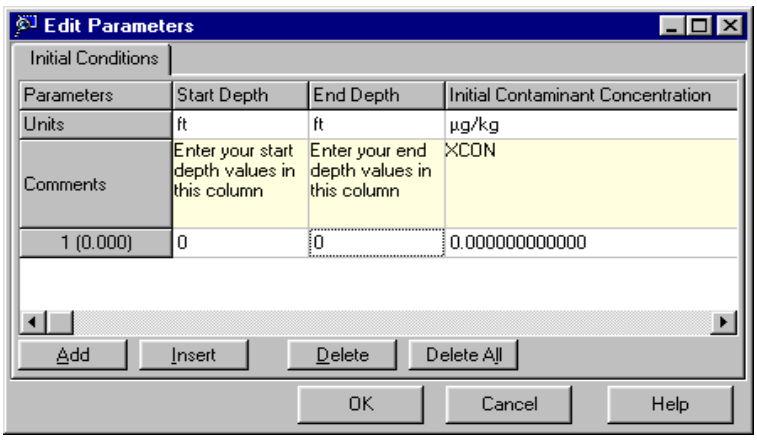


☞ **[OK]** to save changes and close the table.

Initial Conditions

The initial chemical distribution has to be specified for the soil profile.

☞☞ **Initial Conditions** in the **Project Tree View.** The following dialog box will appear:



☞ in the **End Depth** field for the first interval (first line)

Type: **1**.

☞ in the **Initial Contaminant Concentration** field

Type: **100**

☞ **[Add],** to add another interval. When you click **Add** for the first time, the value of the Initial Contaminant Concentration from the first line will be copied.

☞ in the **End Depth** field for the second interval (second line)

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Type: **3**.

☞ in the **Initial Contaminant Concentration** field

Type: **40**

☞ **[Add]**

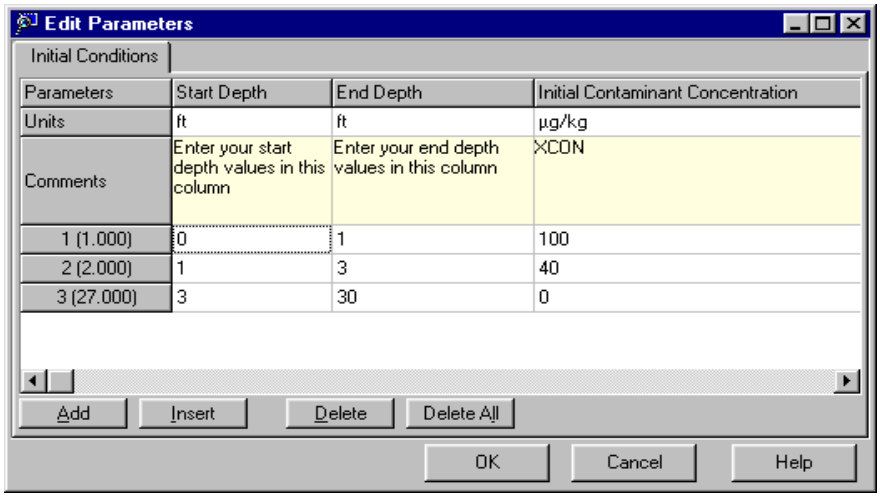
☞ in the **End Depth** field for the third interval (third line)

Type: **30**.

☞ in the **Initial Contaminant Concentration** field

Type: **0**

As a result, the **Initial Conditions** dialog box will appear as below:



☞ **OK** to confirm and save entered initial concentrations.

This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **VLEACH tutorial 1**, which was specifically developed for you and contains all changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

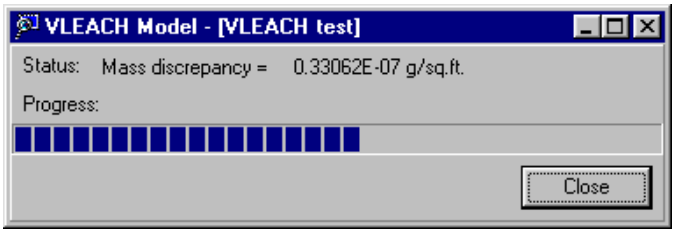
☞ ☞ project **VLEACH tutorial 1.**

Module III: Running the Model, Viewing Output, and Reporting

Running the VLEACH Model

To run the model for a single profile, click the  **Run** icon above the Profile View.

A progress bar will appear to indicate the computation progress, as shown below:



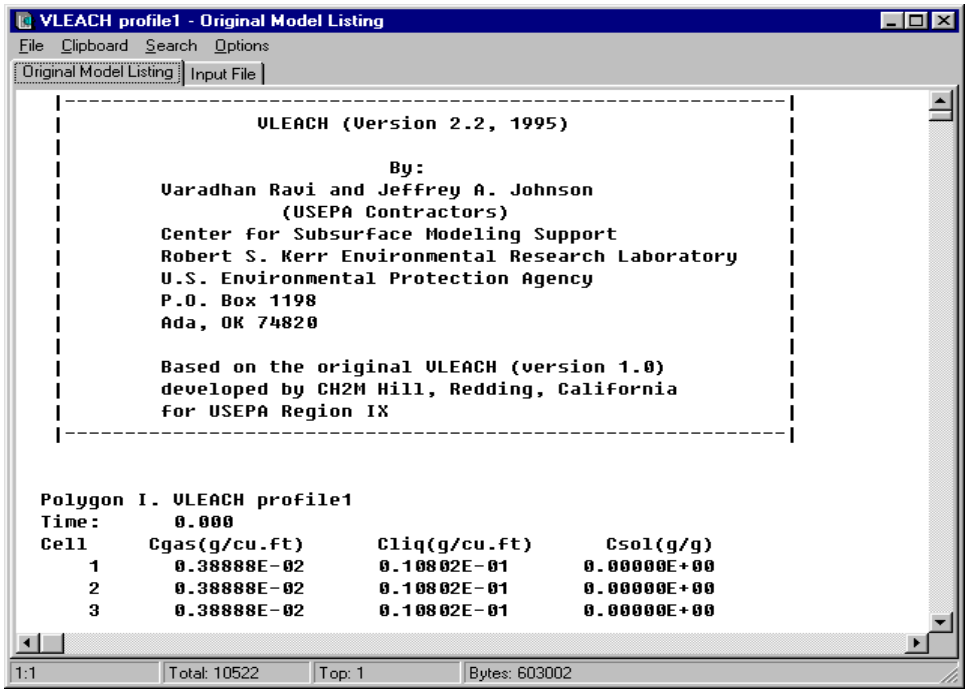
Viewing the Original VLEACH Output File

To view the original VLEACH output:

☞ **Output** in the main menu, and then

☞ **View Original Listing**

The following window will appear:



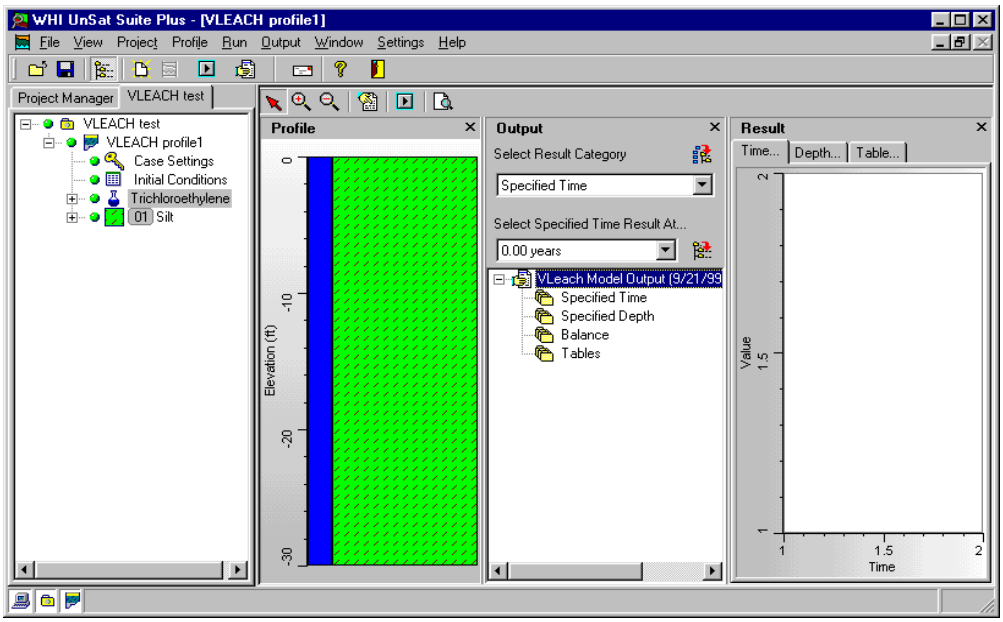
 Copyright 2000 Waterloo Hydrogeologic, Inc.

Here you can view the original listing, find specific expressions, mark and print parts or make a complete printout of the model results.

☞ the **X** in the top right corner to close the original listing.

Viewing the Output Graphs

After the model has successfully ran, the **Output** View and **Result** View window will open and the UnSat Suite window will appear as below:



To enlarge the viewing area click the icon  to close the Project Tree View, or click the '**X**' in the Profile View to close it.

All settings for the output presentation can be edited in the **Output** View.

To select the output category:

☞ the drop-down arrow in the **Select Result Category** list box. The possible result groups will appear in the list box presented below:



The **Result Categories** present the following:

- **Specified Time** - plots of contaminant concentration in soil gas phase, soil water and in soil solid phase (adsorbed) versus depth for specified observation times.
- **Specified Depth** - plots of contaminant concentration in soil gas phase, soil water and in soil solid phase (adsorbed) versus time for specified depths (cell centers).
- **Balance** - plots of the amount of contaminant within the unsaturated profile (per unit area) in soil gas phase, soil water and in soil solid phase (adsorbed) versus time.
- **Tables: Balance** - tabular presentation of the contaminant balance (see above)
- **Tables: Accumulated Balance** - tables showing the accumulated volumes of contaminant leaving the profile to atmosphere and ground water by diffusion and advection.

Viewing Depth Dependent Data for Specified Time

☞ **Specified Time** in the **Select Result Category**

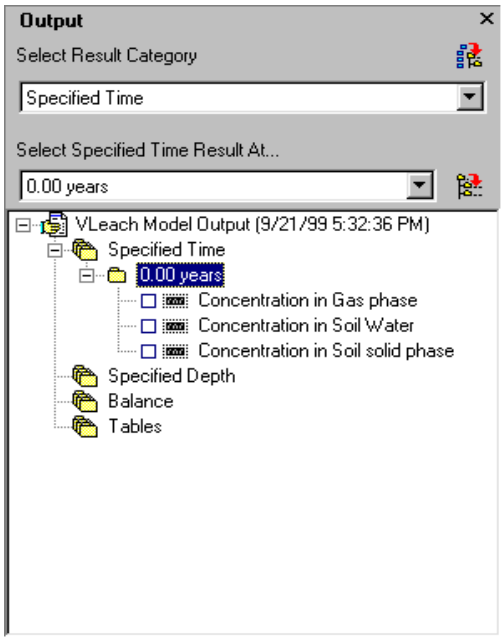
☞ the drop-down arrow in the **Select Specified Time Result at...** list box. The list of observation times will appear:



☞ **0.00 years** which is the first available output and shows initial conditions

To view all results available for this specific time, click the  icon to the right of the **Select Specified Time Result at...** box.

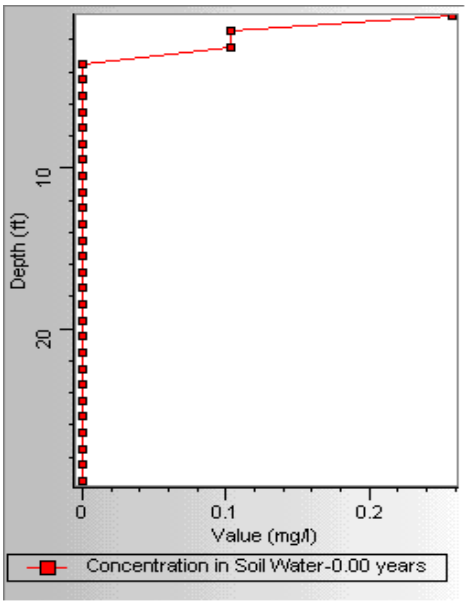
The list of available results will open in the Result Tree:



☞ the check box beside **Concentration in Soil Water.**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

A graph of the variable will appear in the Result View window showing the concentration of contaminant in soil water after the equilibria between the soil gas, liquid and solid phases was calculated using the initial concentration in the soil and partitioning coefficients.



Lets watch how the concentration in the soil water changes in time.

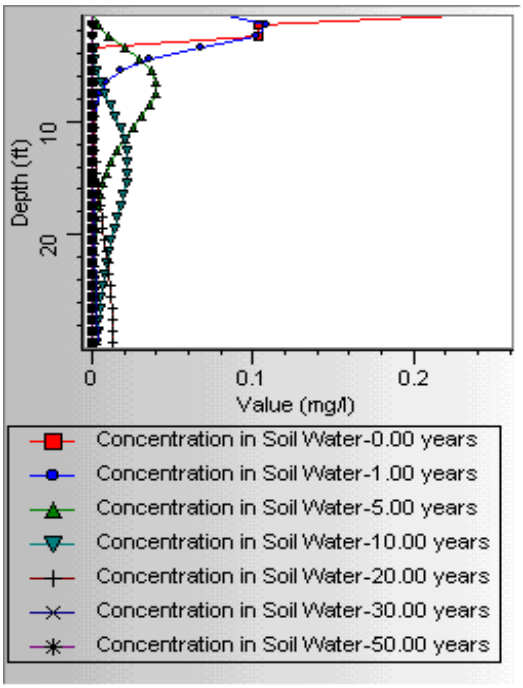
☞ the drop-down arrow in the **Select Specified Time Result at...** list box.

☞ **1.00 year**

click the  icon to the right of the **Select Specified Time Result at...** box to open the list of variables.

☞ the check box beside **Concentration in Soil Water** from the Output Tree

Repeat this operation for **2, 5, 10, 30 and 50 years** (you may need to scroll the list of observation times to reach some of them)**.** As you select the variable for a specific time, it will appear in the graph. Finally, the graph will appear as below:



The graph shows how the wave of contaminant moved down the profile. It can also be noted that the maximum concentration decreases in time and gets close to 0 between 30 and 50 years.

This, in brief, shows you some of the options for viewing depth dependent data for the observation times that you specify. In the same way you may view data for **Concentration in Gas phase** describing the intensity of contaminant volatilization and **Concentration in Soil Solid Phase** describing the intensity of contaminant adsorption.

At this time you may want to start creating your project report and add the above graph to it. Next you will learn how to create a WHI UnSat Suite report.

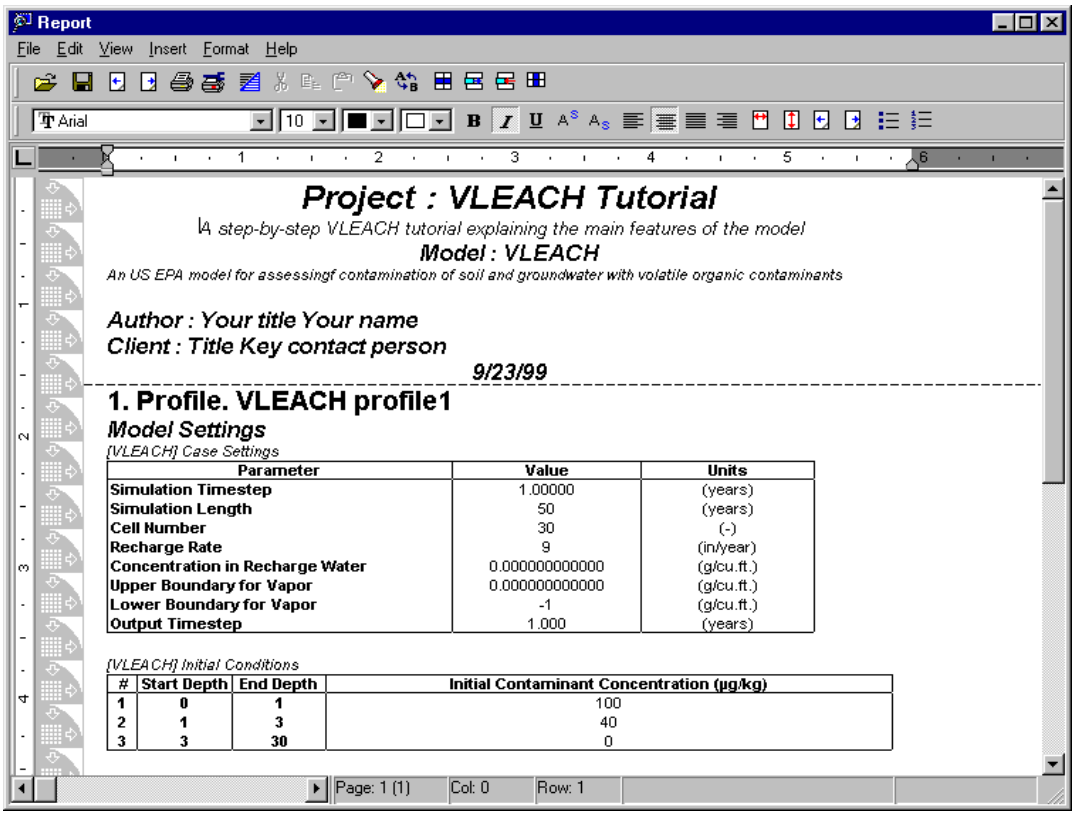
Creating a Report

To present the results of your VLEACH simulation to your clients, you may use the UnSat Suite Report Generator.

To create a report and add the project input data, click the icon  from the Operational Icons tool bar.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The report will appear in a separate window:

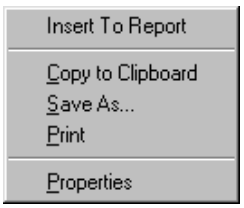


In this window you may edit the report, input your own text and add any types of graphics or table outputs produced by WHI UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph to the report:

Scroll the bar in the **Report** window to the bottom and place the cursor to a position below the last table in the report. Return to the main window of WHI UnSat Suite and <right click> on the graph in the Result View. The following menu will appear:



☞ **Insert To Report**. The graph or table will appear in the report.

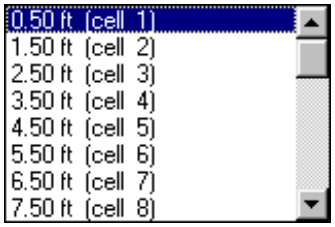
The graph may appear smaller than the original. To view the graph of desired size, click in the **Report** window and stretch the graph until it reaches the proper size.

Now we have the report started. Further analysis of the results will produce additional tables and graphs that can be added at a later date.

Viewing Time Dependent Data for Specified Depth (Breakthrough Curves)

☞ **Select Result Category** list box and select **Specified Depth**

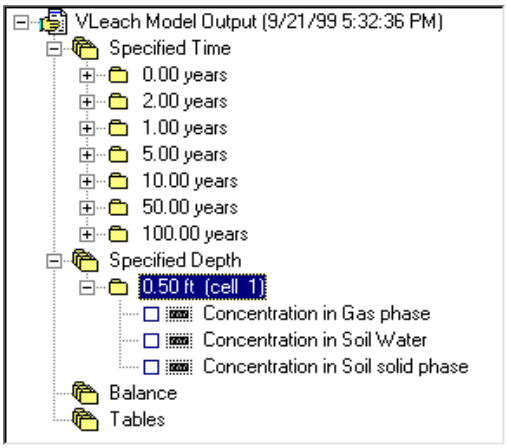
☞ the drop-down arrow in the **Select Specified Depth Result at...** list box. The list of observation times will appear:



☞ **0.50 ft** which is the center of the first cell

To view all results available for this specific depth, click the  icon to the right of the **Select Specified Depth Result at...** box.

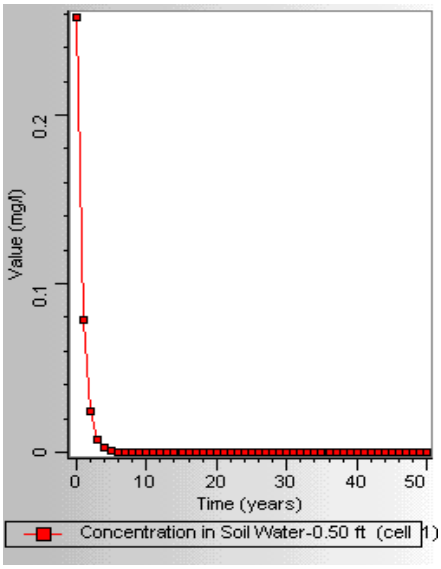
The list of available results will open in the Result Tree:



☞ the check box beside **Concentration in Soil Water.**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

A graph of the variable will appear in the Result View window showing the concentration of contaminant in soil solute at the depth of 0.5 feet:



Lets watch how the concentration in the soil water changes at different depths.

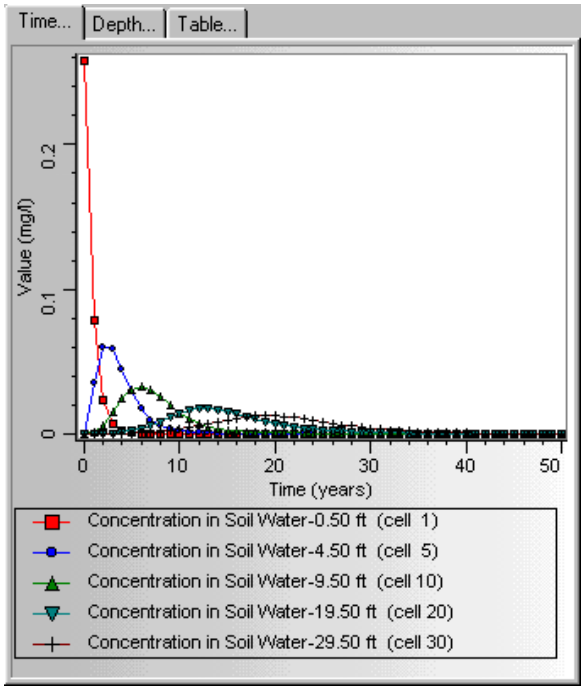
☞ the drop-down arrow in the **Select Specified Depth Result at...** list box.

☞ **4.5 ft (cell 5)**

click the  icon to the right of the **Select Specified Time Result at...** box to open the list of variables.

☞ the check box beside **Concentration in Soil Water** from the Output Tree

Repeat this operation for depths **9.5 (cell 10), 19.5 (cell 20)** and **29.5 (cell 30)** (you may need to scroll the list of observation depths to reach some of them). As you select the variable for specific depth, it will appear in the graph. Finally, the graph will appear as below:



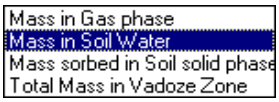
The graph shows that the peak of contaminant concentration will pass the bottom of the profile 20 years after the start of the simulation and the concentration at the bottom of the profile will become zero 36 years after the start of the simulation. It can also be noted that the plume of TCE stretches in length and the contaminant concentration decreases down the profile.

This, in brief, shows you some of the options available for viewing time dependent data for the observation depths that you specify. In the same way, you may view data for **Concentration in Gas Phase** and **Concentration in Soil Solid Phase.**

Viewing Contaminant Balance

☞ **Select Result Category** list box and select **Balance**

☞ the drop-down arrow in the **Select Balance Result at...** list box. The list of available balance variables will appear:



☞ **Mass in Soil Water**

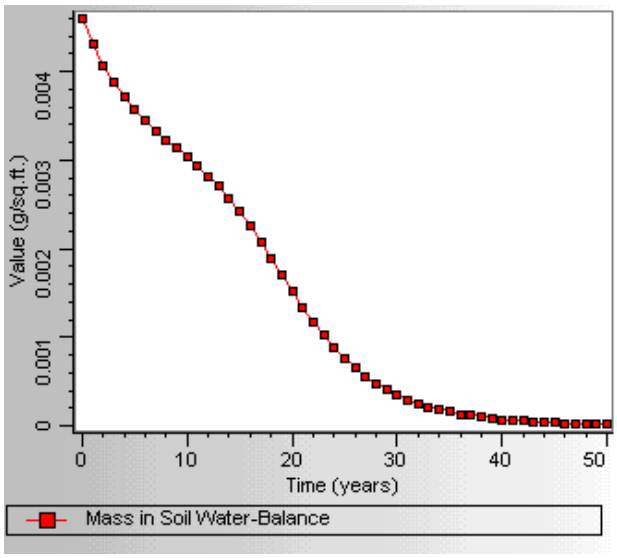
☞ the  icon to the right of the **Select Balance Result at...** box.

The variable with a check box will appear at the bottom of the Result Tree:

☞ the check box beside **Mass in Soil Water.** A warning will appear to indicate that the new variable has different size or units compared to the previously presented variable.

☞ **Yes**

A graph of the variable will appear in the Result View window showing the plot of the amount of contaminant within the unsaturated profile (per unit area) in the soil water versus time.



The graph shows that TCE will be totally removed from the soil water phase after 47 years.

☞ **Mass in Gas Phase** from the **Select Balance Result at...** list box

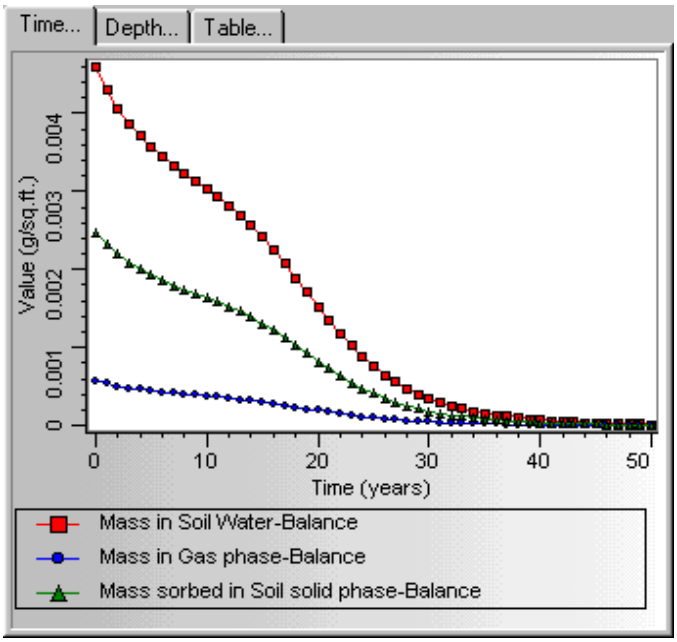
☞ the  icon to the right of the **Select Balance Result at...** box.

The variable with a check box will appear in the Result Tree:

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ the check box beside **Mass in Gas Phase**

Repeat these operations for the variable **Mass sorbed in Soil solid phase.** Each time you select a specific variable, it will appear in the graph. Eventually, the Output window will appear as below:



This graph shows you the changes in the amount of TCE in three phases. However, for hydrogeologists and environmental engineers data on groundwater impact of a contaminant is the most important. These data are presented in table form.

Viewing Tables for Accumulated Contaminant Balance

☞ **Select Result Category** list box and select **Tables**

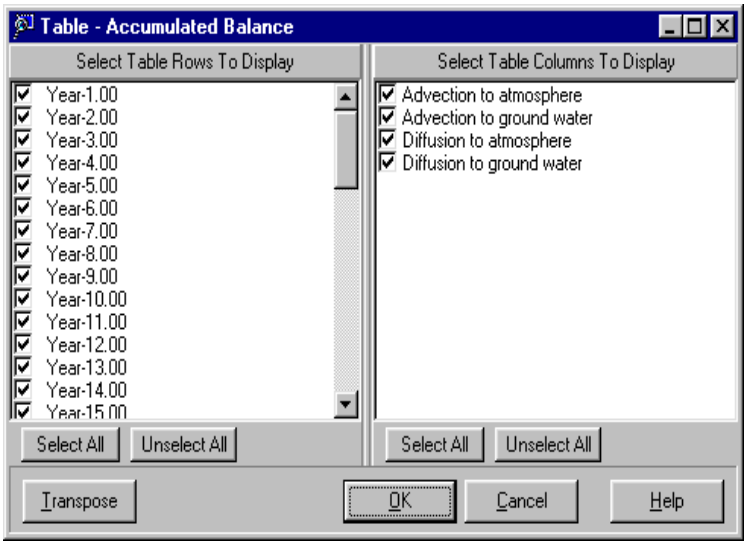
☞ the drop-down arrow in the **Select Tables Result at...** list box. The list of available balance variables includes **Balance** and **Accumulated Balance**. Table **Balance** contains the same data as presented before in graphical form. Table **Accumulated Balance** shows the accumulated volumes of contaminant leaving the profile to the atmosphere and groundwater, the latter of which is the impact of the contaminant on the groundwater.

☞ **Accumulated Balance**

☞ the  icon to the right of the **Select Tables Result at...** box.

The variable with a check box will appear at the bottom of the Result Tree:

☞ the check box beside **Accumulated Balance**. A **Table - Accumulated Balance** dialog box will appear:



In this dialog box you may select which variables and output times you would like to have in your table.

☞ **Unselect All** button in the left part of the **Table - Accumulated Balance** dialog box. The check-boxes beside output times will become clear.

☞ check-boxes beside **Year -1.00, Year -5.00, Year -10.00, Year -20.00, Year -30.00, Year -40.00, Year -50.00** (you may need to scroll the bar in the **Select Table Rows to Display** window to reach some of the output times). Leave all variables in the right pane checked.

☞ ☞. The **Accumulated Balance** table will appear in the **Output** window:

Accumulated Balance (g/sq.ft.)

	Advection to atmosphere	Advection to ground water (g/sq.ft.)	Diffusion to atmosphere (g/sq.ft.)	Diffusion to ground water (g/sq.ft.)
Year-1.00 (g/sq.ft.)	0.0000E+00	4.2994E-11	4.8075E-04	0.0000E+00
Year-5.00 (g/sq.ft.)	0.0000E+00	2.1594E-05	1.6941E-03	0.0000E+00
Year-10.00 (g/sq.ft.)	0.0000E+00	2.0033E-04	2.3980E-03	0.0000E+00
Year-20.00 (g/sq.ft.)	0.0000E+00	2.1804E-03	2.9334E-03	0.0000E+00
Year-30.00 (g/sq.ft.)	0.0000E+00	3.9918E-03	3.0781E-03	0.0000E+00
Year-40.00 (g/sq.ft.)	0.0000E+00	4.4138E-03	3.1094E-03	0.0000E+00
Year-50.00 (g/sq.ft.)	0.0000E+00	4.5029E-03	3.1161E-03	0.0000E+00

This table shows that until year 20, diffusion to the atmosphere was the main pathway of TCE emission from the profile. However, after 30 years, advection (transport with water flow) to the ground water becomes a more significant factor of TCE emission. The numbers from this table can be used as input in the MODFLOW model.

To add this table to the project report, in the **Report** window place the cursor to a position below the last graph. Return to the main window of the WHI UnSat Suite and <right click> in the Result View.

☞ **Insert To Report**. The table will appear in the report. Now you may print your report.

In the main menu of the **Report** window:

☞ **File/Print**. The report will be printed by your default printer.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

This concludes the VLEACH exercise. Close the WHI UnSat Suite by clicking **X** in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools the WHI UnSat Suite offers. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us at (519) 746-1798 or visit our web page, www.flowpath.com.

WHI UnSat Suite Tutorial: PESTAN

Modeling Fate of Pesticides in the Unsaturated Zone

WHI UnSat Suite for Windows 95/98/NT/2000

The WHI UnSat Suite combines three popular unsaturated zone models in a powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. These models are:

SESOIL - a popular US EPA model which is capable of simultaneously modeling water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess contaminant fate and transport for regulatory requirements.

PESTAN (PESTicide **AN**alitycal**)**- a model for assessments of groundwater contamination by pesticides used in agricultural practice.

VS2DT (Variably **S**aturated **2D** flow and **T**ransport**)** - a finite difference numerical model for determining the fate of agricultural chemicals, landfill leachate, and accidental chemical spills in the unsaturated zone.

VLEACH (**V**adose Zone **LEACH**ing model)- a model for evaluating the fate of volatile organic contaminants in the unsaturated zone.

All these popular models are seamlessly integrated with the WHI UnSat Suite and each one is compiled and optimized to run as a 32-Bit, native Windows application.

The WHI UnSat Suite shares the majority of its features with the other WHI software product Visual HELP which, besides its direct function, landfill designing, can be used as an efficient tool for the assessment of soil surface water balance. The two products are connected functionally and logically and in many cases the joint use of two products in consulting projects is very beneficial. Both products are joined together in the software package called WHI UnSat Suite Plus. To learn more about Visual HELP, review the demo exercise for this product.

Learning Objectives

In this demonstration exercise, you will:

- create an UnSat Suite project
- Use PESTAN as a tool for simulating the transport of Alachlor in the unsaturated zone;
- Prepare a project report

The instructions for this exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

Background Information for PESTAN Exercise

Alachlor was applied at an agricultural field with sweet corn. The area of the plot is 7 acres.

The application schedule was as follows:

- On June 1 the application was 2 kg/ha
- On June 20 the application was 1 kg/ha

In this exercise you will use UnSat Suite to simulate the transport and fate of Alachlor in the unsaturated zone and leaching of this contaminant to the groundwater. According to the WHI Unsaturated Zone modeling technology, the HELP and the PESTAN models are used in conjunction to assess the concentration of alachlor in soil water reaching the groundwater at the site. HELP is used to provide the high accuracy in assessment of the average groundwater recharge. PESTAN is used to simulate the unsaturated flow, transport and transformation of Alachlor in the unsaturated zone for a 1 year period. The Visual HELP simulation showed that the average infiltration rate into the soil profile is 9 in/year.

The following are parameters of the profile simulated with the PESTAN model:

The profile consists of one layer which has a thickness of 30 feet and is composed of silt. The known parameters of silt are:

- dry bulk density 1.5 g/cm3
- effective porosity 0.47
- fraction organic content 0.1

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type-	Type in the given word or value
select-	Click the left mouse button where indicated
α -	Press the **<Tab>** key
↵ -	Press the **<Enter>** key
☞ -	Click the left mouse button where indicated
☞ ☞ -	Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

Starting WHI UnSat Suite

Once the WHI UnSat Suite is installed, click Windows **Start** button, choose **Programs**, navigate to the folder that contains WHI UnSat Suite, and click **UnSat Suite**.

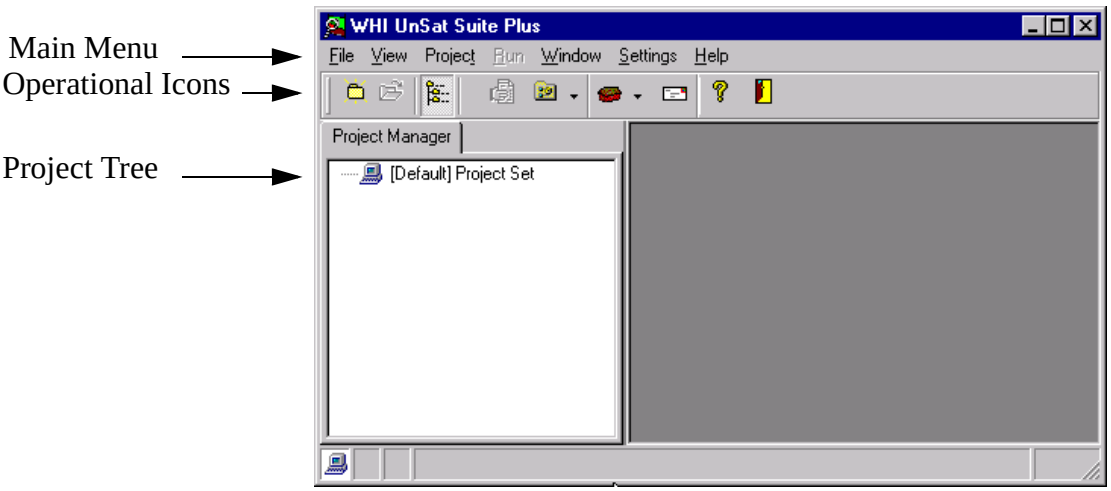
The installation program will also place an icon  on your desktop, which you can also double click to start the WHI UnSat Suite exercise.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

An introductory screen will be displayed when you open WHI UnSat Suite:



After the program is loaded, the main WHI UnSat Suite window will appear on your display:



The WHI UnSat Suite interface has been designed to help you navigate through the program with ease. The main menu is located at the top with operational icons located directly below the main menu items to allow quick access to many options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

Module I: Creating a New Project

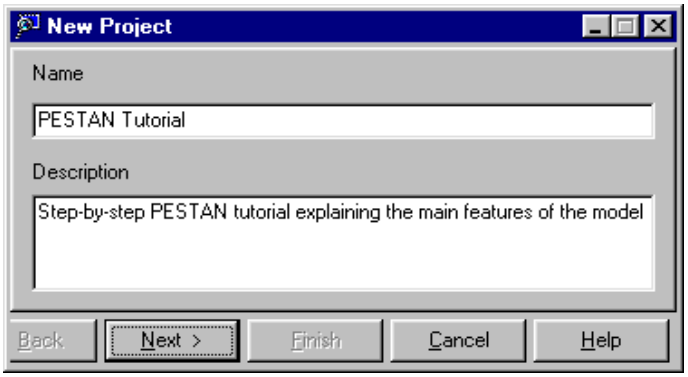
To create a new project, go the the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new WHI UnSat Suite project.

In the **Name** box type: **PESTAN Tutorial**.

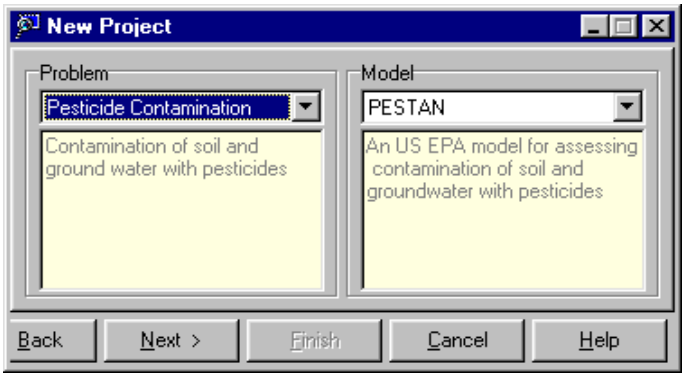
In the Description box type: **Step-by-step PESTAN tutorial explaining the main features of the model.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, to view problems which can be examined with the WHI UnSat Suite Plus. Also notice that the WHI UnSat Suite automatically recognizes the appropriate model that should be used with the selected Problem type.

☞ **Pesticide Contamination.** The **PESTAN** model will appear in the **Model** list-box.



☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The WHI UnSat Suite Unit Converter was developed to maximize the flexibility for data input and output unit conversion. With WHI UnSat Suite you may create your unit templates to fit the format of your data.

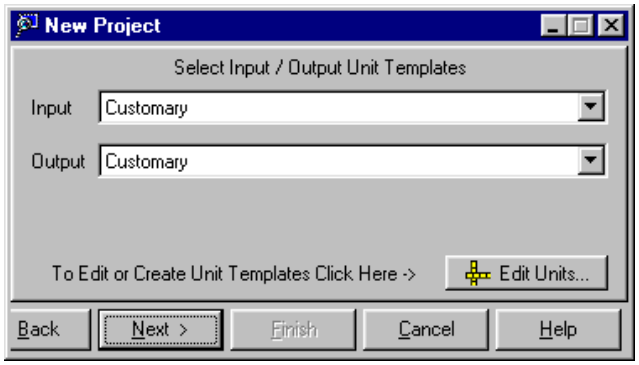
For this exercise we will specify that all input and output be in **Customary** (US) units.

From the **Input** drop-down list,

☞ **Customary**

From the **Output** drop-down list

☞ **Customary**



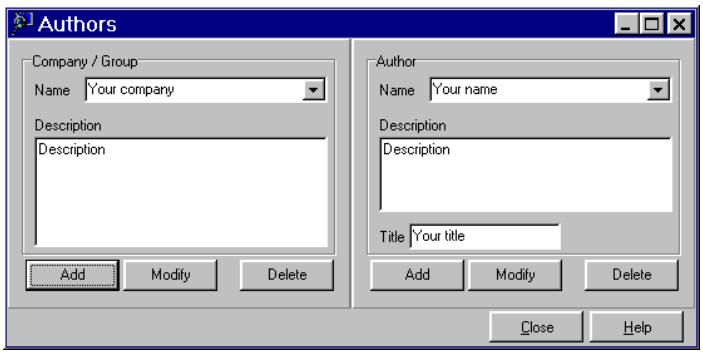
☞ **[Next]**

Assigning Authors and Clients

WHI UnSat Suite allows you to create and maintain your own database of project Authors and Clients for use in future projects. The information which you input/specify here will appear automatically in your project report. To add a new author to the database

☞ **[Authors]**

The **Authors** dialog box will appear:



Each author is associated with a company.

Company/Group

In this area, you can add, modify, or delete companies and groups. To add your company to the database, in the **Name** box:

Type your company name

In the **Description** box:

type your address and phone number.

☞ **[Add]**

Author

In this area, you can add, modify, or delete authors from companies. To add a new author, input your information in the **Name** box, **Title** box, and **Description** box.

☞ **[Add]**

☞ **[Close]** to return to the **New Project** dialog box.

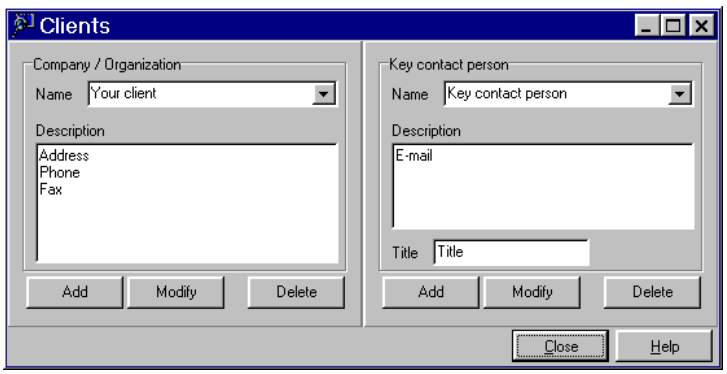
while back in the **New Project** dialog box, click the drop-down arrow and select your name from the list of the Authors.

Clients

To add a new client to the database:

☞ **[Clients]**

The **Clients** dialog box will appear



Each contact is associated with a company or organization.

Company/Organization

In this section you can add, modify, or delete companies and organizations. To add a new company, edit the information in the **Name** box and **Description** box, and click **[Add]**. To update an old company with current information, edit the information, and click **[Modify]**. To delete a company, select the company from the list, and click **[Delete]**.

Key Contact Person

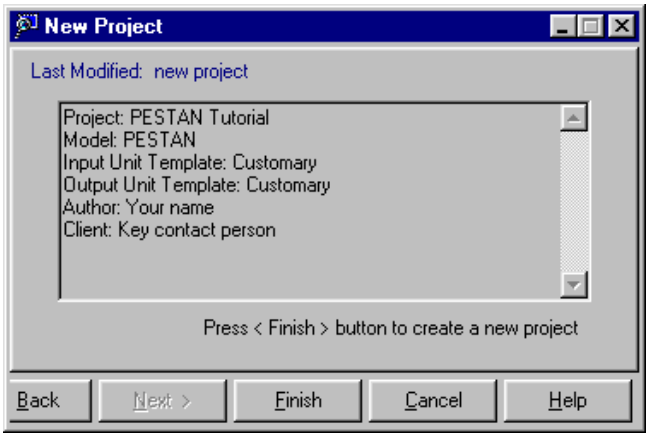
In this area you can add, modify, or delete contacts from companies. To add a new contact, edit the information in the **Name** box, **Title** box, and **Description** box, and click **[Add]**. To update the information about a client, edit the information, and click **[Modify]**. To delete a client, select the client from the list, and click **[Delete]**.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **[Close]** to return to the **New Project** dialog box. Select an existing client from the list, or create a new client by clicking **[Clients].**

☞ **[Next]**

All the project information will be presented, as seen in the figure below:



☞ **[Finish]** to save the information as a project in the database and add it to the Project Tree.

After the project settings are specified, the **New Profile Wizard** will open.

Creating a Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:



In this dialog box you may select an already existing profile template or you can build a new profile 'from scratch'. In this demonstration exercise you will build a new profile.

☞ **[create new profile]**

☞ **[Next]**

The dialog box with default profile parameters will appear. In this dialog box we may assign the parameters of the simulated profile.

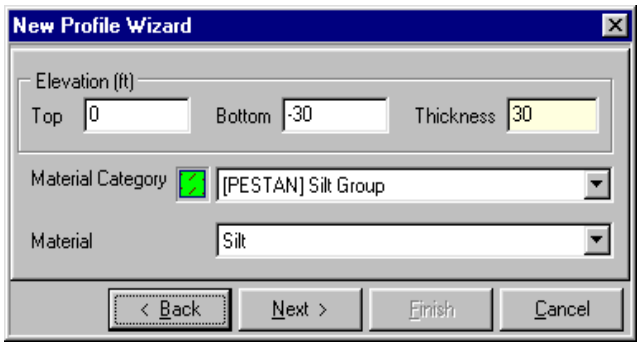
☞ in the **Bottom** text box and input **-30** for the elevation of profile bottom. The value of **Thickness** will change accordingly.

☞ the drop-down arrow in the **Material Category** drop-down list box.

Select **[PESTAN] Silt group**. The default material **Silt** will appear in the **Material** drop-down list box.

Note: You may add as many materials to specific Material Category group as you need, once inside the model, in Material Designer.

As a result of editing, the **New Profile Wizard** window will appear as below:



☞ **[Next]**

All the profile information will be presented:



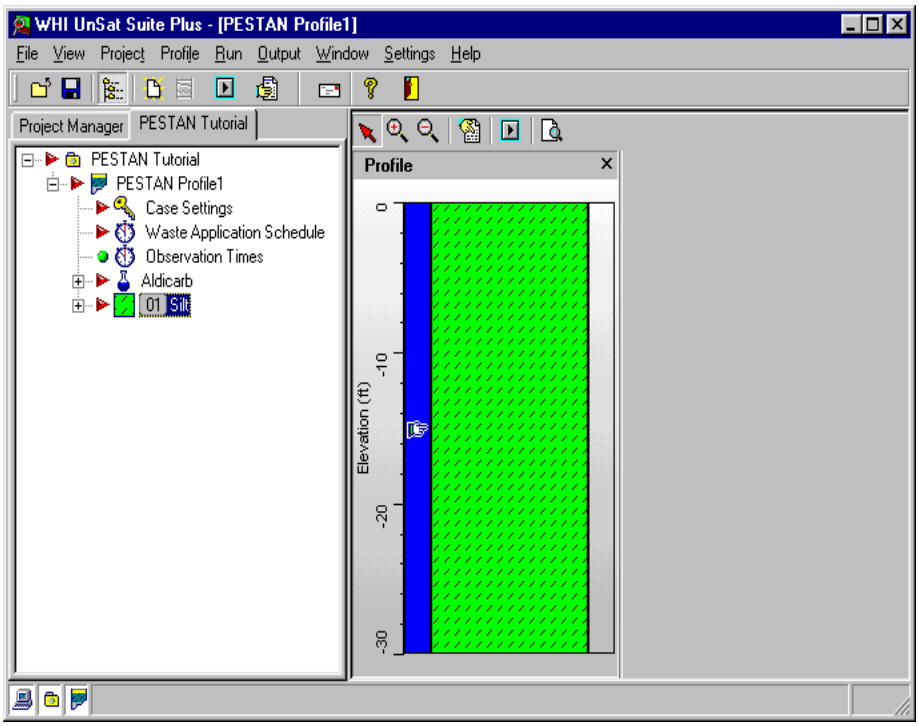
☞ **[Finish]** to load the profile.

The new project details will be added to the Project Tree View, the project will open and the profile picture will appear in the Profile View. This concludes creating a new project using the WHI UnSat Suite. We will now continue with designing the profile and setting specific simulation details.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Module II: Designing a Profile and Setting Case Specifications

The following picture will appear on your display:



In the Profile View, located in the center of the WHI UnSat Suite window, you will see the loaded profile with a depth scale at the left.

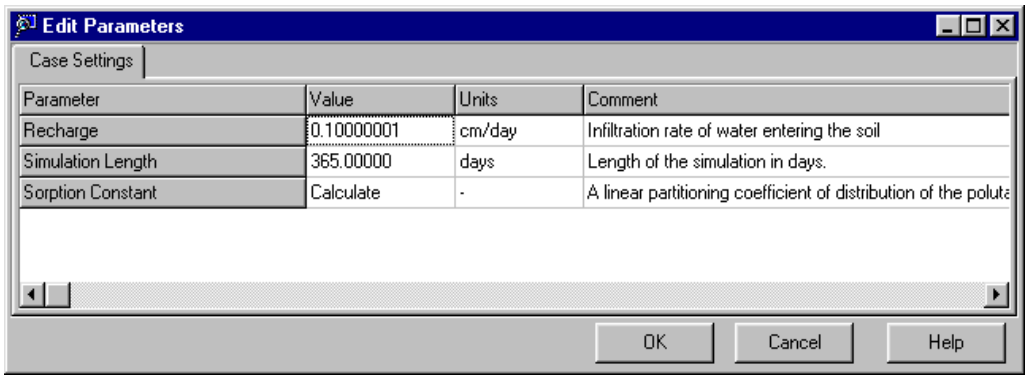
A description of a layer will appear when the mouse pointer is pointed at the layer.

The Project Tree View in the left part of the WHI UnSat Suite window provides access to parameters for your simulation scenario.

Case Settings

Constructing the unique simulation case requires you to input several simulation parameters and initial conditions of the profile. These parameters may be edited in the project parameter group labeled **Case Settings**.

☞☞ on **Case Settings** in the **Project Tree View**. The following dialog box will appear:



☞ **Units** field beside **Recharge**

☞ the drop-down arrow

☞ **in/year** from the appeared drop-down list

☞ **Value** field beside **Recharge**

Type: **9**

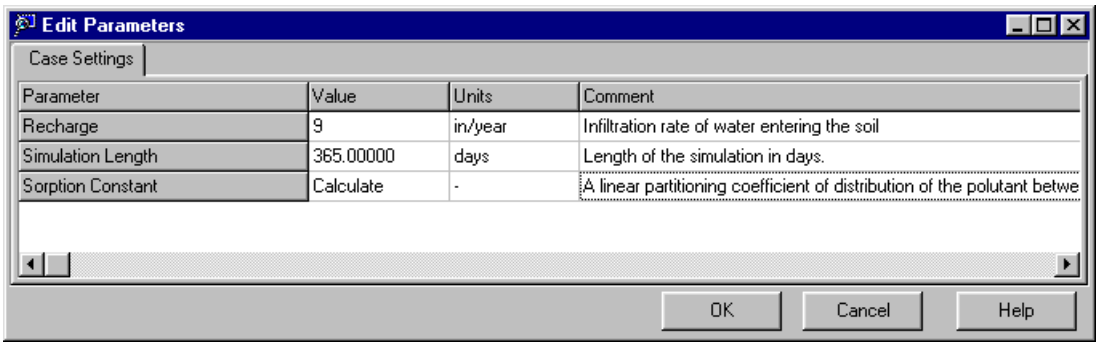
Leave the parameter **Simulation Length** which is 365 days by default.

Next you have to specify the way you are going to input **Sorption Constant.** This parameter, which is the linear Freundlich sorption coefficient (**Kd**), is not commonly known for this particular pair of soil and pesticide. However, it can be determined as a function of two widely available characteristics. For many cases **Kd** can be presented as a product of the fraction organic content of the soil (**Foc**) and the organic carbon partition coefficient (**Koc**) of the pollutant:

$$Kd = Koc * Foc, \qquad (1)$$

With the WHI UnSat Suite you may either input the value of the Sorption Constant Kd directly or calculate it according to (1). In the latter case you would have to input values of the fraction organic content of the soil (**Foc**) under the soil properties and organic carbon partition coefficient (**Koc**) of the pollutant under the chemical properties. This option, which is default, will be used in the exercise. To use this option, leave the setting **Calculate** for **Sorption Constant.**

Finally, the Case Settings dialog window will appear as below:



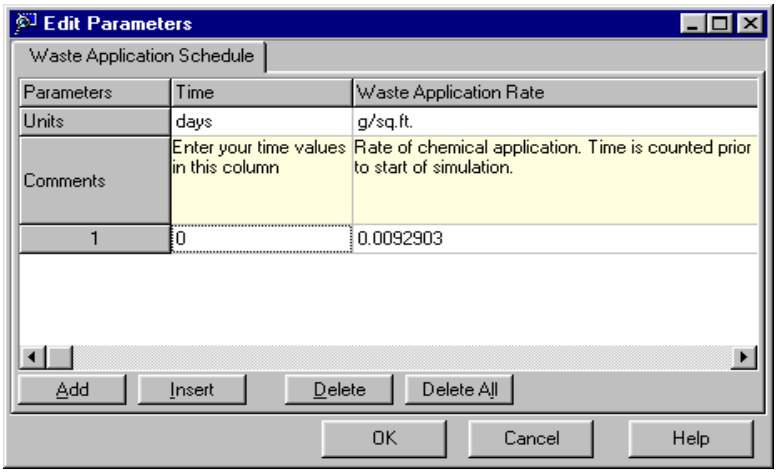
☞ **OK** to confirm and save entered settings.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Setting Waste Application Schedule

The Waste Application Schedule has to be specified for the soil profile. The PESTAN simulation technology assumes that the pollutant is applied at the soil top several times before the infiltration starts. Before the begining of infiltration the pollutant is a subject to solid-phase decay. Right at the time when the infiltration starts, all remaining pollutant instantly dissolves and from this moment is subject to the processes of advection, sorption and liquid-phase decay. To use this technology for practical cases, it is advisable to set the start of the infiltration at the moment when the pollutant was applied the last time. The time of waste application is measured backwards from the start of infiltration. In your case, this means that you should consider June 20th as a start date of infiltration (**Time** 0) with a loading of 1 Kg/ha. The June 1st application occurred 20 days prior to the start of infiltration, thus at **Time** 20, the pollutant load was 2 Kg/ha.

☞☞ **Waste Application Schedule** in the **Project Tree View.** The following dialog box will appear:



☞ in the **Units** row (second row) under the **Waste Application Rate**

☞ the drop-down arrow

☞ **kg/ha**

☞ in the **Waste Application Rate** field for the last pollutant application (fourth row)

Type: **1**

☞ **[Add],** to add another pollutant application. When you click **Add**, the value of the Waste Application Rate from the previous line will be copied.

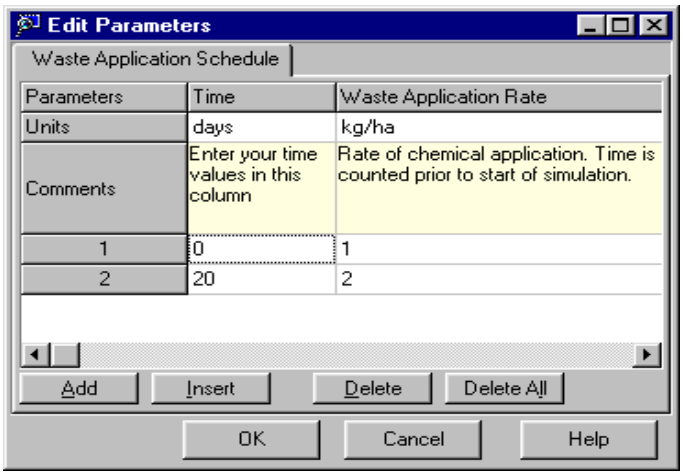
☞ in the **Waste Application Rate** field for the second pollutant application

Type: **2**

☞ in the **Time** field for the second application

Type: **20** (interval in days between two applications).

As a result, the **Waste Application Schedule** dialog box will appear as below:



☞ **OK** to confirm and save entered schedule.

Setting Output Times

UnSat Suite allows you to view model results for the output times that you specify yourself.

☞☞ **Observation Times** in the Project Tree

☞ in the **Time** field for the first period (first line)

Type: **10**. Leave the **Is output Desired?** field with default **yes** selection.

☞ **[Add],** to add another observation time. When you click **Add** for the first time, the first line will be copied.

☞ in the **Time** field for the second period

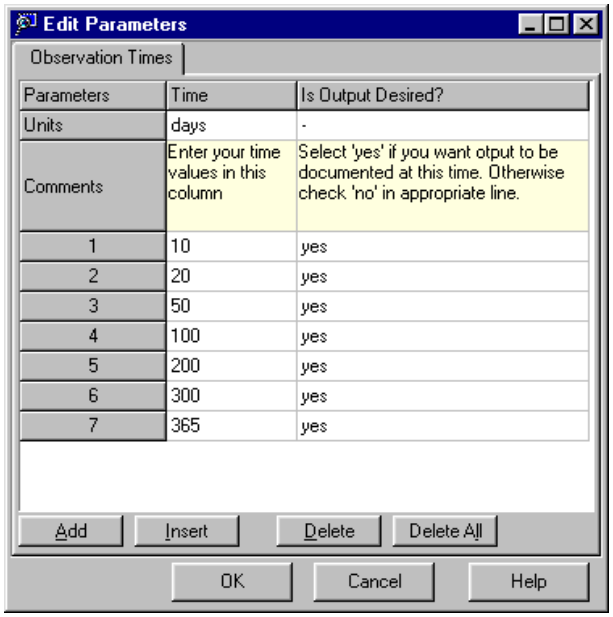
Type: **20**

☞ **[Add]**

Type: **50** in line 3, **100** in line 4, **200** in line 5, **300** in line 6 and **365** in line 7 clicking **[Add]** after each input.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

As a result, the **Observation Times** dialog box will appear as below:

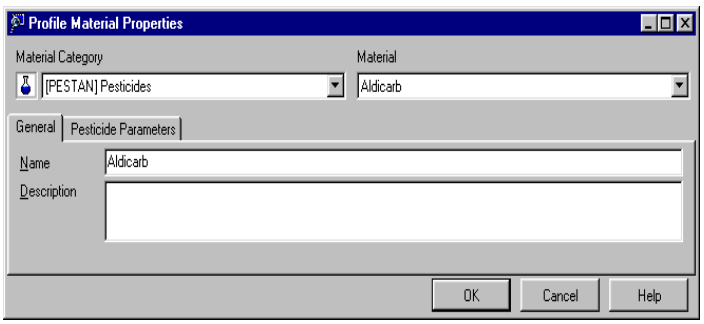


☞ **OK** to confirm and save entered observation times.

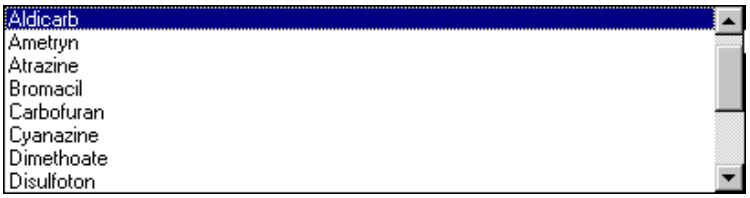
Substituting the Pollutant

According to the simulation scenario, you are going to simulate the fate of Alachlor in the soil. However, by default the model has loaded parameters of Aldicarb into your project. Next you have to substitute the pollutant.

☞ ☞ the name of the default pesticide **Aldicarb** in the Project Tree. The **Profile Material Property** dialog box will appear:



☞ the drop-down arrow of the **Material** listbox. The list of available pesticides will appear:



Scroll up the list and,

☞ **Alachlor**.

Note: After you have selected the pesticide, you may edit its parameters to make them completely match your case.

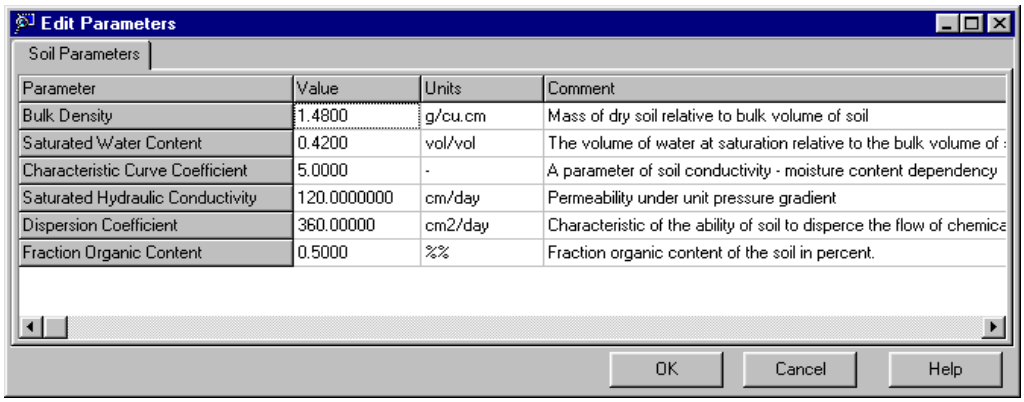
☞ **OK** to confirm changes.

Finally, you have to edit layer parameters.

Editing Layer Parameters

☞ **'+'** sign in the box to the left of the **Silt** icon in the Project Tree.

☞ ☞ **Soil Parameters**. The **Edit Parameters** dialog box will appear.



In this dialog box, you can view and edit general parameters of the layer.

☞ **Value** field beside the **Bulk Density**

Type: **1.5**

☞ **Value** field beside the **Saturated Water Content**

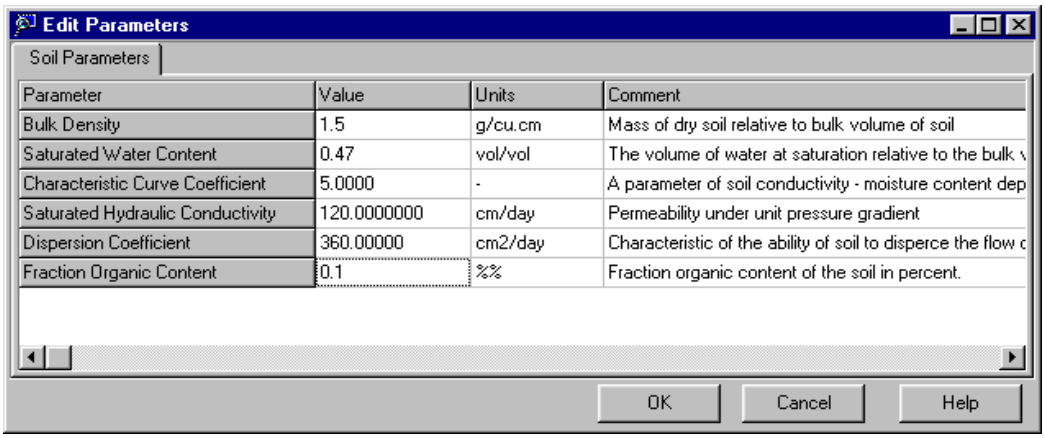
Type: **0.47**

☞ **Value** field beside the **Fraction Organic Content**

Type: **0.1**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

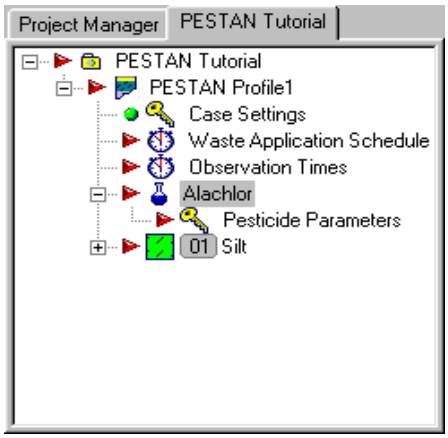
The new **Edit Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-],** beside **Silt** in the Project Tree.

Eventually, the Project Tree should appear as below:



This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **PESTAN tutorial 1**, which was specifically developed for you and contains all the changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

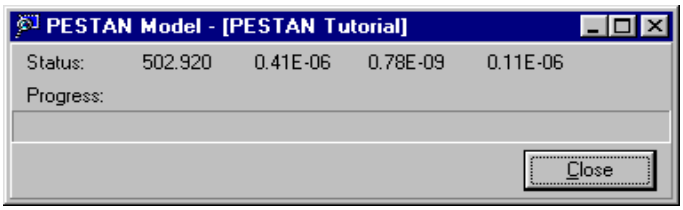
☞ ☞ project **PESTAN tutorial 1.**

Module III: Running the Model, Viewing Output, and Reporting

Running the PESTAN Model

To run the model for a single profile, click the  **Run** icon above the Profile View.

A progress bar will appear to indicate the computation progress, as shown below:



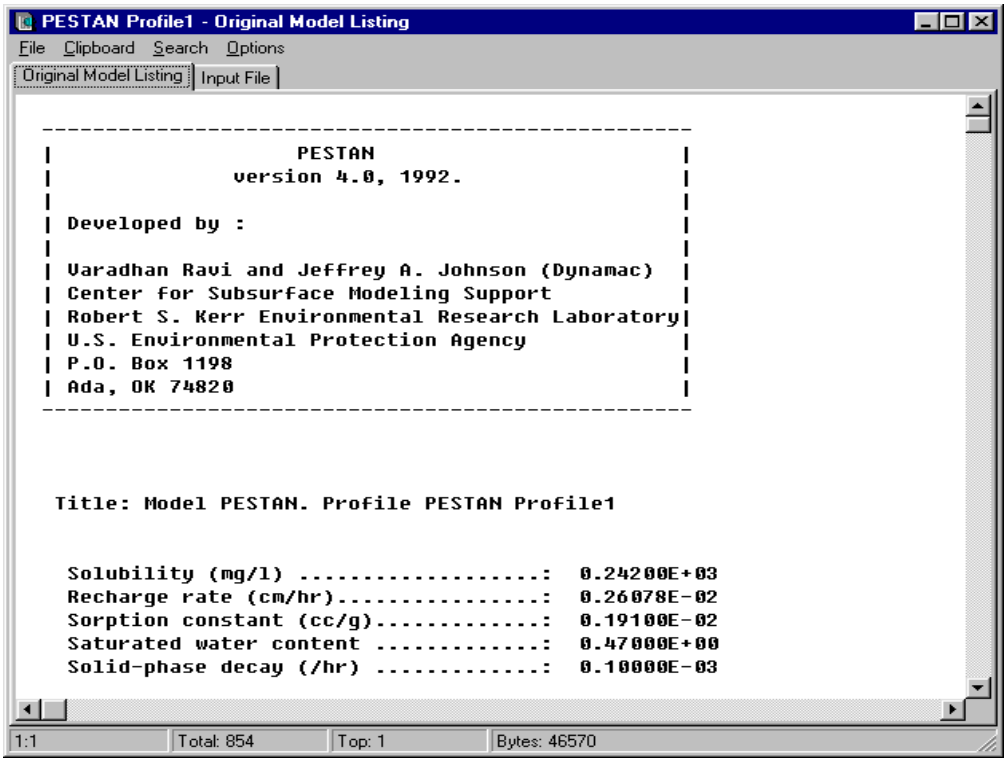
Viewing the Original PESTAN Output File

To view the original PESTAN output:

☞ **Output** in the main menu, and then

☞ **View Original Listing**

The following window will appear:



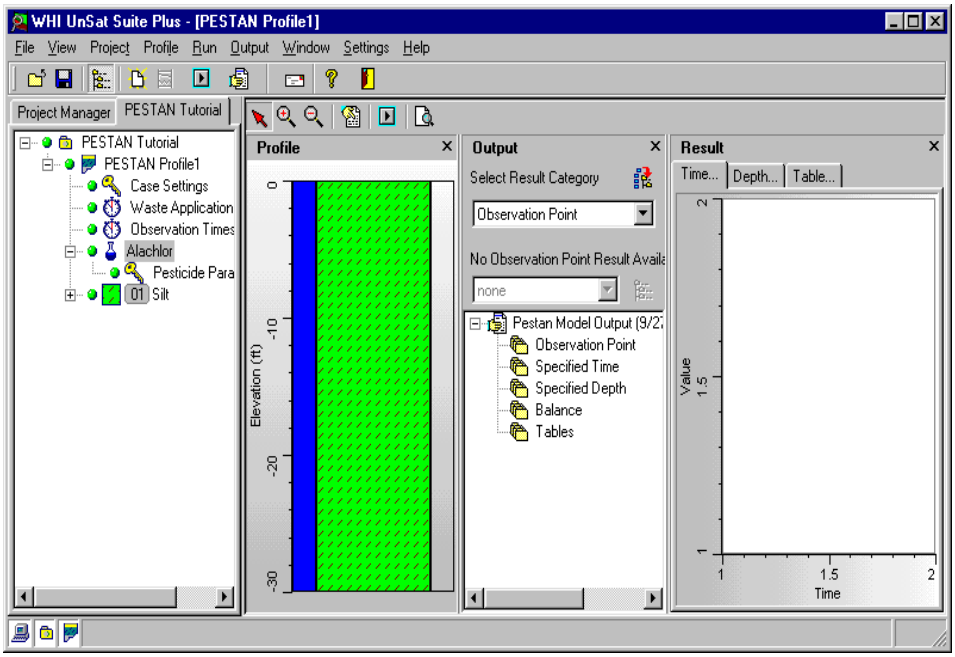
 Copyright 2000 Waterloo Hydrogeologic, Inc.

Here you can view the original listing, find specific expressions, mark and print parts or make a complete printout of the model results.

☞ the **X** in the top right corner to close the original listing.

Viewing the Output Graphs

After the model has successfully ran, the **Output** View and **Result** View window will open and the UnSat Suite window will appear as below:

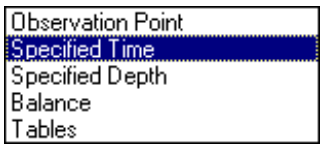


To enlarge the viewing area click the icon  to close the Project Tree View, or click the '**X**' in the Profile View to close it.

All settings for the output presentation can be edited in the **Output** View.

To select the output category:

☞ the drop-down arrow in the **Select Result Category** list box. The possible result groups will appear as in the list box presented below:



The **Result Categories** present the following:

- **Observation Point -** presents results for the observation point specified by the user (not considered in this exercise).
- **Specified Time** - plots of contaminant concentration in soil water, in soil solid phase (adsorbed) and total concentration versus depth for specified observation times.
- **Specified Depth** - plots of contaminant concentration in soil water, in soil solid phase (adsorbed) and total concentration versus time for specified depths (cell centers).
- **Balance** - plots of the amount of contaminant remaining within the unsaturated profile (per unit

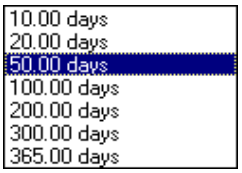
area) in soil water and in soil solid phase (adsorbed), total mass of remaining pollutant and decayed mass in liquid phase versus time.

- **Tables: Balance** - tabular presentation of the contaminant balance (see above)

Viewing Depth Dependent Data for Specified Time

☞ **Specified Time** in the **Select Result Category**

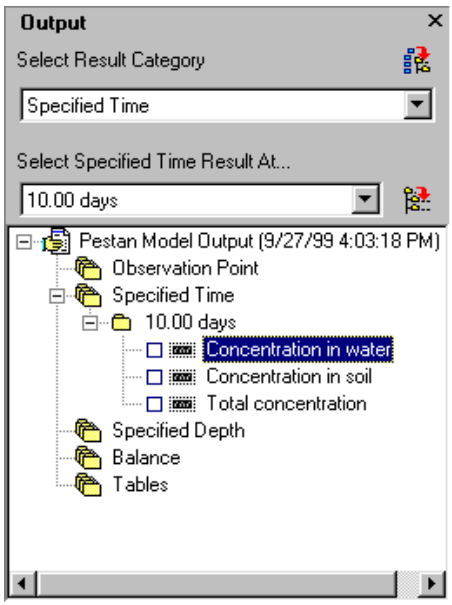
☞ the drop-down arrow in the **Select Specified Time Result at...** list box. The list of observation times will appear:



☞ **10.00 days** which is the first available output and shows initial conditions

To view all results available for this specific time, click the  icon to the right of the **Select Specified Time Result at...** box.

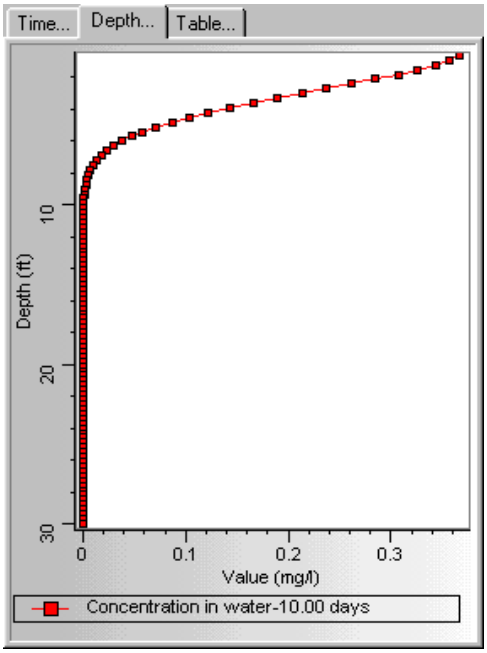
The list of available results will open in the Result Tree:



☞ the check box beside **Concentration in water.**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

A graph of the variable will appear in the Result View window showing the concentration of contaminant in soil water:



Lets watch how the concentration in the soil water changes in time.

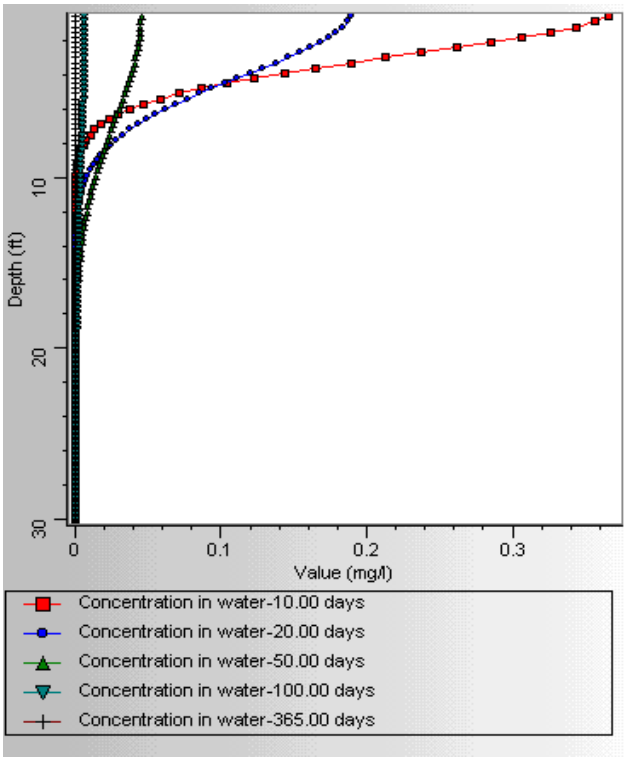
☞ the drop-down arrow in the **Select Specified Time Result at...** list box.

☞ **20.00 days**

click the  icon to the right of the **Select Specified Time Result at...** box to open the list of variables.

☞ the check box beside **Concentration in water** from the Output Tree

Repeat this operation for **50, 100 and 365 days.** As you select the variable for specific time, it will appear in the graph.



The graph shows how the wave of contaminant moved down the profile. It can also be noted that the maximum concentration decreases in time and gets close to 0 after 100 days.

This, in brief, shows you some of the options available for viewing depth dependent data for the observation times that you specify. In the same way you may view data for **Concentration in soil** and **Total Concentration.**

At this time you may want to start creating your project report and add the above graph to it. Next you will learn how to create a WHI UnSat Suite report.

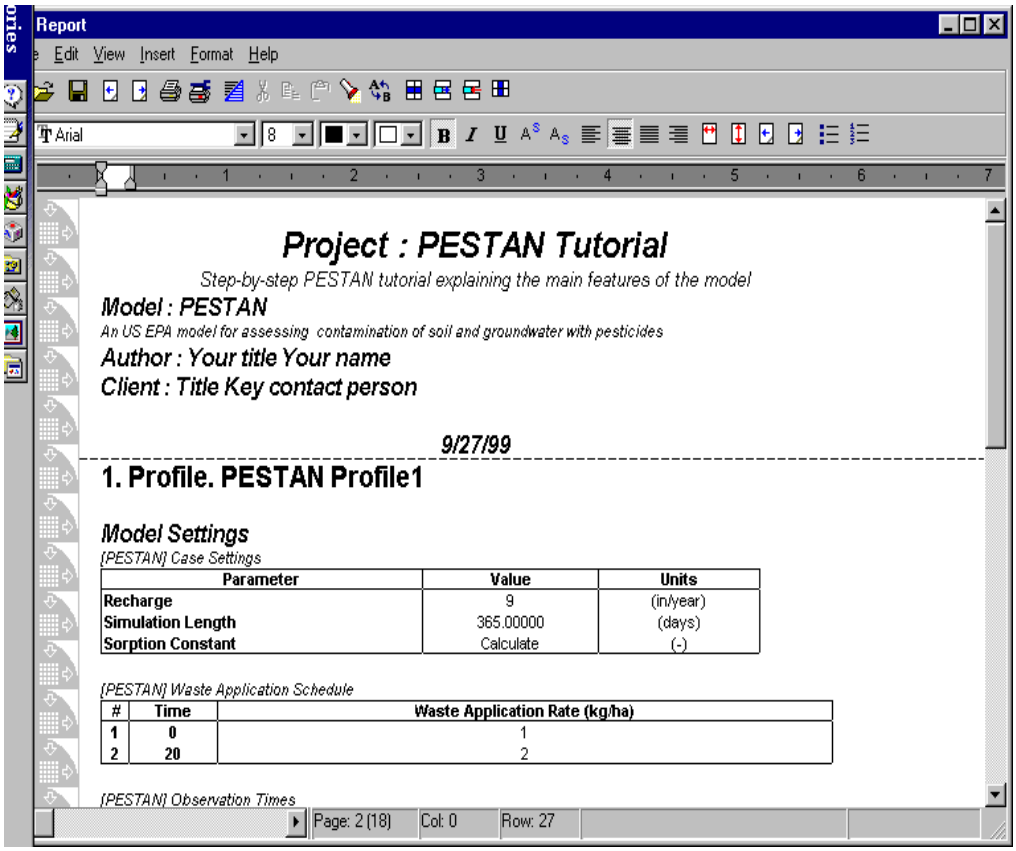
Creating a Report

To present results of your PESTAN simulation to your clients you may use the UnSat Suite Report Generator.

To create a report and add the project input data, click the icon  from the Operational Icons tool bar.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The report will appear in a separate window:

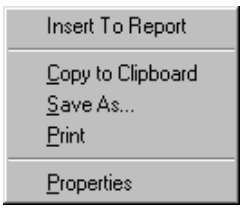


In this window you may edit the report, input your own text and add any types of graphics or table outputs produced by WHI UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph to the report:

Scroll the bar in the **Report** window to the bottom and place the cursor to a position below the last table in the report. Return to the main window of WHI UnSat Suite and <right click> on the graph in the Result View. The following menu will appear:



☞ **Insert To Report**. The graph appears in the report.

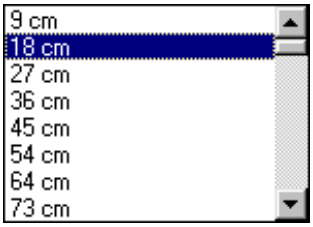
The graph may appear smaller than the original. To view the graph of desired size, click in the **Report** window and stretch the graph until it reaches the proper size.

Now we have the report started. Further analysis will produce additional tables and graphs that can be added at a later date.

Viewing Time Dependent Data for Specified Depth (Breakthrough Curves)

☞ **Select Result Category** list box and select **Specified Depth**

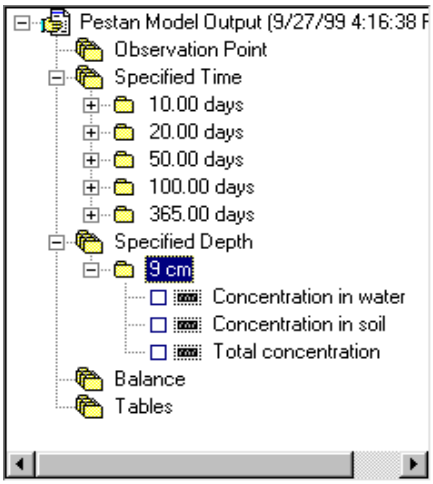
☞ the drop-down arrow in the **Select Specified Depth Result at...** list box. The list of observation times will appear:



☞ **9 cm** which is the center of the first cell

To view all results available for this specific depth, click the  icon to the right of the **Select Specified Depth Result at...** box.

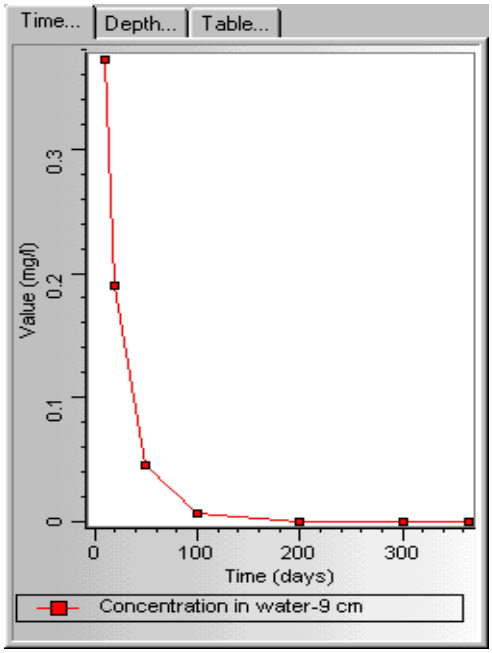
The list of available results will open in the Result Tree:



☞ the check box beside **Concentration in water.**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

A graph of the variable will appear in the Result View window showing the concentration of contaminant in soil solute at the depth of 9 cm:



Lets watch how the concentration in the soil water changes at different depths.

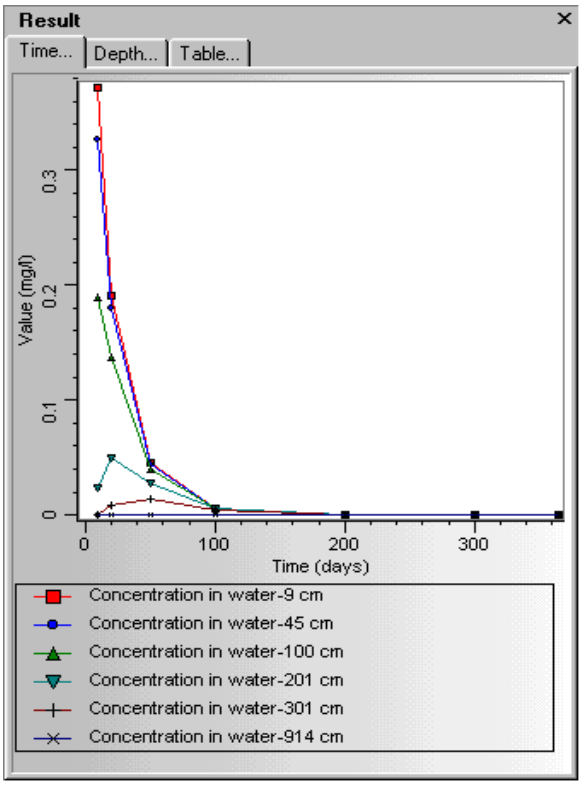
☞ the drop-down arrow in the **Select Specified Depth Result at...** list box.

☞ **45 cm**

click the  icon to the right of the **Select Specified Time Result at...** box to open the list of variables.

☞ the check box beside **Concentration in water** from the Output Tree

Repeat this operation for depths **100 cm, 201 cm, 301 cm** and **914 cm** (you may need to scroll the list of observation depths to reach some of them)**.** As you select the variable for specific depth, it will appear in the graph. Finally the graph will look like following:



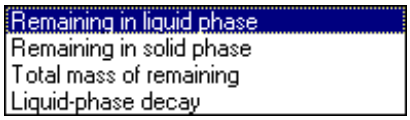
The graph shows that the contaminant will not reach the bottom of the profile (depth 914 cm). Peak concentration of Alachlor decreases down the profile and the majority of contaminant transformation happens during the first 100 days.

This, in brief, shows you some of the available options for viewing time dependent data for the observation depths that you specify. In the same way you may view data for **Concentration in Gas phase** and **Concentration in Soil Solid Phase.**

Viewing Contaminant Balance

☞ **Select Result Category** list box and select **Balance**

☞ the drop-down arrow in the lower **Select Balance Result at...** list box. The list of available balance variables will appear:



☞ **Remaining in liquid phase**

☞ the  icon to the right of the **Select Balance Result at...** box.

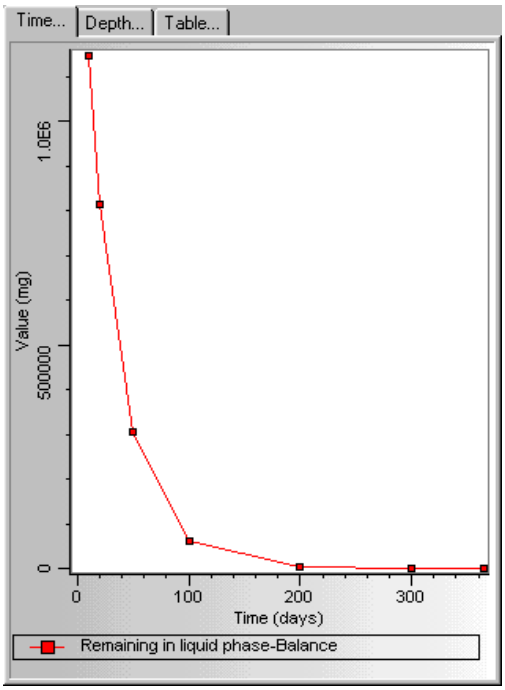
The variable with a check box will appear at the bottom of the Result Tree:

☞ the check box beside **Remaining in liquid phase.** A warning will appear to indicate that the new variable has different size or units compared to the previously presented variable.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **Yes**

A graph of the new variable will appear in the Result View window showing the plot of the amount of contaminant within the unsaturated profile in the soil water versus time.



The graph shows that Alachlor was totally removed from the soil water phase after 250 days.

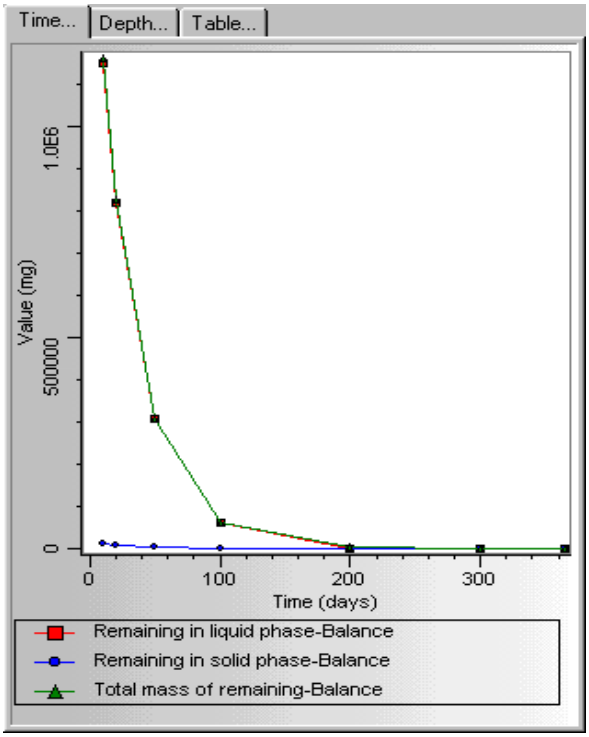
☞ **Remaining in solid phase** from the **Select Balance Result at...** list box

☞ the  icon to the right of the **Select Balance Result at...** box.

The variable with a check box will appear in the Result Tree:

☞ the check box beside **Remaining in solid phase**

Repeat these operations for the variable **Total mass remaining.** Each time you select a specific variable, it will appear in the graph. Eventually, the Output window will appear as below:



This graph shows you the changes in the amount of Alachlor in two phases. These data are also presented in table form.

Viewing Tables for Accumulated Contaminant Balance

☞ **Select Result Category** list box and select **Tables**

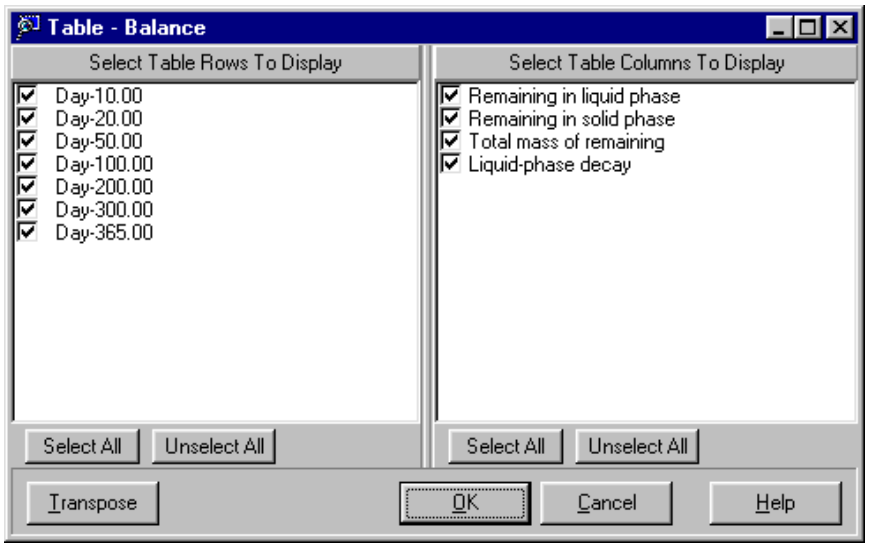
Table **Balance** will appear in the lower **Select Tables Result at...** list box.

☞ the  icon to the right of the **Select Tables Result at...** box.

The variable with a check box will appear in the bottom of the Result Tree.

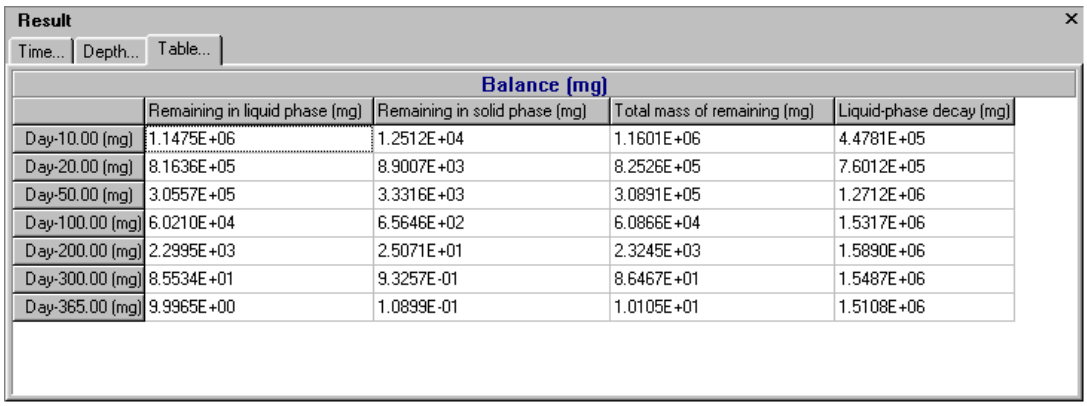
 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ the check box beside **Balance**. A **Table - Balance** dialog box will appear:



In this dialog box you may select which variables and output times you would like to have in your table.

☞ **OK**. The **Balance** table will appear in the **Output** window:



	Remaining in liquid phase (mg)	Remaining in solid phase (mg)	Total mass of remaining (mg)	Liquid-phase decay (mg)
Day-10.00 (mg)	1.1475E+06	1.2512E+04	1.1601E+06	4.4781E+05
Day-20.00 (mg)	8.1636E+05	8.9007E+03	8.2526E+05	7.6012E+05
Day-50.00 (mg)	3.0557E+05	3.3316E+03	3.0891E+05	1.2712E+06
Day-100.00 (mg)	6.0210E+04	6.5646E+02	6.0866E+04	1.5317E+06
Day-200.00 (mg)	2.2995E+03	2.5071E+01	2.3245E+03	1.5890E+06
Day-300.00 (mg)	8.5534E+01	9.3257E-01	8.6467E+01	1.5487E+06
Day-365.00 (mg)	9.9965E+00	1.0899E-01	1.0105E+01	1.5108E+06

To add this table to the project report, in the **Report** window place the cursor to a position below the last graph. Return to the main window of the WHI UnSat Suite and <right click> in the Result View.

☞ **Insert To Report**. The table will appear in the report. Now you may print your report.

In the main menu of the **Report** window:

☞ **File/Print**. The report will be printed by your default printer.

This concludes the PESTAN exercise. Close the WHI UnSat Suite by clicking **X** in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools the WHI UnSat Suite offers. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us at (519) 746-1798 or visit our web page, www.flowpath.com.

Introduction to Visual HELP for Windows 95/98/NT 3
The HELP Model 3
History 3
New Features of Visual HELP version 2.1 4
System Requirements 4
Installing Visual HELP 4
Visual HELP Demonstration Exercise 4
How to Contact Waterloo Hydrogeologic Inc. 5
Demonstration Exercise 6
Learning Objectives 6
Terms and Notation 6
Starting Visual HELP 7
Module I: Creating a New Project 8
Selecting the Location 9
Selecting Units 12
Assigning Authors and Clients 12
Selecting an Existing Profile 13
Module II: Designing a Landfill Profile 15
Viewing the Default Profile 15
Editing the Layer Structure of the Landfill Profile 16
Resizing Layers 17
Deleting a Layer 18
Restoring a Layer 19
Splitting a Layer 19
Inserting a Layer 20
Editing Layer Parameters 22
Setting Initial Moisture Conditions & Runoff Parameters 24
Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting 26
Viewing Weather Data 27
Running Visual HELP 29
Viewing Results 29
Viewing the Output Graphs 29
Viewing Tables 31
Creating a Report 32

Introduction to Visual HELP for Windows 95/98/NT

The HELP Model

HELP (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile model used to test the effectiveness of landfill designs and also to predict landfill hydrologic processes and groundwater recharge rates. Now, HELP has become a requirement for obtaining landfill operation permits in the U.S.A. and is becoming the standard for landfill optimization around the world.

HELP is a quasi-two-dimensional hydrologic model and uses the following input data to create the landfill profile:

- Weather (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil (porosity, field capacity, wilting point, and hydraulic conductivity)
- Engineering design data (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can be multi-layered, consisting of a combination of natural (soil) and artificial materials (waste, geomembranes) with an option to install horizontal drainage. HELP also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

Built-in Database tools include:

- Weather Generator, a tool for synthetic generation for up to 100 years of daily values of precipitation, air temperature and solar radiation.
- Soil, waste and geomembrane database which contains parameters for 42 materials.

History

WHI has developed a Windows 95/98/NT interface for version 3.07 of the HELP model which was released in May 1998 under the name Visual HELP version 1.101. Visual HELP can be used as a pre- and post-processor for the HELP model, allowing you to perform all of the functions of the DOS version 3.08 of HELP. Visual HELP was presented at the trade show at SWANA's WASTECON 1998 and ISWA World Congress 1998 in Charlotte, North Carolina. Currently, more than 300 copies of this product are being used by consulting companies, government regulators and Universities in U.S.A., Canada, U.K., Germany, Australia, Sweden, and Hungary.

After running through this demonstration tutorial, we trust you will agree that Visual HELP is now the most complete and easy-to-use modeling tool for designing & optimizing landfills.

Visual HELP is also now available as part of the **WHI UnSat Suite Plus**. This ultimate suite of 1-D unsaturated flow models is comprised of Visual HELP, VS2DT, VLEACH, and PESTAN. Now you can design landfills, predict leachate & contaminat migration, and show degradation of pesticides in one package! For details about the **WHI UnSat Suite Plus,** please call (519) 746-1798 or visit our web page at www.flowpath.com.

New Features of Visual HELP version 2.1

Profile Viewing and Editing:

- A layer can now be split into two separate layers
- Layers are systematically numbered, and labelled in the Project Tree, which simplifies interpreation of the results

Weather Generating:

- New databases and GIS searching tools have been developed for the major regions of the world (more than 3000 weather stations)
- Data in NOAA format can now be imported automatically
- Diagnostics are provided to detect missing records in NOAA files

Output Presentation:

- Output units can be changed now without restarting the project
- Water Balance Tables have been added
- New tree-like interface structure allows the user to easily select results
- Report Generator (a new feature of Visual HELP) allows you to display, print and export to Microsoft Word project input settings and output graphs and tables.

System Requirements

To run Visual HELP, you will need the following minimum system configuration:

•Windows 95/98, or Windows NT 3.5.1 or later;

•IBM PC or compatible,

•Pentium 100 processor;

•CD-ROM drive for software installation;

•32 Mb of RAM;

•60 Mb of free hard disk space; and

• Minimum display resolution of 1024 x 768

The following fonts should be installed on your computer: MS Sans Serif, Arial, and Courier New.

Installing Visual HELP

The Visual HELP 2.1 demo must be installed on your computers hard disk. Please ensure that your system meets the minimum requirements before proceeding with the installation. To install, simply navigate to the Visual HELP page found on the Demo CD ROM and select the appropriate button.

Visual HELP Demonstration Exercise

We have included example data with the installation to help you familiarize yourself with the Visual HELP 2.1 interface. The examples are found as the default projects in Visual HELP. Since this is a demonstration installation, you are restricted to only running the default projects, however are free to browse the entire Visual HELP interface.

How to Contact Waterloo Hydrogeologic Inc.

If after reviewing the Visual HELP demo, you would like to contact Waterloo Hydrogeologic Inc.(WHI), please use the address below:

Waterloo Hydrogeologic, Inc.
180 Columbia Street West, Unit 1104
Waterloo, Ontario, CANADA
N2L 3L3
Phone: (519) 746-1798
Fax: (519) 885-5262

To purchase the product: contact the WHI Sales Department at sales@flowpath.com or visit our web page order form at: http://www.flowpath.com/order_form.html.

Please forward addtional comments or suggestions to the Visual HELP development team at:

Mikhail Gogolev, Product Manager mgogolev@flowpath.com

Dmitri Eidenzon, Chief Programmer deidenzon@flowpath.com

Demonstration Exercise

Learning Objectives

In this demonstration exercise, you will:

- Use Visual HELP as a tool for landfill design and hydrologic simulation;
- Use the Weather Generator to predict meteorological data for a twenty year period and use the predicted data as input to Visual HELP;
- Run the Visual HELP simulation, view the water balance of the landfill, assess the performance of the drainage and leachate collection system, and
- Prepare a report.

The instructions for this demo exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

The default landfill profile used in this exercise was designed in accordance with the EPA's recommendations. This design is also used in the official HELP 3.0 User's Manual. The layer properties for the default landfill profile were taken from the HELP database. The simulated landfill is located near Toronto, Ontario, Canada. The area of the landfill is 1 acre.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type- Type in the given word or value

select- Click the left mouse button where indicated

a - Press the **<Tab>** key

↵ - Press the **<Enter>** key

☞ - Click the left mouse button where indicated

☞☞- Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

NOTE: All theoretical parts of the demo tutorial are italicized

Starting Visual HELP

To start Visual HELP, you must have it installed on your hard disk. Once installed, click Windows **Start** button, choose **Programs**, navigate to the folder that contains Visual HELP, and click **Visual HELP**.

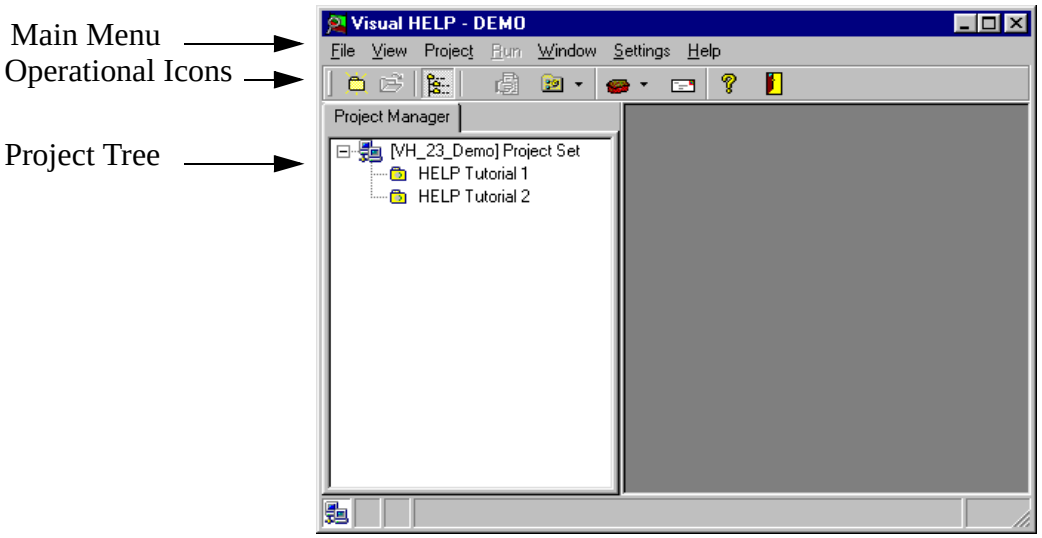


The installation program will also place an icon on your desktop, which you can also click to start the Visual HELP demo exercise.

An introductory screen will be displayed when you open Visual HELP



After the program is loaded, the main Visual HELP window will appear on your display:



The Visual HELP interface structure has been designed to help you navigate through the program with ease. The main menu is located at the top, similar to our other programs. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

The default projects **HELP Tutorial 1**, and **HELP Tutorial 2** are snapshots of a HELP Tutorial project taken at different stages of model development. Since the demo version does not allow you to save or edit new projects, these default projects have already been prepared for you to use. In the full working version of Visual HELP, all results of the editing process are saved within the same project.

Module I: Creating a New Project

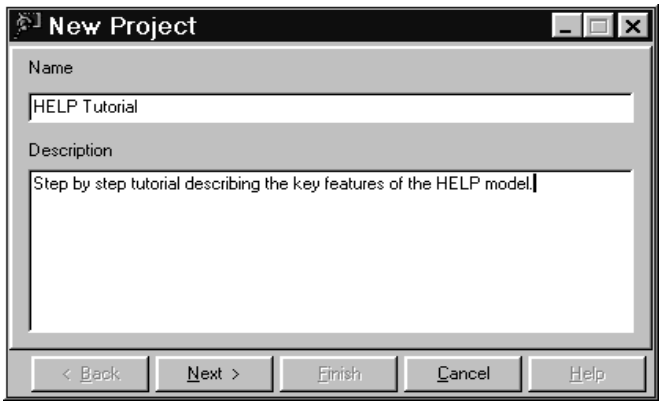
To create a new project, go the the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new Visual HELP project.

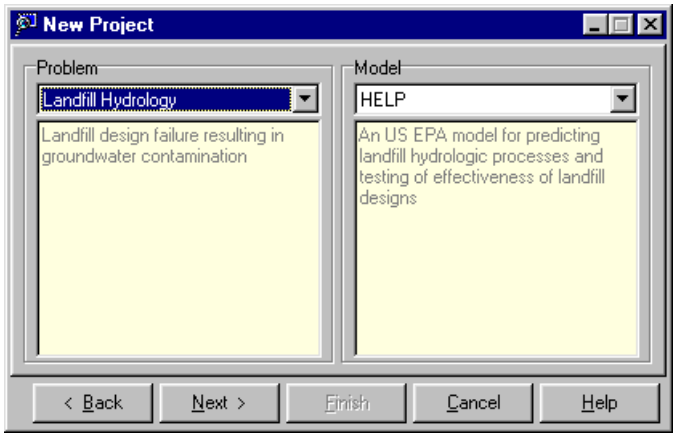
In the **Name** box type: **HELP Tutorial**.

In the Description box type: **Visual HELP step-by-step tutorial.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Problem class that should be used with the select Problem type.

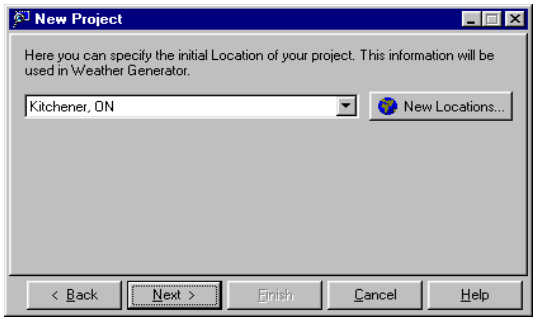


☞ **[Next]**

Once completed, the site location must be defined.

Selecting the Location

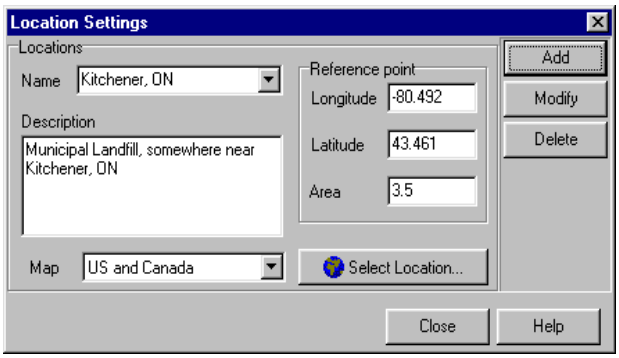
We will now select the location of our project using the GIS - WHI Locator. The following dialog box should already appear on your screen.



At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



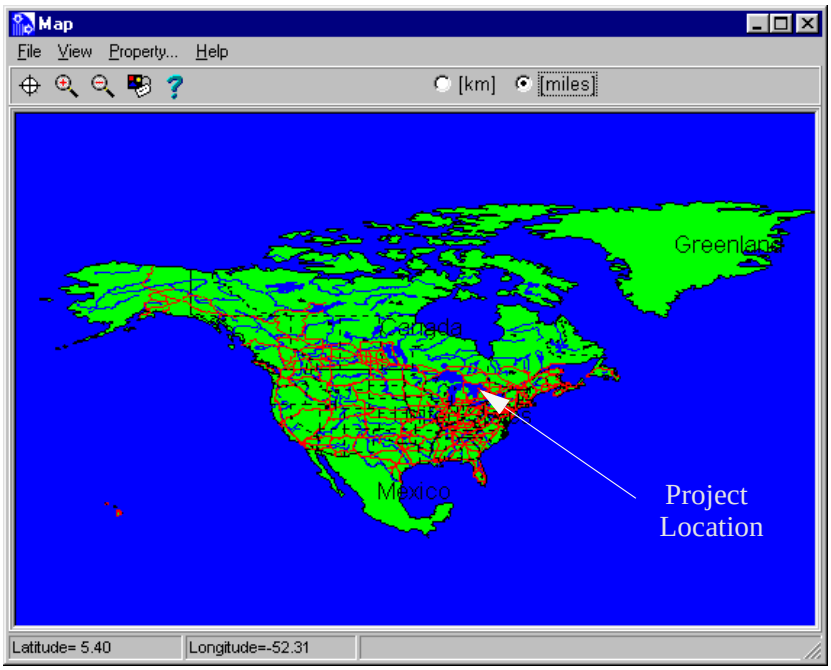
From the **Map** drop-down list,

☞ **North America,** to enable the WHI Locator, developed by WHI. The additoinal maps found in the list refer to various global regions that will help you with build projects from around the world!



☞ **[Select Location...],** to define our specific project location using the WHI Locator map.

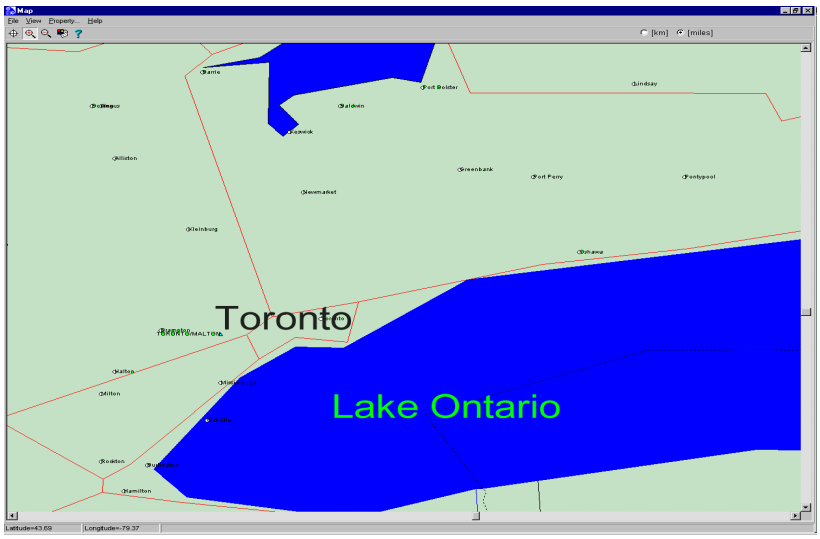
As shown below, a map of USA, Canada, and Mexico appears on your display:.



Now lets select Toronto, Ontario, Canada as our site location.

☞  , to zoom into our location.

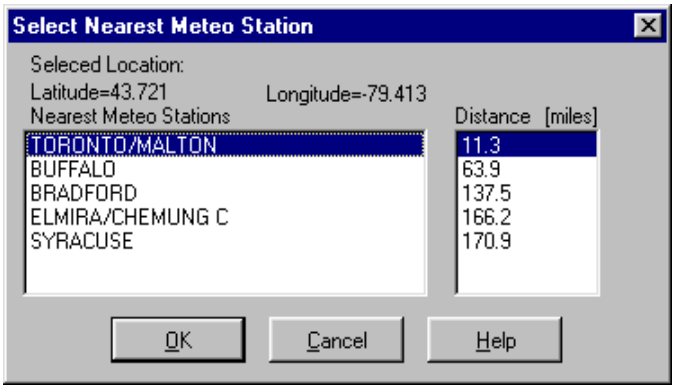
Move the mouse to the right of the region where the landfill project is located (in center of the Great Lakes region). The coordinates, seen in the bottom left of the screen, should read approximatelly Latitude:46.8, Longitude:-84.7. Press the left mouse button and stretch a zoom rectangle to the right of the Lake Ontario, release the button (approximate Latitude:43.3, Longitude:-76.2). After one or two zooms, figure below should resemble your screen.



Now select Toronto to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

Move the crosshair to the spot on the map where Toronto is located and click on it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you wish to calculate the distance in kilometers, simply click the appropriate option button at the top of the map.

In the "Select Nearest Meteo Station" dialog box,

☞ **[TORONTO/MALTON],** as the nearest weather station.

☞ **[OK]**

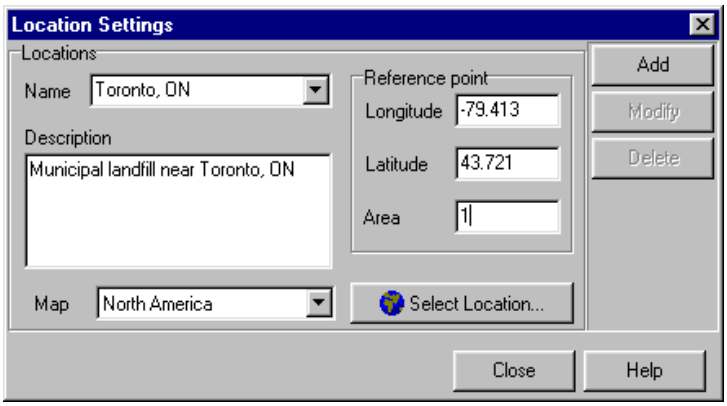
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: Toronto, ON., in the **Name** box.

Type: Municipal landfill near Toronto, ON., in the **Description** text box.

Type: 1 , in the landfill area text box: **1**.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database for future use.

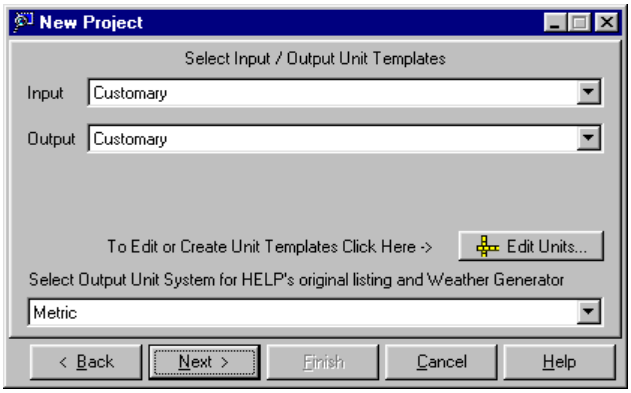
☞ **[Close]**

In the **New Project** dropdown list, select **Toronto, ON**.

☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.



The Visual HELP Unit Converter was developed to maximize the flexibility for data input and output unit conversion. To show you the range of possibilities, just view the list of available units for hydraulic conductivity, by opening the **[Edit Units]** option.



For this demo excercise we will specify that all input units will be in **Metric**, and output will be in **Customary** units.

From the **Input** drop-down list,

☞ **Metric**

From the **Output** drop-down list

☞ **Customary (in)**.

From the **Select Output Unit System for original listing and Weather Generator** list box,

☞ **Metric**

☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

All the project information will be presented, as seen in the figure below.



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings were specified, the **New Profile Wizard** will open.

Profiles and Multiple Profiles in Visual HELP. *For the purpose of hydrological simulation, a profile represents a part of a landfill that is assumed to have the same cross-section throughout. The profile contains all of the layers of the landfill that it represents. It may also contain details of engineering components, such as subsurface drainage, leachate recirculation systems, geomembranes, geonets, and composite liners. The surface of the profile can be sloped, which is typical for the peripheral parts of the landfill, or flat, which is typical for the central part of the landfill. The slope of the landfill usually mimics the shape of the waste body. At the time of landfill closure, the waste layer at the periphery of the landfill is usually sloped and the entire landfill site is capped by several layers, which form the landfill cap. Although drainage pipes are not usually installed in the landfill cap, permeable sand layers in the cap can transport water to the bottom drain or sump that surrounds the landfill. This process can be simulated with Visual HELP 2.1.*

A typical landfill can be represented as a set of profiles. This is done by creating several profiles in one project. This is useful because it allows you to use one profile to simulate the middle of the landfill, and several other profiles to simulate the edges, where the cross-section is tapering.

Selecting an Existing Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:

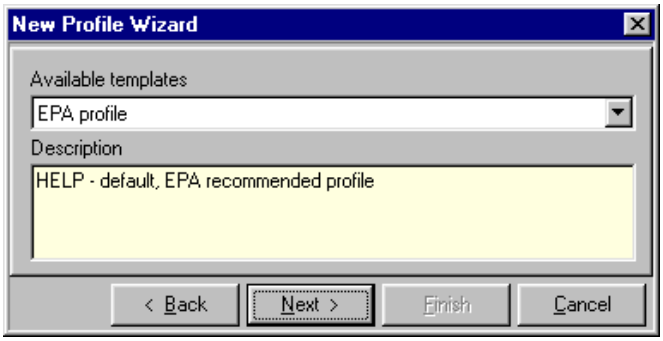


In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the default profile template.

☞ **[use existing profile template]**

☞ **[Next]**

The **Available templates** list will appear:



☞ **EPA Profile**

☞ **[Next]**

All the profile information will be presented:.



☞ **[Finish]** to load the profile.

The new project details will be added to the project tree. This concludes creating a new project using Visual HELP. We will now continue with designing the landfill profile.

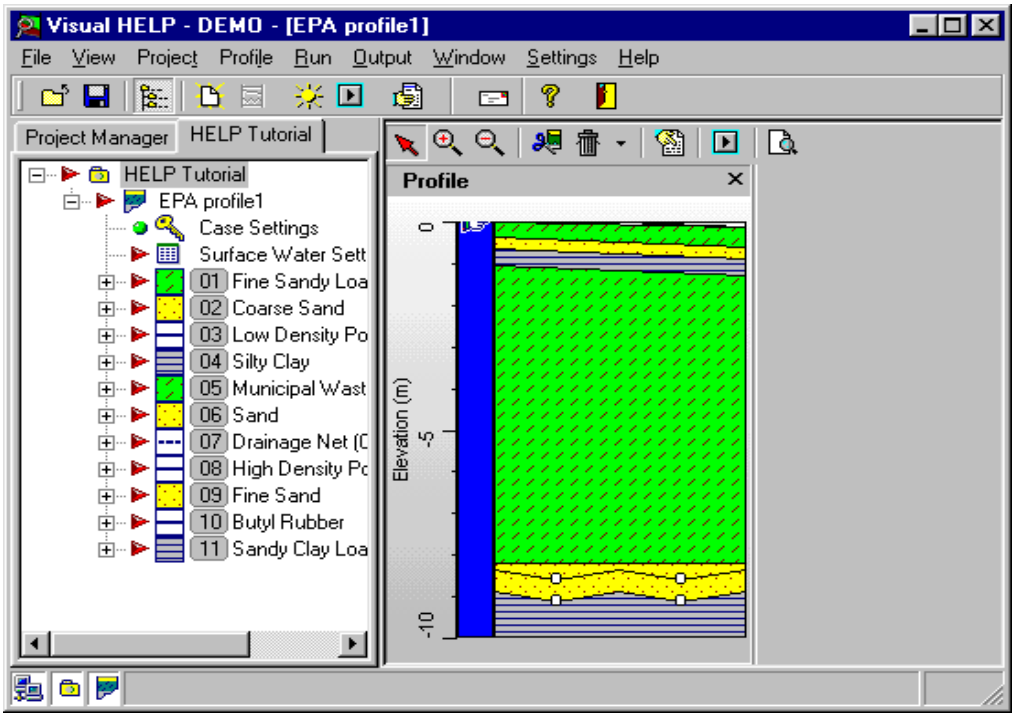
Module II: Designing a Landfill Profile

If you have skipped the steps in Module I and wish to start the Visual HELP demo exercise from this point,

☞☞ **HELP Tutorial 1,** from the Project Tree.

If you have run through Module I, you may continue with the project **HELP Tutorial.**

In both cases, the following picture will appear on your display.



Viewing the Default Profile

In the Profile View, located in the center part of the Visual HELP window, you will see the loaded default landfill profile with a depth scale.

A description of each landfill layer will appear when the mouse cursor is pointed at the layer.

As you can see, this profile consists of:

- a sloped landfill cap in the depth interval 0.0 - 1.2 m, consisting of fine sandy loam, coarse sand and silty clay
- a 7 m thick waste layer in the depth interval 1.2 - 8.2 m; and
- a double liner leachate collection and removal system in the depth interval 8.2-9.6 m.

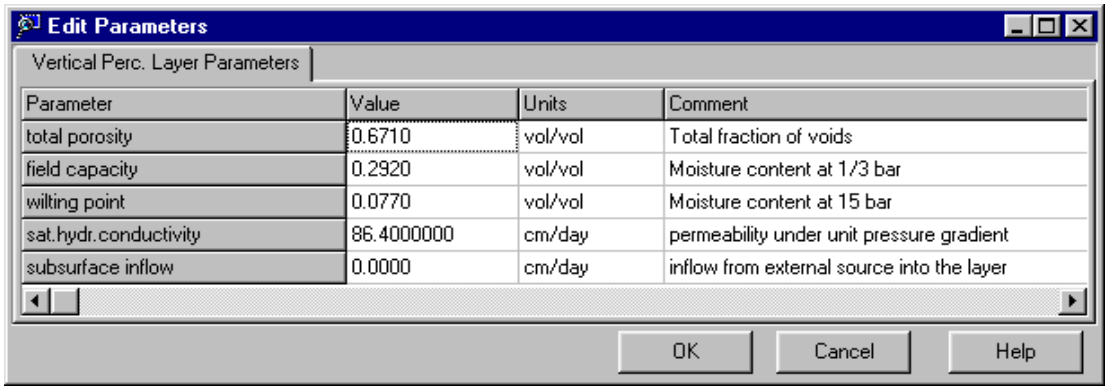
The profile construction can also be seen in the Project Tree View in the left part of the screen.

The Project Tree View is also used to view and modify the parameters of specific layers.

☞ **'+'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

☞☞ **Vertical Percolation Layer Parameters**. The **Edit Parameters** dialog box will appear.

The **Edit Parameters** menu allows you to modify the values of specific parameters.



☞ **[Cancel],** in the **Edit Parameter** dialog box to exit.

☞ **'-'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

As you can see, the Project Tree View can be used to easily access specific Profiles, Profile Layers, and Layer Parameters.

Now that you have become familiar with some viewing functions of Visual HELP, you will now learn how to edit the profile. In this part of the exercise we will:

- learn how to edit the layer structure of the landfill profile
- learn how to edit parameters of individual layers
- learn how to set initial moisture conditions and runoff parameters of the profile

Editing the Layer Structure of the Landfill Profile

To assist you in designing a landfill, Visual HELP offers many convenient tools for resizing, splitting, inserting and deleting layers. Visual HELP is a graphical user interface for the US EPA HELP model and, as such, obeys all conventions of the original HELP model.

The layers in the landfill are classified by the hydraulic function that they perform. Four categories of layers are available in original HELP:

•vertical percolation layers,

•lateral drainage layers,

•barrier soil liners, and

•geomembrane liners.

The topsoil and waste layers are generally vertical percolation layers. Sand layers above liners are typically lateral drainage layers; compacted clay layers are typically barrier soil liners. Geomembranes are typed as geomembrane liners. Composite liners are modeled as two layers. In the original HELP, geotextiles are not considered as layers unless they perform a unique hydraulic function.

In Visual HELP, taking into account that this class of landfill materials is growing extensively, geotextiles and geonets were defined as a separate category.

Although HELP allows a wide range of layer combinations, there are some basic rules and restrictions for the arrangement of layers in a profile that should be followed. These rules are obeyed by Visual HELP and you will get a warning if you try to delete or insert a layer incorrectly.

Resizing Layers

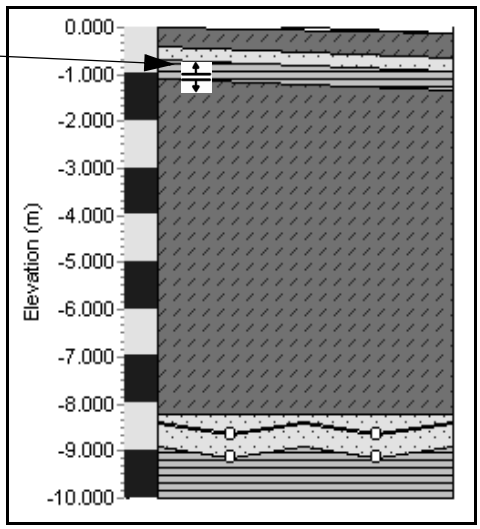
In Visual HELP, the layers are either resizable or non-resizable.

Resizable Layers *Layers built from soil and waste, grouped in the Vertical Percolation, Lateral Drainage, Barrier Soil design categories, are resizable.*

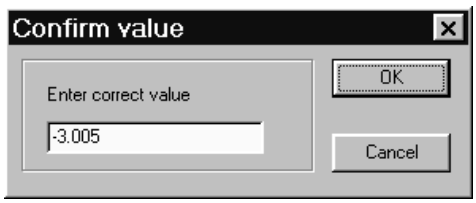
Non-Resizable Layers *Layers built from industrially produced materials with fixed thickness, grouped in the Geomembrane Liner and Geotextile and Geonet categories, are non-resizable.*

To test the resizing feature, move the mouse arrow on the **Municipal Waste (312 kg/cub.m)** layer in the profile view and move the arrow upwards to the top of the layer. As soon as the cursor encounters the layer's boundary, the cursor symbol changes:

New Cursor Symbol



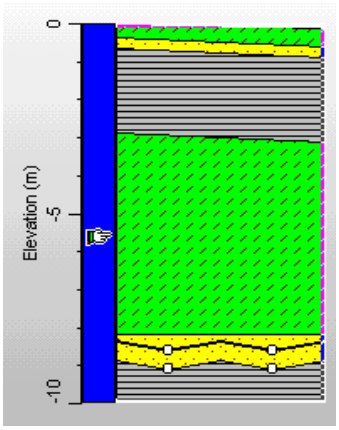
Press the left mouse button and, holding the mouse button, move the cursor downwards to approximately **-3.000** and release the mouse button. A **Confirm Value** dialog box will appear.



Type: -3.000 in the **Enter correct value** box.

☞ **[OK]**

The profile will change according to the new setting specified.



Note: ***Moving the layer's boundary resizes both layers which are separated by this boundary, however it does not change the total thickness of the profile. You can also change the thickness of an individual layer. This feature will be described later in this tutorial.***

Now resize the municipal waste layer back to its original size.

Place the mouse arrow on the **municipal waste** layer in the profile view, and move the arrow up until the cursor symbol changes.

Click and drag the boundary to approximately -1.200.

Type: -1.2, in the **Enter correct value** text box.

☞ **[OK]**

Deleting a Layer

We will now examine how to delete a layer.

☞ **Silty Clay**, in the Project Tree View.



.

Click the right mouse button. A shortcut menu will appear

☞ **Delete**

The layer **Silty Clay** will disappear from the Profile View. Notice an 'X' beside the silt clay layer to represent a deleted layer.

Restoring a Layer

Now let us restore the **Silty Clay layer**.

<Right click> the layer's name in the Project Tree View.

Click **Restore,** the original layer returns.

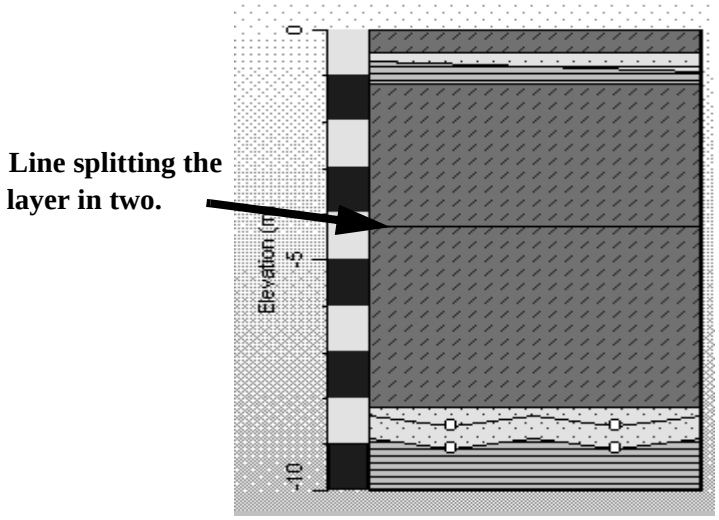
Splitting a Layer

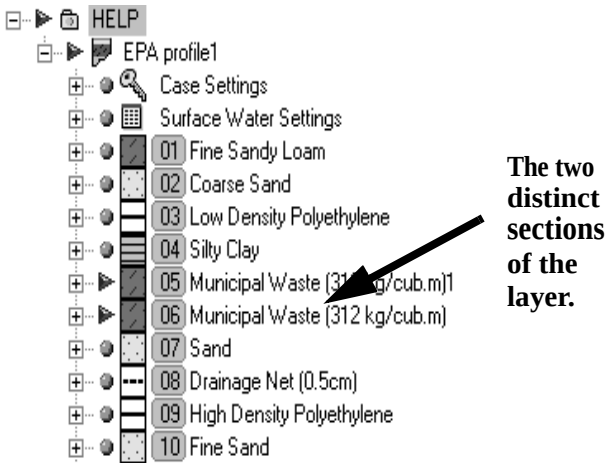
You can split a layer up into multiple sections and substitute materials for each section or assign different values of parameters for each section.

<right click> on the **Municipal Waste (312 kg/cub.m)** layer in the profile view.

☞ **Layer/Split**

A line will appear through the layer at cursor position and a new layer will appear in the project tree. Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.



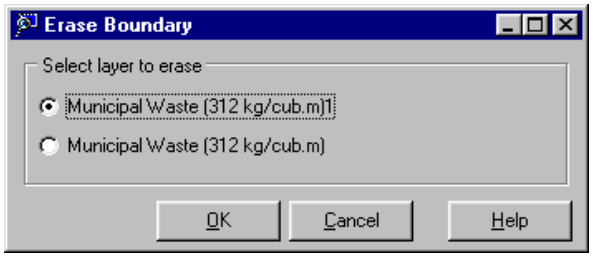


You may also substitute a material in the new layer or insert another layer between two separate parts of the layer. To return to the original profile construction, simply merge the two layers.

Place a cursor at the boundary between two parts of a layer and **<right click>**,

☞ **Merge Layers**.

In the **Erase Boundary** dialog box select the layer to erase: **Municipal Waste (312 kg/cub.m)1**



☞ **[OK]**.

Inserting a Layer

<right click> on **Sand**, in the Project Tree View

The following shortcut menu will appear



:

☞ **[Insert Layer]**

An empty **Profile Material Properties** dialogue box will appear. You can now specify the new material to insert.

Note: The new layer will always be inserted above the current layer.

Note: To insert the new layer inside the existing layer, split the layer first with the split function.

Material Category Click a category from the **Material Category** list. The list will be limited to those categories that are specified by the HELP Model only.

☞ **[HELP] Barrier Soil Liner.**

Material Click the appropriate material from the **Material** list. Each layer category is associated with several textures that can be used in the design of the layer. The texture number in Visual HELP corresponds directly to the texture number in the original DOS HELP.

☞ **[Clay Loam**]

Once the material is selected, the top and bottom settings are activated.

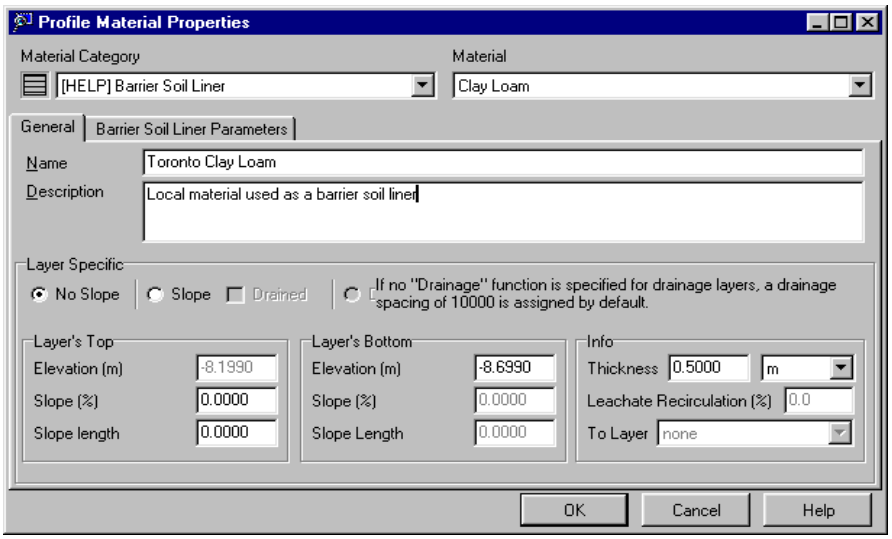
Name **Type: Toronto Clay Loam,** as the unique name for the material.

Description **Type: Local material used as a barrier soil liner**, in the comments field.

No Slope ☞ **No Slope**

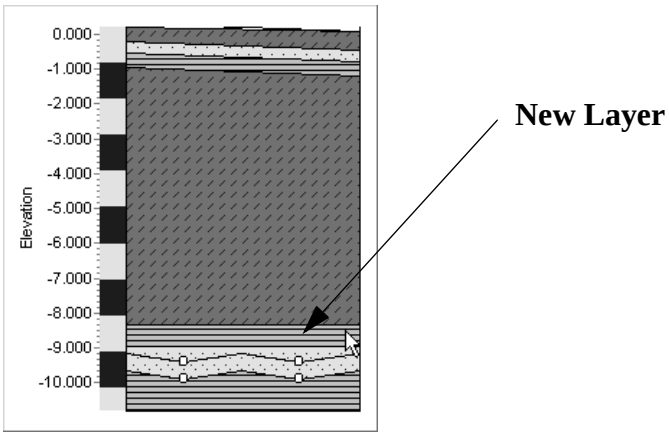
Type: 0.5 in the **Thickness** box.

Once completed, your **Profile Material Properties** dialogue box will appear as the figure below:



☞ **[OK]**

The new layer is now defined::



☞  icon, to save changes.

☞ **[OK],** to confirm the changes

The **Toronto Clay Loam** has now become part of the profile and you can modify it like the rest of profile layers.

To return to the original profile construction, which will be used for the rest of this tutorial, delete the recently inserted layer.

<right click> on **Toronto Clay Loam,** in the Project Tree View.

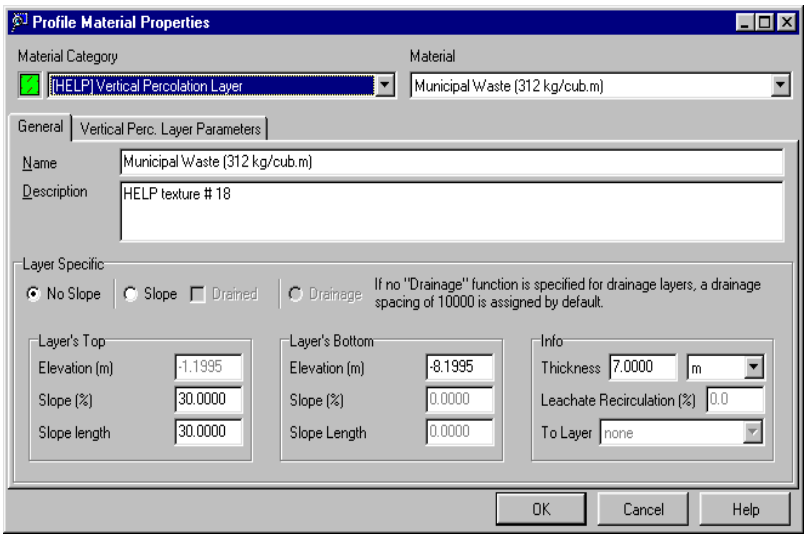
☞ **[Delete]**.

Editing Layer Parameters

The Visual HELP parameters of a landfill material depend on the material category.

☞ ☞ **Municipal Waste (312 kg/cub.m),** in the Project Tree.

The **Profile Material Properties** dialogue box will appear.



In this dialogue box, you can view and edit information about the layer including;

category, material texture, slope, top or bottom elevation, and thickness.

☞ **Vertical Perc. layer Parameters** tab to edit the material properties of particular layer.

☞ **Value** beside **total porosity**

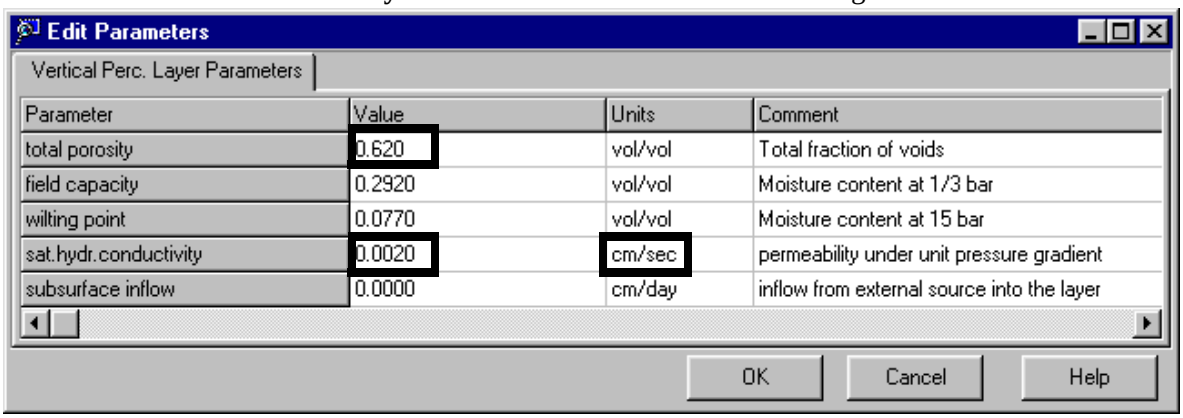
Type: 0.62

☞ **Units** beside the **sat. hydr. conductivity**, a drop-down list with available units will appear.

☞ **cm/sec**

Type: 0.002, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-] ,**beside **Municipal Waste (312 kg/cub.m)** in the Project Tree.

Now lets edit the properties of the Lateral Drainage Layer. Layers from this category have the same hydraulic parameters as Vertical Percolation Layers. In addition, the drainage and leachate recirculation parameters may be specified.

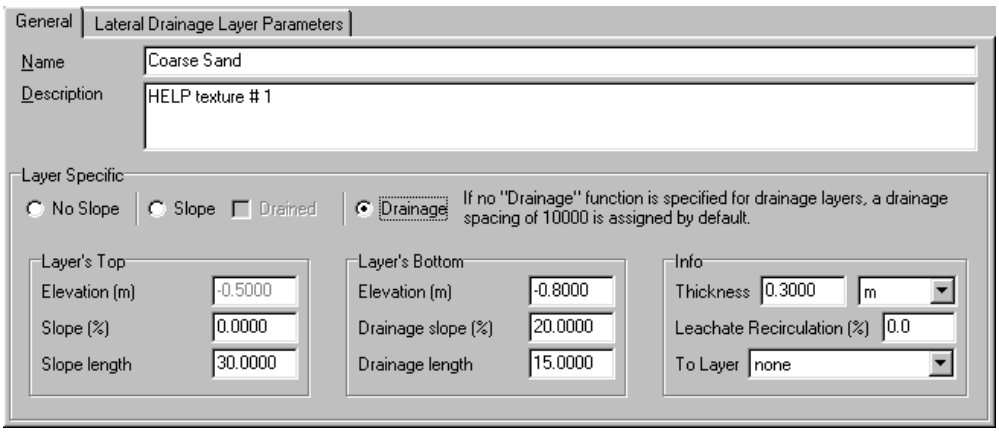
☞☞ **Coarse Sand** in the Project Tree. By default, the **Slope** function is selected for this layer.

Select **Drainage** by clicking the appropriate radial button. The name of parameter **Slope** in the **Layer's Bottom** box will change to **Drainage Slope** and the name of parameter **Slope Length** will change to **Drainage Slope**.

Type: 20 (%) for **Drainage Slope**

Type: 15 (m) for **Drainage Length**.

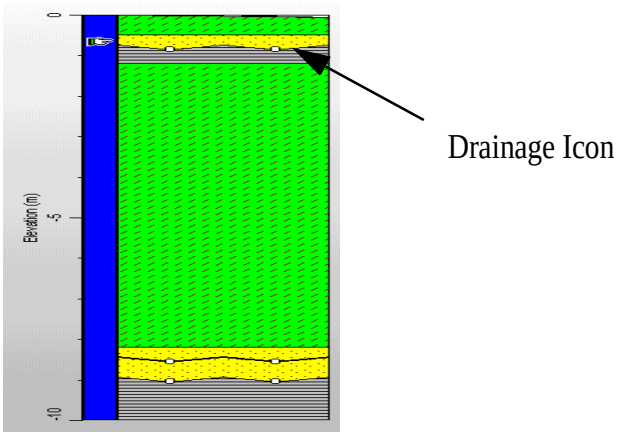
The **General** tab for **Coarse Sand** should be similar to the figure below.



☞ **[OK]**

☞ **[-],** beside **Coarse Sand** in the Project Tree.

The Profile View will change.



Finally, we will edit parameters of a Geomembrane Liner.

☞ **[+],** beside the **Low Density Polyethylene**

☞☞ **[Geomembrane Liner Parameters]**

☞ **Units,** beside the **sat. hydr. conductivity**

☞ **[cm/sec],** from the drop-down list.

Type: 7.00E-13, in the **Value** field.

☞ **[Units]**, beside the **pinhole density.**

☞ **[# / acre],** from the drop-down list.

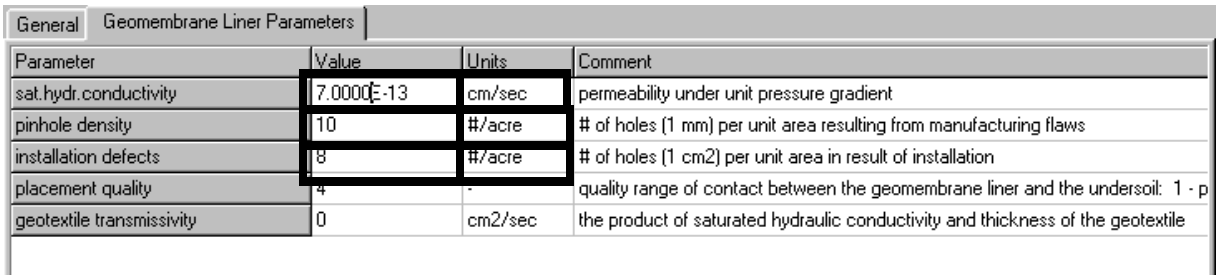
Type: 10, in the **Value** field.

☞ **Units,** beside the **installation defects.**

☞ **[# / acre],** from the drop-down list.

Type: 8, in the **Value** field.

The **Vertical Perc. layer Parameters** tab for LDPE liner will look like the figure below.



☞ **[OK]**

☞ **[-],** beside **Low Density Polyethylene** in the Project Tree.

With this action completed, we have finished editing the parameters of individual layers of our profile. However, there are more ways to do this, including the possibility of reassigning geometrical properties (slope, slope length) or drainage properties for a set of layers that perform a specific hydraulic function within a landfill (e.g. landfill cap).

Setting Initial Moisture Conditions & Runoff Parameters

Finally, we will now define our landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** parameter group located in the project tree..

As with the original HELP model, Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- *the model simulate values, or*
- *the user specify values.*

Depending on the selection made, runoff can be:

- *calculated by the model which will account for the type of material, slope, slope length and type of land cover,*
- *specified by the user,*
- *specified by the user and further adjusted to the slope and slope length.*

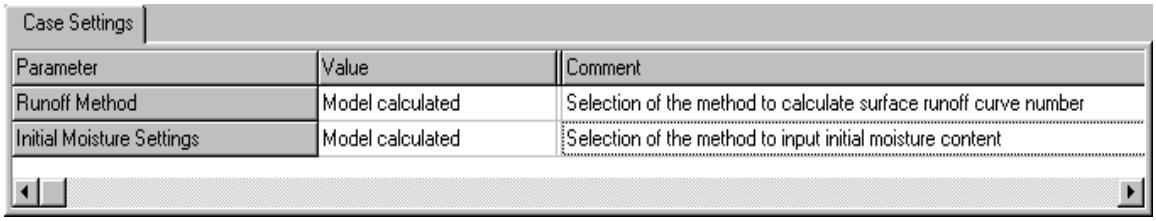
In the first case, which is the default, the model will assign realistic values for the initial moisture storage and simulate one year of landfill life. The values of moisture storage obtained from this simulation will be used as initial values.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. To set the automatic calculation,

☞☞ **Case Settings**, found in the Project Tree**.**

In the **Value** field beside **Runoff Method,** select **Model Calculated.** The program will automatically pick up the slope and slope length values for the top layer and use them in the runoff calculation.

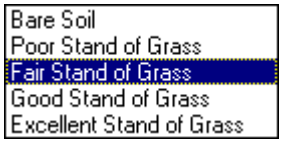
We will also leave the **Initial Moisture Settings** as **Model Calculated**



☞ **[OK]**

To specify additional parameters for surface runoff,

☞☞ **Surface Water Settings** from the Project Tree View.



To set the various vegetation classes, click in the **Value field**, found beside the **Vegetation Class.** The drop down list of available types of the land cover will appear.

☞ **Fair Stand of Grass**

☞ **[OK]**

Note: Surface Slope and Slope Length which regulate the runoff also, may be editied at the General Tab of the top profile layer.

You have now have completed creating your specific landfill profile. With a fully functional version of Visual HELP, you could save your work at this point however, as a demo, you will need to load a pre-run profile included with this demo. To continue,

☞ **File** from the main menu

☞ **Close Project**

☞ **[NO]** to close the project without saving your changes.

The next step is to generate your site specific weather using the USDA Weather Generating program that synthetically generates daily values for precipitation, temperature and solar radiation.

NOTE: Visual HELP has also an effective tool for generating aquifer recharge values that can be used in other models such as Visual MODFLOW. To calculate accurate recharge rates for your project, simply set your model profile to represent you site specific hydrogeological conditions and run!

Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting

We will now create site specific weather data that will be considered when modeling our landfill profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

☞☞ **HELP Tutorial 2** in the Project Tree. Note that two profiles (DEMO A & B) appear. **DEMO profile A** is a continuation fo the previous changes made in Module II.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill, and simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

For synthetic generation of daily values of precipitation, mean temperature, and solar radiation DOS HELP and Visual HELP version 1 included a Weather Generator developed by the Agricultural Research Service of the USDA (U.S. Department of Agriculture), as well as parameters for generating synthetic data for 139 U.S. cities. After Visual HELP 1.01 was released, WHI received requests from our clients worldwide to expand the area of Weather Generator application to other regions of the world.

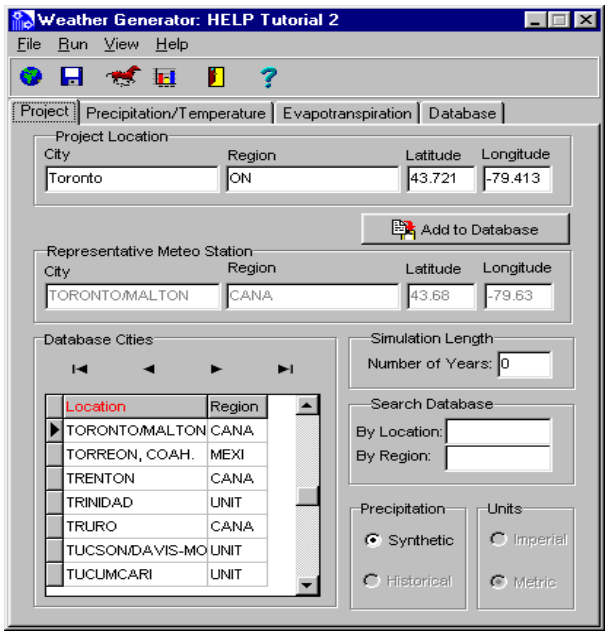
Trying to meet these requests, WHI developed a global database that includes more than 3000 stations and a GIS feature for searching the nearest stations globally for the Visual HELP version 2.1. If your landfill site is not located in our database, we recommend that you choose the closest city to your site and use the generated data for that location for your simulations. Visual HELP will search for the closest location of a given set of co-ordinates. This co-ordinate set can be entered as actual values or interactively with a map.

To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the main menu

☞ **Weather Generator**

After an introductory splash screen, the Weather Generator dialogue box will appear.



For this tutorial, you will be using data for the TORONTO/MALTON weather station. The TORONTO/MALTON station was selected among five stations located nearest to our landfill site.

Parameters for TORONTO/MALTON appear automatically in the text boxes throughout the **Weather Generator** dialogue boxes after it starts (if you have time, click **Precipitation/ Temperature, Evapotranspiration and Database** tabs to see the options to customize the weather parameters).

Let's generate weather data for our site for the next 20 years.

☞ **Number of Years** text box,

Type: 20, to represent the number of years you wish to build weather data for.

To run the Weather Generator,

☞  ,on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  ,to save the generated files

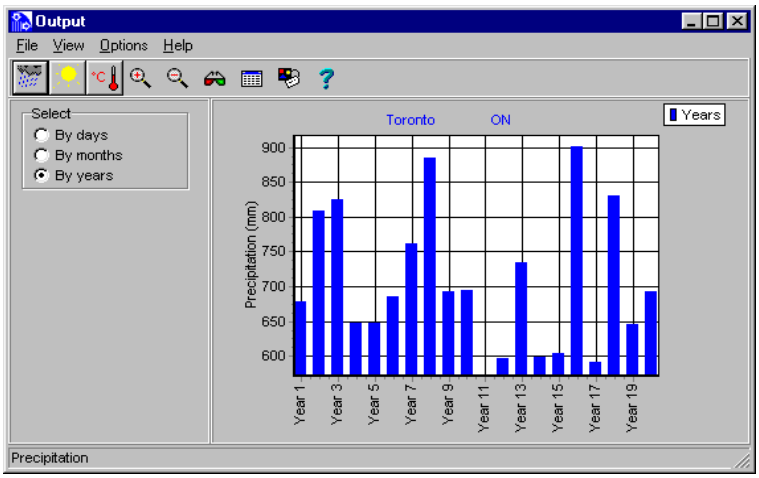
Viewing Weather Data

To view the results for the weather data you just generated,

☞  ,on the toolbar to view the **Output** dialogue box.

☞  ,on the toolbar to view generated precipitation values

By default annual totals will be presented, as shown below.

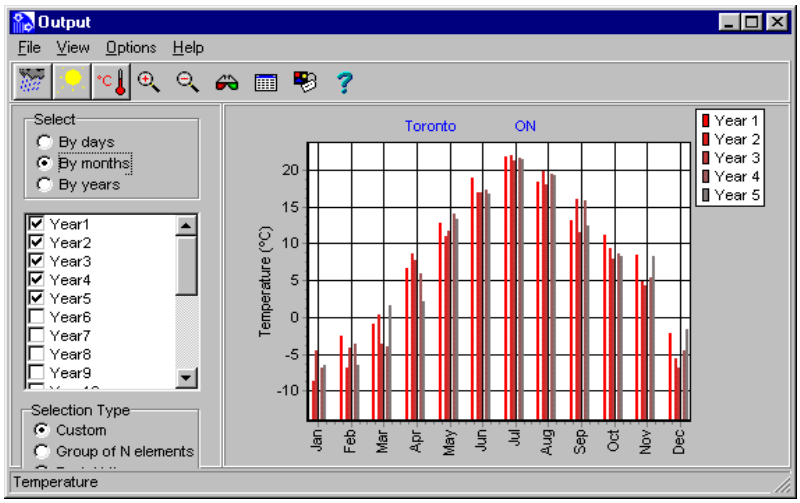


☞  ,to view the generated solar radiation data.

☞  ,to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  ,to view the generated raw data in table format.

☞  ,to close the table.

☞  ,to close the graphics.

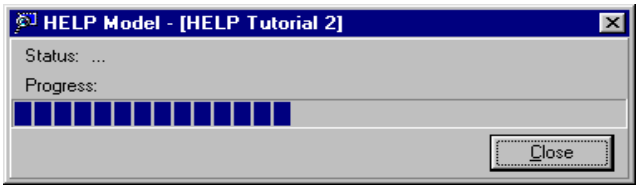
☞  ,to exit the Weather Generator.

You are now ready to run Visual HELP for 20 years using the landfill profile you created.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

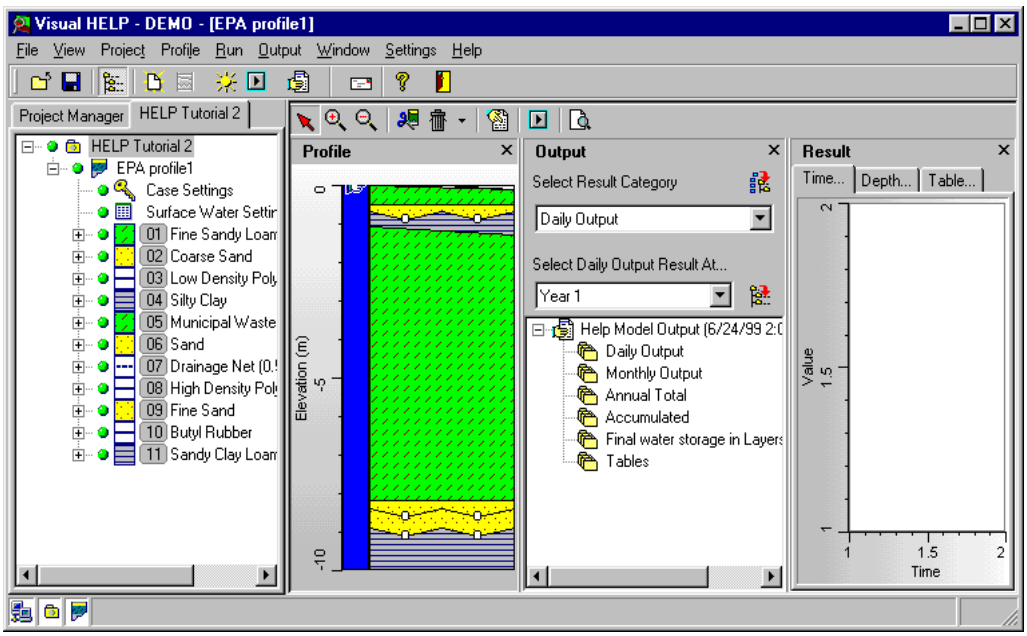


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

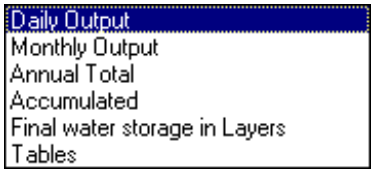
After the model has successfully run, Visual HELP will display the Output View and Result View windows.



☞  ,to enlarge the graphs viewing area

☞ **X** ,in the Profile View to close the window.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



You may select the category you wish to view however for this exercise, we recommend viewing your results with an **Accumulated** category. The option to graph **Accumulated Volumes** is a new feature, offered only in Visual HELP 2.1. This plot will show you the total volumes of water that were drained during the simulation period. This allows you to examine the total volume of leachate percolated through the landfill bottom during the simulated time and assess the total volumes of other water balance constituents.

☞ **Accumulated**

The results for the the **Accumulated** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox.

☞ **Select Accumulated Result at...** , to view the available balance types. ie) rate, volume

Rate means the annual rate of the balance constituent, and **volume** means the volume for the area represented by the profile.

☞ **rate**

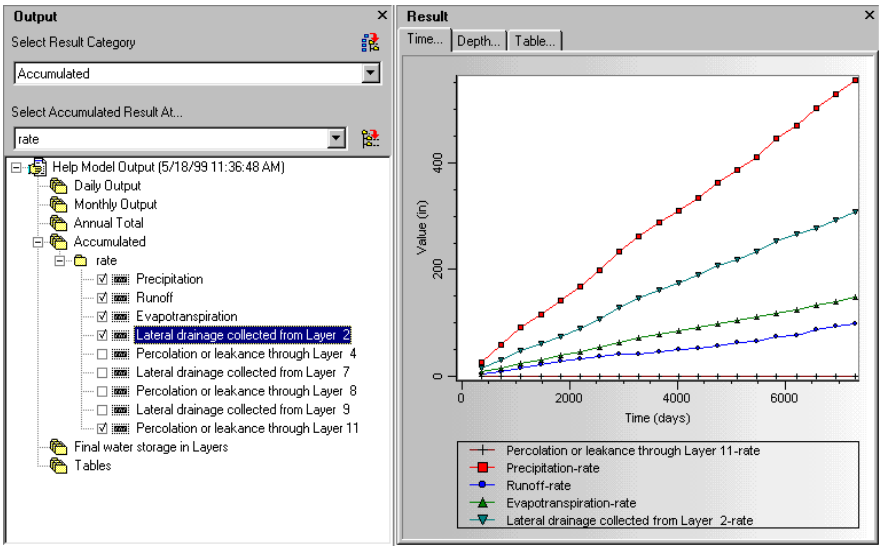


To view all the available results for this specific type of balance, click the icon to the right of the **Select Accumulated Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Lateral Drainage collected from Layer 2** (the first drainage layer), **Percolation or leakance through Layer 11** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



To erase a variable from the Result View window, deselect the corresponding check box in the Result Tree.

From the graph, one may conclude that Runoff, Evapotranspiration, and Lateral Drainage collected from Layer 2 play an important role in the landfill balance while Percolation through Layer 11 (the landfill bottom) is equal or close to 0. For specific values, the landfill balance constituents may be viewed in table format.

Viewing Tables

To view HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Accumulated rate**, from the **Select Table Result at..** lower listbox.

☞  **Add to Output Tree** icon to the right of the lower output listbox

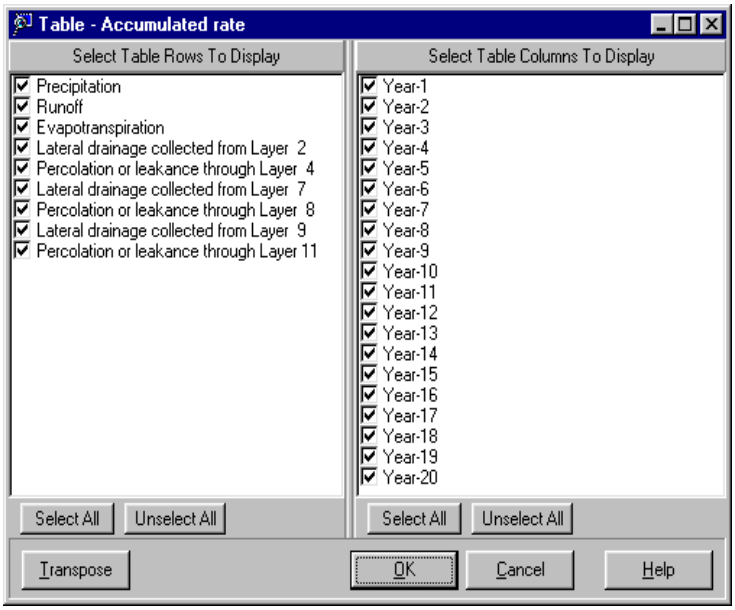
The table name **Accumulated rate** will appear in the Output Tree. For future use, let's place the table results into a new Result Window.

To do this, **<right click>** the name, **Accumulated rate** in the Output Tree. The following menu will appear:



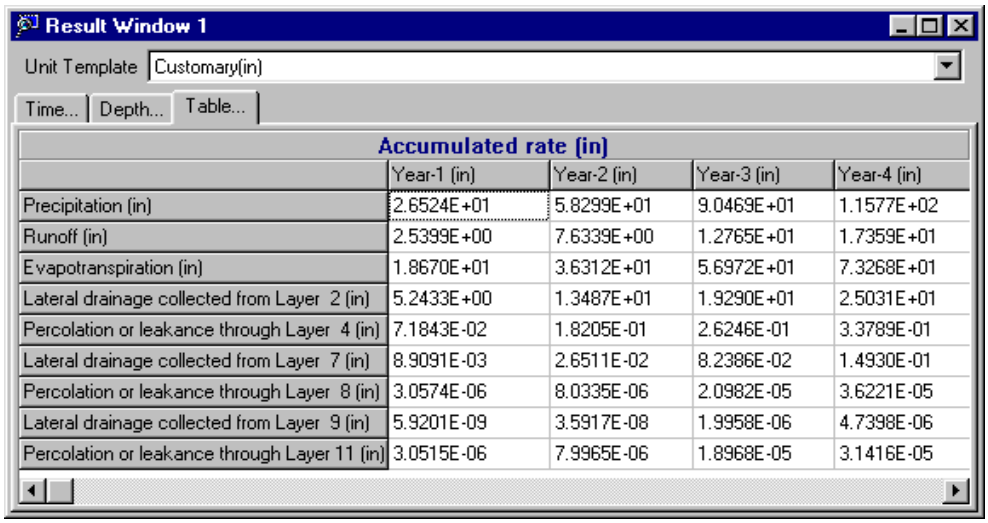
Choose **New Result Window**.

The New **Result Window 1** will open at the background and the table editing dialog box will appear.



☞ **[OK]**

The table will appear in **Result Window 1**:



Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation (in)	2.6524E+01	5.8299E+01	9.0469E+01	1.1577E+02
Runoff (in)	2.5399E+00	7.6339E+00	1.2765E+01	1.7359E+01
Evapotranspiration (in)	1.8670E+01	3.6312E+01	5.6972E+01	7.3268E+01
Lateral drainage collected from Layer 2 (in)	5.2433E+00	1.3487E+01	1.9290E+01	2.5031E+01
Percolation or leakance through Layer 4 (in)	7.1843E-02	1.8205E-01	2.6246E-01	3.3789E-01
Lateral drainage collected from Layer 7 (in)	8.9091E-03	2.6511E-02	8.2386E-02	1.4930E-01
Percolation or leakance through Layer 8 (in)	3.0574E-06	8.0335E-06	2.0982E-05	3.6221E-05
Lateral drainage collected from Layer 9 (in)	5.9201E-09	3.5917E-08	1.9958E-06	4.7398E-06
Percolation or leakance through Layer 11 (in)	3.0515E-06	7.9965E-06	1.8968E-05	3.1416E-05

Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation **553.98 in**

Runoff **74.84 in**

Evapotranspiration **359.14 in**

Lateral Drainage collected from Layer 2 **119.43 in**

Percolation or leakance through Layer 11 **0.00 in**

These values show that the default EPA landfill design which was slightly modified by you during the exercise, provides good ground water protection for our project location. Having made such an optimistic conclusion, lets prepare the Visual HELP report.

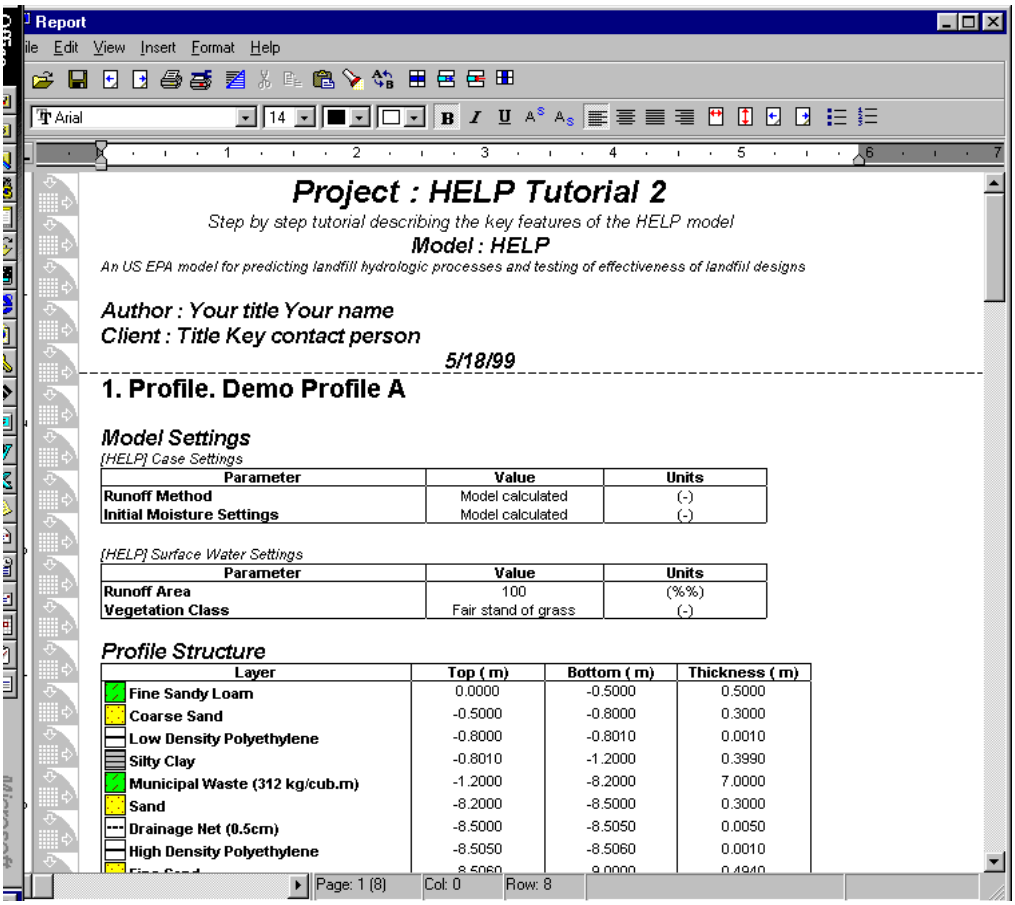
☞ , to close Result Window 1.

Creating a Report

As an added feature of Visual HELP, we have developed a time saving, report generator that will help you prepare profession reports of the model simulation.

To create a report and add the project input data, click the icon from the Operational Icons toolbar.

The Visual HELP Report Generator will display your report in a separate window.



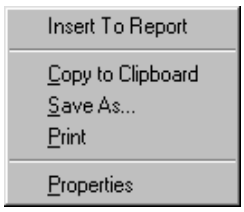
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add a graph form the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the graph in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The graph will be inserted into the report.

Go to the "Project" in the main menu and select **[view report]** to continue editing. To re-size the graph you just inserted, click the graph in the **Report** window and stretch it until it reaches the proper size.

To add a table from the **Result Window 1** to the report:

In the **Report** window place the cursor below the imported graph.

<right click> the table in the **Result Window 1**

3) ☞ **Insert To Report**. The table will appear in the report. Again, go to the "Project" in the main menu and select **[view report]** to continue editing or to view your report at any time.

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to , insert a headers /footers, change fonts/ letters size, etc..

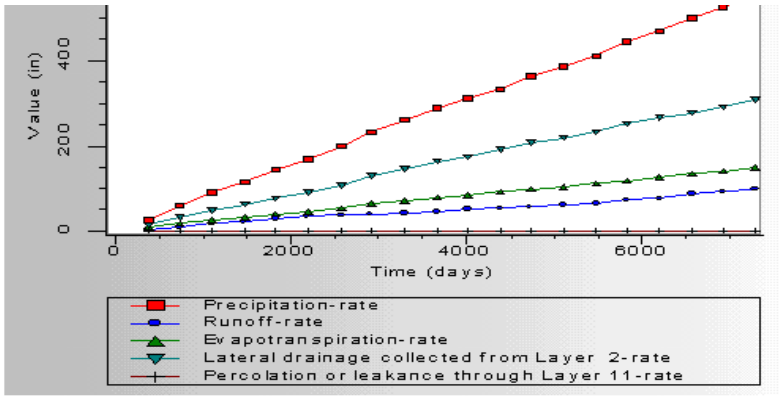


Fig.1 Accumulated balance constituents.

Table 1. Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation	2.6669E+01	5.8476E+01	9.0929E+01	1.1642E+02
Runoff	3.9219E+00	9.5287E+00	1.6193E+01	2.2387E+01
Evapotranspiration	8.2591E+00	1.5974E+01	2.4371E+01	3.1858E+01
Lateral drainage collected from Layer 2	1.4484E+01	3.1264E+01	4.8366E+01	6.2130E+01
Percolation or leakance through Layer 4	3.7569E-03	6.5516E-03	9.2846E-03	1.2289E-02
Lateral drainage collected from Layer 7	3.7532E-03	6.5417E-03	9.2717E-03	1.2271E-02
Percolation or leakance through Layer 8	3.7459E-06	8.9520E-06	1.2983E-05	1.7380E-05
Lateral drainage collected from Layer 9	3.5588E-06	8.4902E-06	1.2188E-05	1.6448E-05
Percolation or leakance through Layer 11	1.8707E-07	4.6209E-07	7.9524E-07	9.3185E-07

Once you are satisfied with your report, you may print the report and/or save it for future use. Let's take a look at additional output features available with Visual HELP.

Peak Daily Values

To assess the landfill hydrologic performance in extreme conditions, it is important to know when the peak hydrologic events occured. The **Peak Daily Values** table picks up the dates of such events.

☞ **[Tables]** from the **Select Result Category** drop-down listbox.

☞ **[Peak daily values]** from the **Select Table Result at..**

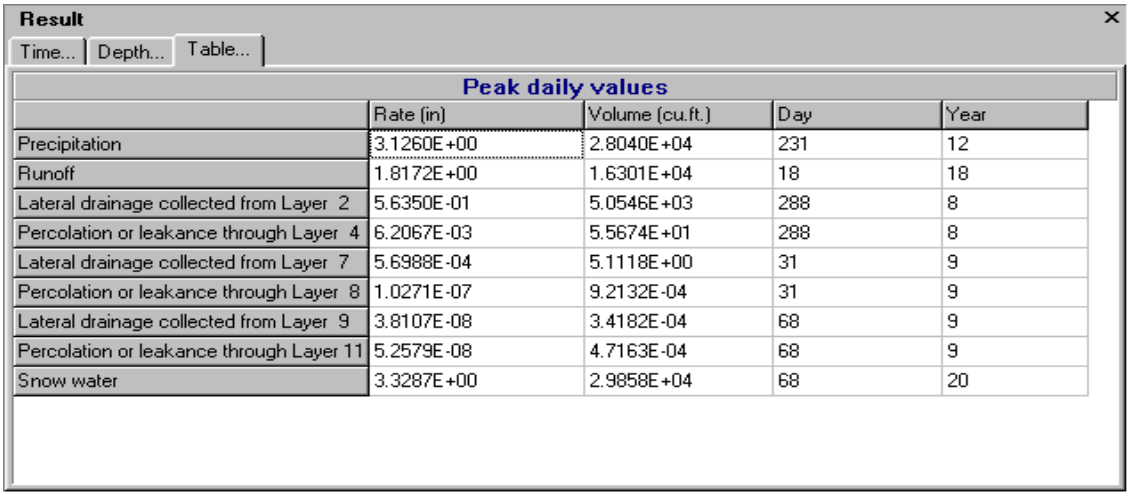


☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name "**Peak daily values**" will appear in the Output Tree. Click the check box beside it.

☞ **[OK]** in the table editing dialog box.

The following table will appear:



Peak daily values

	Rate (in)	Volume (cu.ft.)	Day	Year
Precipitation	3.1260E+00	2.8040E+04	231	12
Runoff	1.8172E+00	1.6301E+04	18	18
Lateral drainage collected from Layer 2	5.6350E-01	5.0546E+03	288	8
Percolation or leakance through Layer 4	6.2067E-03	5.5674E+01	288	8
Lateral drainage collected from Layer 7	5.6988E-04	5.1118E+00	31	9
Percolation or leakance through Layer 8	1.0271E-07	9.2132E-04	31	9
Lateral drainage collected from Layer 9	3.8107E-08	3.4182E-04	68	9
Percolation or leakance through Layer 11	5.2579E-08	4.7163E-04	68	9
Snow water	3.3287E+00	2.9858E+04	68	20

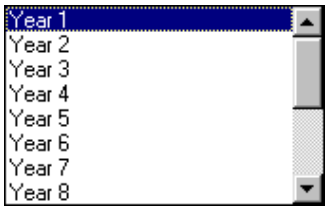
You may conclude that no leakage occurred through the bottom of the landfill (Peak daily percolation/leakage through layer 11). You may also find that the peak precipitation (3.12 in) happened on day 231 of year 12 and that peak runoff (1.81 in) happened on day 18 year 18.

To learn more about the circumstances of these events, Daily Output graphs may be studied.

Daily Output

☞ **[Daily Output]** from the **Select Result Category** drop-down listbox.

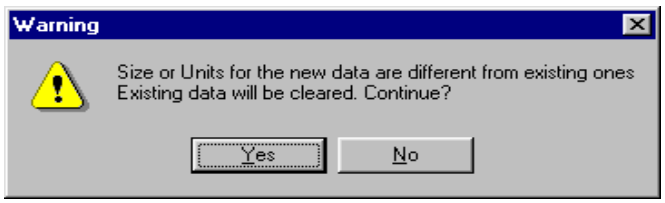
Next select the year for which you would like to have the results to be displayed from in the listbox.





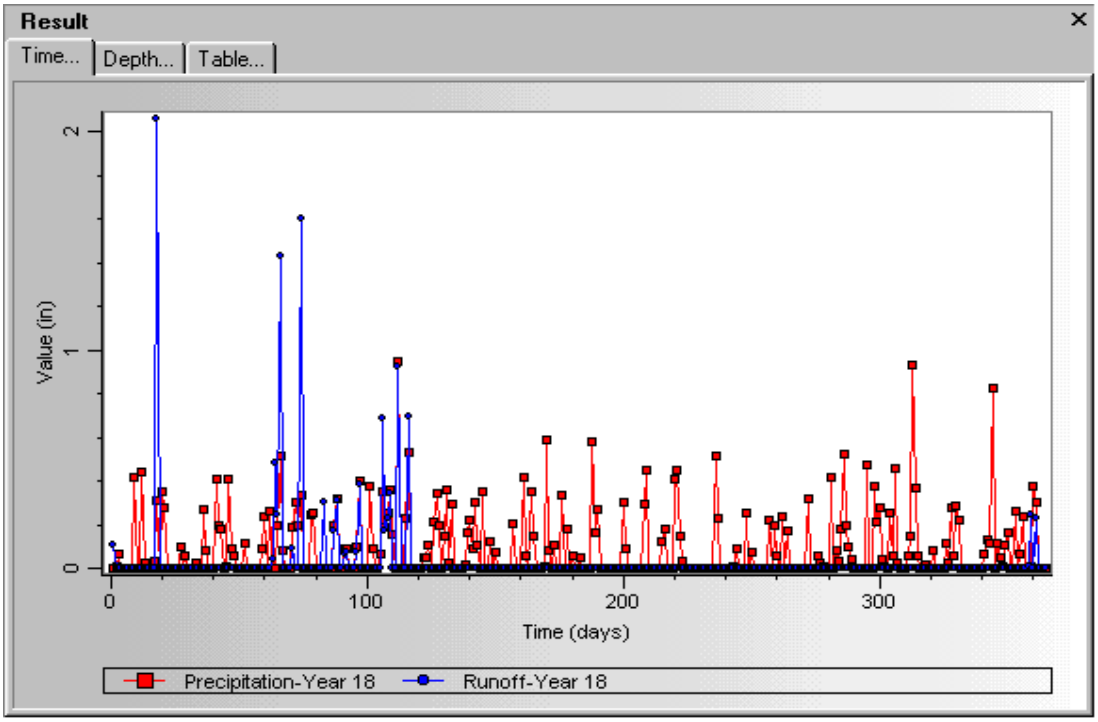
Scroll the list of years, and select the desired year: **18**. To view the available results for this specific type of balance, click the **Add to Output Tree** icon to the right of the lower output listbox.

The tree of available variables for year 18 will appear. Select **Precipitation** and **Runoff** by clicking the appropriate check boxes. This means that you are going to place these graphs to the Main Result Window. A warning will appear:



☞ **[Yes]**

The graph for the daily values of these variables will appear:



This graph shows that the maximum runoff event was not preceded by very intensive precipitation. However, after reviewing the weather data, it appears that day 18 was the first warm day with an average temperature of 5 degrees C (41 Fahrenheit) after a long span of frosty days. The peak runoff was caused by the intensive thaw of snow accumulated during the previous cold period.

Click to close the graph.

This concludes the Visual HELP demonstration exercise. Close Visual HELP by clicking in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers. If you have time, we would encourage that you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us.

Visual HELP Lab Exercise: Recharge

Estimating Groundwater Recharge with Visual HELP

The HELP Model

The **HELP** model (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile program used to design, evaluate and optimize landfill hydrology, and estimate groundwater recharge rates. The HELP model is used and recognized all over the world as the accepted standard for modeling landfill hydrology, and has become an integral component for projects involving landfill operating and closure permits.

The HELP model is a quasi-two-dimensional, multi-layer hydrologic model requiring the following input data for each model profile:

- Weather data (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil properties (porosity, field capacity, wilting point, and hydraulic conductivity)
- Design information (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can consist of a combination of natural (soil) and artificial materials (waste, geomembranes) with options to install horizontal drainage layers. The HELP model also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

Learning Objectives

After completion of this exercise you should be able to…

- Apply Visual HELP to obtain an estimate of a groundwater recharge rate from precipitation through the unsaturated zone;
- Understand the basic principals and limitations of the water balance method for estimating recharge from precipitation;
- Apply some of the many useful features of Visual HELP and the Weather Generator utility to actual projects.

Introduction

Visual HELP can be used to calculate a groundwater recharge rate from precipitation (rain or snow, or both) under the influence of site-specific soil properties, solar radiation, air temperature, wind speed, and vegetative properties. The HELP model (Schroeder, et al., 1994) predicts an infiltration rate under quasi two-dimensional influences through one or more uniform soil layers. The HELP model (Schroeder, et al., 1994) has been applied by contemporary groundwater practitioners to substantiate values of net recharge from precipitation obtained from historical

studies for use in groundwater flow and contaminant transport models (see Abraham, B., et al., USAE Waterways Experimental Station, 1998).

In this exercise you will use the HELP model (Schroeder, et al., 1994) in Visual HELP to calculate the potential net recharge to groundwater from precipitation under site-specific soil, vegetative, and climatic conditions for a fictitious site near San Francisco, California using the water balance method. The water balance method calculates potential net recharge as a residual of all other water balance components and is expressed as:

$$R = P - D - ET_a - \Delta W \qquad (1)$$

where,

R is potential net recharge, P is precipitation, D is net runoff, ET_a is actual evapotranspiration, and ΔW is the change in soil water storage.

Limitations of the Water Balance Method

In arid and semi-arid regions application of the water balance method is more difficult to apply than in humid regions because precipitation in these regions is only slightly larger than evapotranspiration. Therefore, small measurement errors in either of these two components of the water balance equation can cause large errors in recharge rate estimates (Simmers, I., et al., 1998; Lerner, D. et al., 1990). Care should therefore be taken when applying the water balance method in water scarce regions, and should be used to calculate recharge events for one- to ten-day intervals to detect ephemeral recharge events common to these regions.

Note: All theoretical components of this tutorial are italicized.

Terms and Notations

For this tutorial, the following terms and notations will be used:

type	- type in the given word or value
☞ , select	- click the left mouse button where indicated
√	- press the <Tab> key
↵	- press the <Enter> key
+	- click the left mouse button where indicated
☞☞	- double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click or values to type in.

[….**]** denotes a button to click, either in a window, or in the side or bottom menu bars.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Background Information

Based on laboratory analysis of numerous soil samples collected at the site from split spoon and Shelby tube sampling, test pit soil profile descriptions, and a comprehensive vegetation survey, the following characterization of the unsaturated soil profile and vegetative cover was made:

1. The unsaturated zone exists to a depth of 10 feet, where the water table is encountered;
2. The ground surface is covered with an excellent stand of mature grass;
3. The root zone extends to a depth of 12 inches;
4. The soil is classified as a fine sandy loam;
5. Surface slope is 12 percent;
6. Leaf area index is 4;
7. Total Porosity: 0.48 (vol/vol)
8. Field Capacity: 0.25 (vol/vol)
9. Wilting point: 012 (vol/vol)
10. Initial soil water content: 0.28 (vol/vol)
11. Saturated hydraulic conductivity: 5.0 x 10^{-4} (cm/sec)

Starting Visual HELP

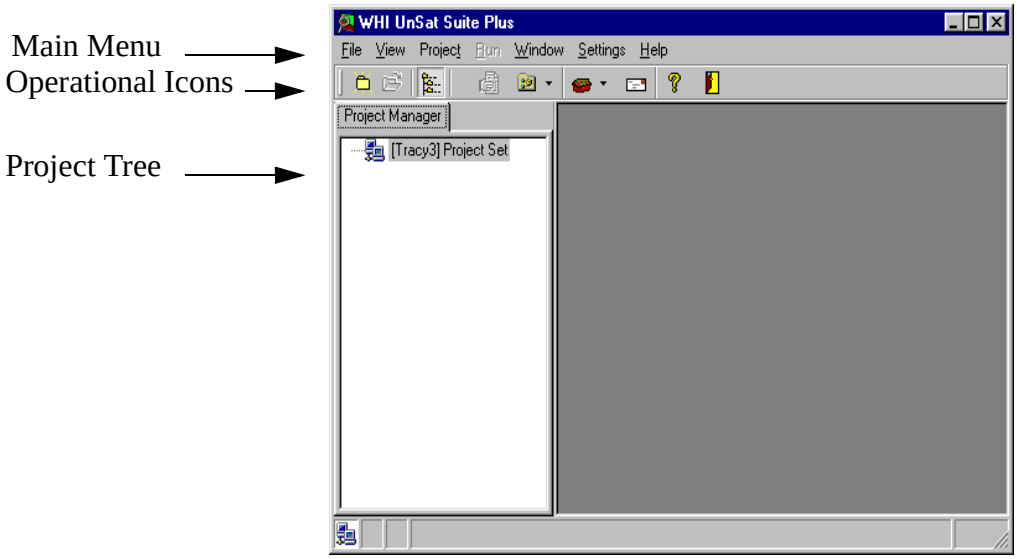
Visual HELP is a component of the WHI UnSat Suite Plus package. To start the WHI UnSat Suite Plus, double click the following icon on your desktop.



An introductory screen will be displayed when you open WHI Visual HELP/UnSat Suite



After the program is loaded, the main WHI Visual HELP/UnSat Suite Plus window will appear on your display:



The WHI UnSat Suite Plus interface structure has been designed to help you navigate through the program with ease. The main menu is located at the top, similar to our other programs. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the various ongoing projects, and their specific structure.

Creating a New Project

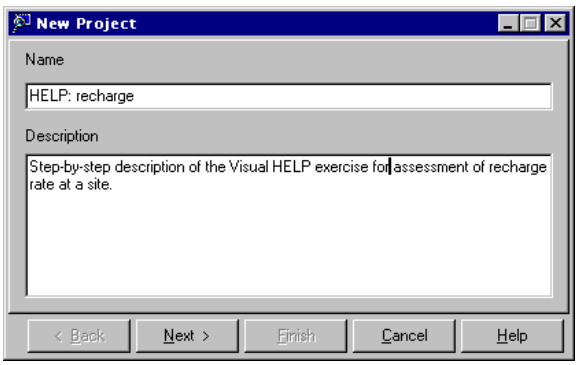
To create a new project, go the **File Menu**,

☞ **New Project**, the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to setup a new Visual HELP project.

In the **Name** box type: **HELP: recharge**.

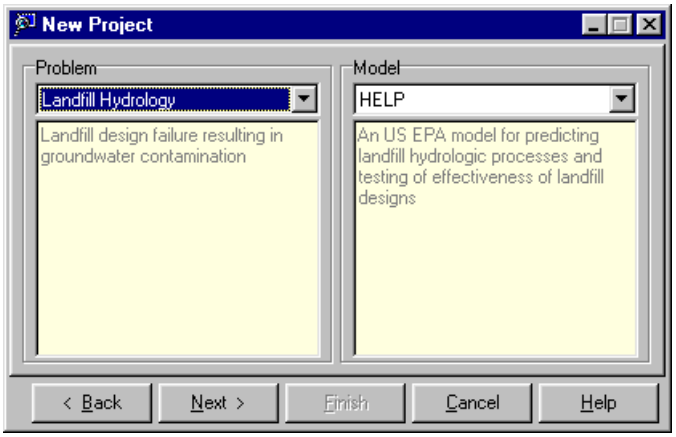
In the Description box type: **Step-by-step description of the Visual HELP exercise for assessment of recharge rate at a site.**



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on.

☞ **[Problem]** dropdown menu, select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Problem class that should be used with the selected Problem type.



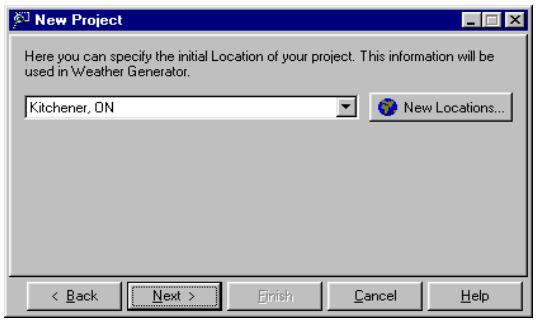
☞ **[Next]**

Once completed, the site location must be defined.

Selecting the Location

We will now select the location of our project using the GIS - WHI Locator.

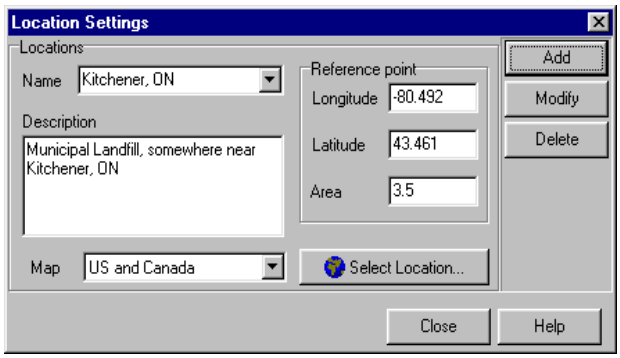
The following dialog box should already appear on your screen.



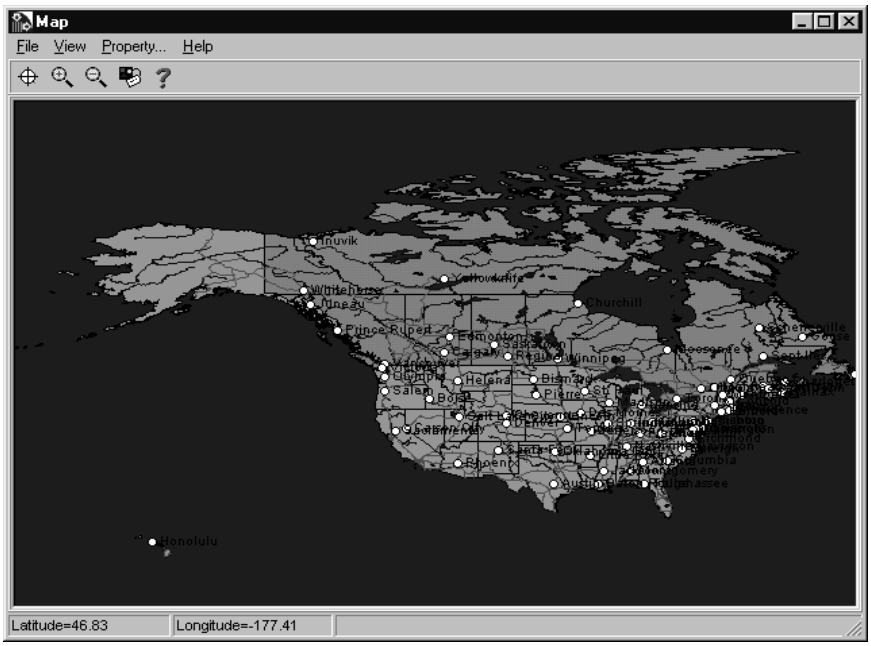
At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



☞ **[Select Location..]** button**,** to enable the Site Locator, developed by WHI. As shown below, a map of USA, Canada, and Mexico appears on your display:



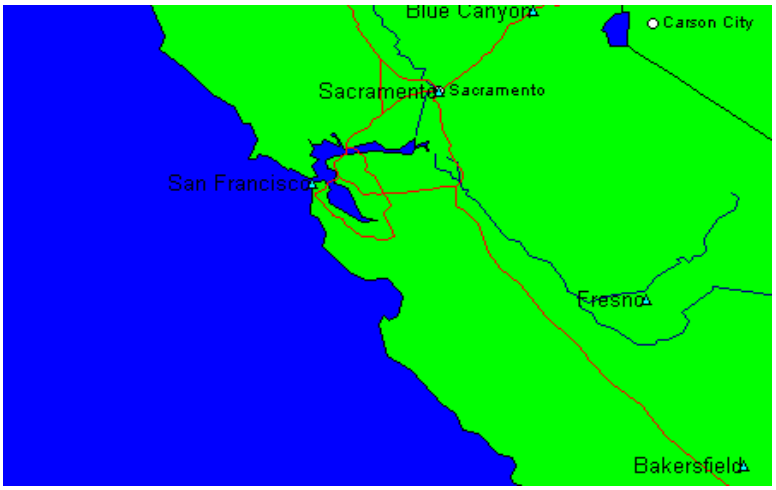
Now lets select San Francisco as your site location.

☞  , to zoom into our location.

Move the mouse to the right of the region where the landfill project is located (the West Coast). The coordinates, seen in the bottom left of the screen, should read approximately Latitude: 41, Longitude:-127. Press the left mouse button and stretch a zoom rectangle to the bottom and right,

 Copyright 2000 Waterloo Hydrogeologic, Inc.

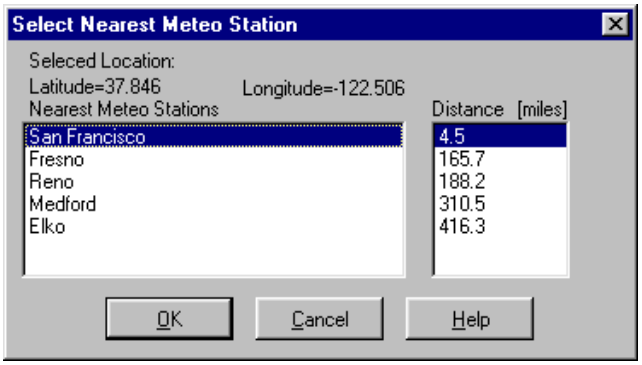
release the button (approximate Latitude: 33, Longitude:-115). After one or two zooms, the figure below should resemble your screen.



Next you will select San Francisco to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

Move the crosshair to the spot on the map where San Francisco is located and click to the north of it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you click directly on the city, the location will be forwarded to the database automatically. No nearest stations will show up.

In the "Select Nearest Meteo Station" dialog box,

☞ **[San Francisco],** as the nearest weather station.

☞ **[OK]**

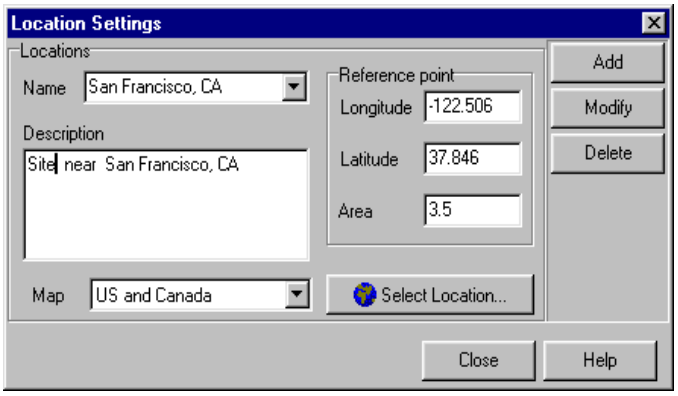
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: San Francisco, CA in the **Name** box.

Type: Site near San Francisco, in the **Description** text box.

Type: 1 , in the landfill area text box.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database.

☞ **[Close]**

While back in the **New Project**, select **San Francisco, CA** from the drop-down list.

☞ **[Next]**

Selecting Units

The next step is to define the units. This is completed in the "Select Input/Output Unit Templates" dialog box which allows you to define the units for both input and output. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.

The Visual HELP Unit Converter has been developed to maximize the flexibility for data input and output unit conversion. To show you the range of possibilities, just view the list of available units for hydraulic conductivity:



For this demo exercise we will specify that all input units will be in **Customary**, and output will be in **Customary (in)** units. The difference between the two unit templates is that Length in **Customary** template is set in feet while in **Customary (in)** it is set in inches.

From the **Input** drop-down list,

☞ **Customary**

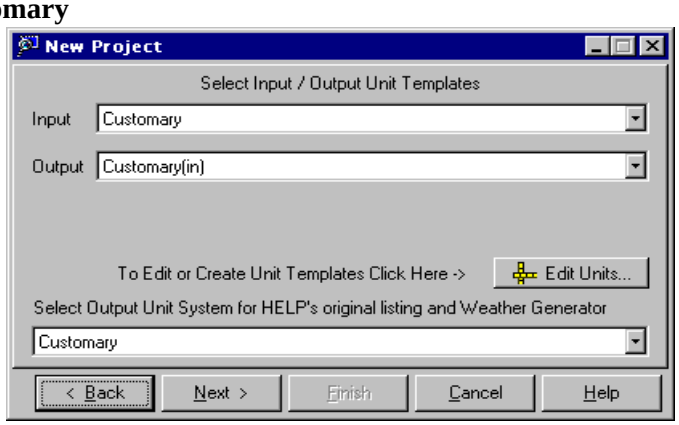
From the **Output** drop-down list

☞ **Customary (in)**

From the **Select Output Unit System for original listing and Weather Generator** list box,

 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **Customary**



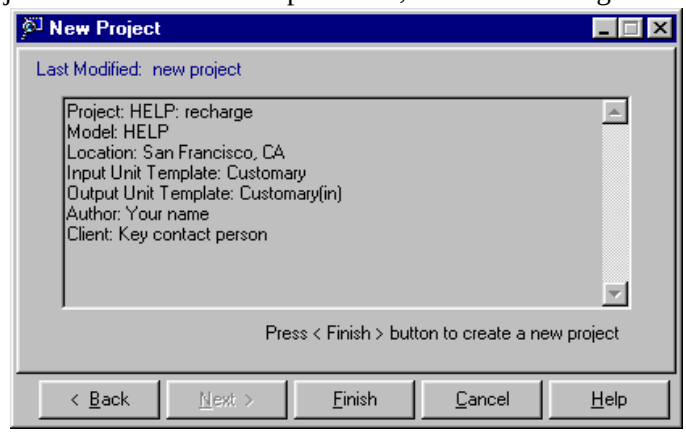
☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

All the project information will be presented, as seen in the figure below:



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings are specified, the **New Profile Wizard** will open.

Creating the Soil Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:



In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the later option.

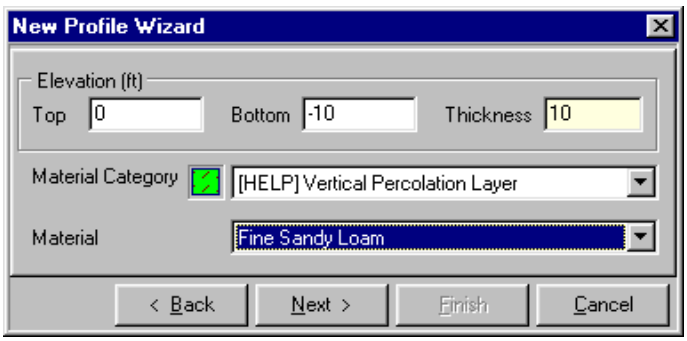
☞ **[create new profile]**

☞ **[Next]**

The dialog box with default profile parameters will appear. In this dialog box you may assign the parameters of the simulated profile. In your case the default profile thickness matches your profile characteristic.

☞ the drop-down arrow in the **Material** drop-down list box.

Select: **Fine Sandy Loam**. As a result of editing, the **New Profile Wizard** window will appear as shown below:



☞ **[Next]**

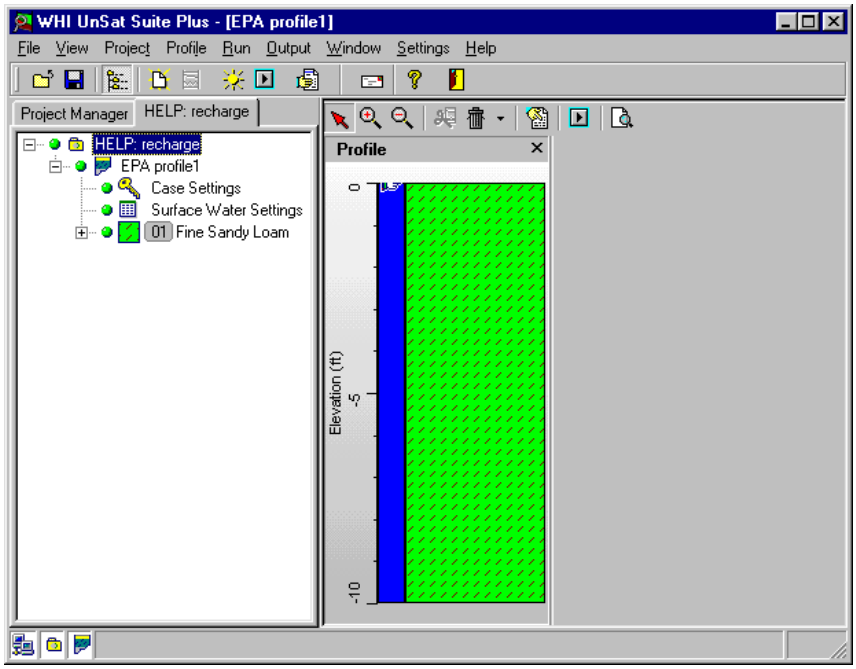
All the profile information will be presented:



☞ **[Finish]** to load the profile.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The new project details will be added to the project tree. The following picture will appear on your display:



This concludes creating a new project using Visual HELP

Editing Layer Parameters

To edit parameters of the Fine Sandy Loam layer,

☞ ☞ **Fine Sandy Loam,** in the Project Tree.

The **Profile Material Properties** dialog box will appear. In this dialog box, you can view and edit information about the layer including:

category, material texture, slope, top or bottom elevation, and thickness.

Under **Layer Specific**,

☞ **Slope** option button to enable surface runoff simulation.

Under **Layer Top**,

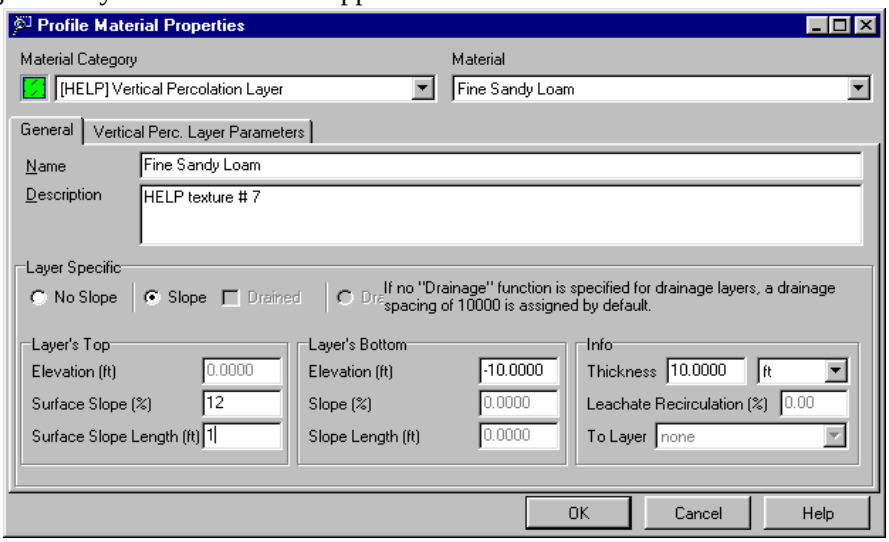
In the **Surface Slope (%)** text box:

type: **12**

In the **Surface Slope Length (ft)** text box:

type: **1** (to set unit length of slope for this exercise)

Finally, the dialog box on your screen should appear as shown below:



☞ **Vertical Perc. layer Parameters** tab to edit the material properties of a particular layer.

☞ **Value** beside **total porosity**

Type: 0.48

☞ **Value** beside **field capacity**

Type: 0.25

☞ **Value** beside **wilting point**

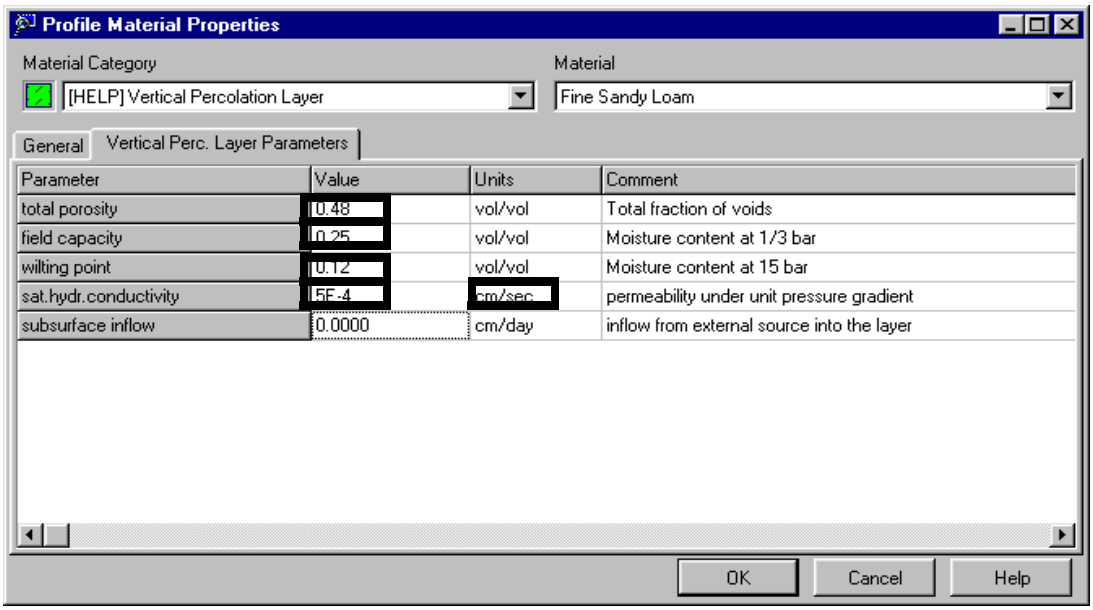
Type: 0.12

☞ **Units** beside the **sat. hydr. conductivity**, a drop-down list with available units will appear.

☞ **cm/sec**

Type: 5E-4, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



 Copyright 2000 Waterloo Hydrogeologic, Inc.

☞ **[OK]**

☞ **[-] ,**beside **Fine Sandy Loam** in the Project Tree.

Setting Initial Moisture Conditions & Runoff Parameters

Finally, you will define your landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** and **Surface Water Settings** parameter groups located in the project tree.

Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- the model simulated values, or
- the user specified values.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. These settings, accessible through the **Case Settings** parameter group in the Project Tree are default and do not require your intervention.

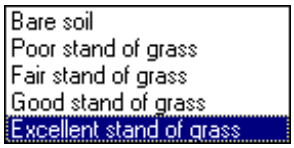
To edit additional parameters for the surface runoff simulation:

☞☞ **Surface Water Settings** from the Project Tree View.

To set the vegetation class,

☞ in the **Value** field, found beside the **Vegetation Class.** The drop down list of available types of the land cover will appear.

☞ **Excellent Stand of Grass:**



☞ **[OK]**

This completes the data preparation process. With a fully functional version of Visual HELP you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **HELP Recharge tutorial 1**, which was specifically developed for you and contains all changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

☞ ☞ project **HELP Recharge tutorial 1.**

You will now proceed with generating the required weather data and, subsequently, running Visual HELP.

Generating Weather Data

We will now create site specific weather data that will be considered when modeling your profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill, and simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

For synthetic generation of daily values of precipitation, mean temperature, and solar radiation DOS HELP and Visual HELP version 1 included a Weather Generator developed by the Agricultural Research Service of the USDA (U.S. Department of Agriculture), as well as parameters for generating synthetic data for 139 U.S. cities. After Visual HELP 1.01 was released, WHI received requests from our clients worldwide to expand the area of Weather Generator application to other regions of the world.

Trying to meet these requests, WHI developed a global database that includes more than 3000 stations and a GIS feature for searching the nearest stations globally for the Visual HELP version 2.1. If your landfill site is not located in our database, we recommend that you choose the closest city to your site and use the generated data for that location for your simulations. Visual HELP will search for the closest location of a given set of co-ordinates. This co-ordinate set can be entered as actual values or interactively with a map.

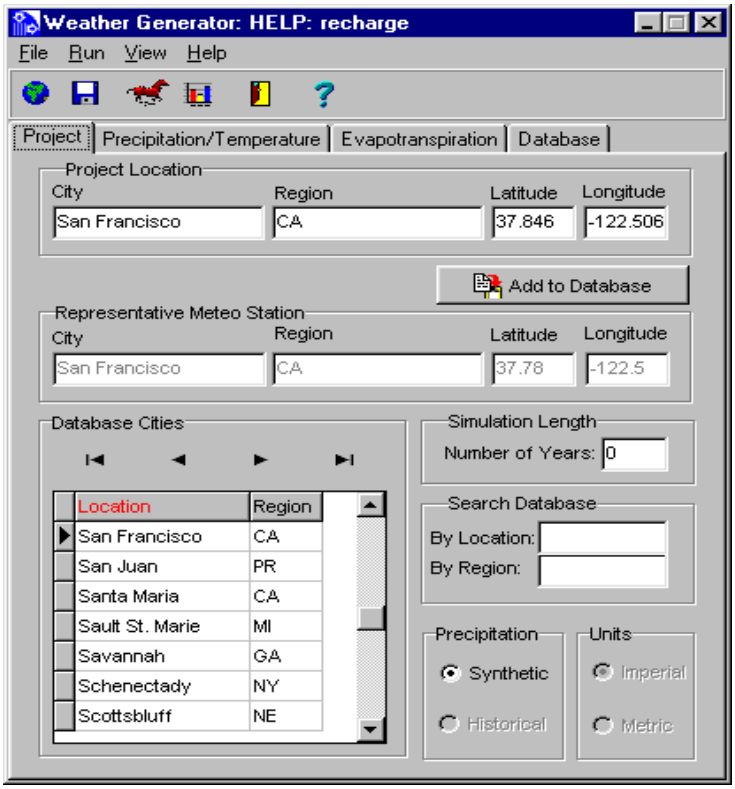
To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the main menu

☞ **Weather Generator**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

After an introductory splash screen, the Weather Generator dialog box will appear.



For this tutorial, you will be using data for the San Francisco weather station.

Parameters for San Francisco appear automatically in the text boxes throughout the **Weather Generator** dialog boxes after it starts (if you have time, click **Precipitation/Temperature and Database** tabs to see the options to customize the weather parameters).

According to the exercise scenario, you have to edit values for evaporative zone depth and leaf area index.

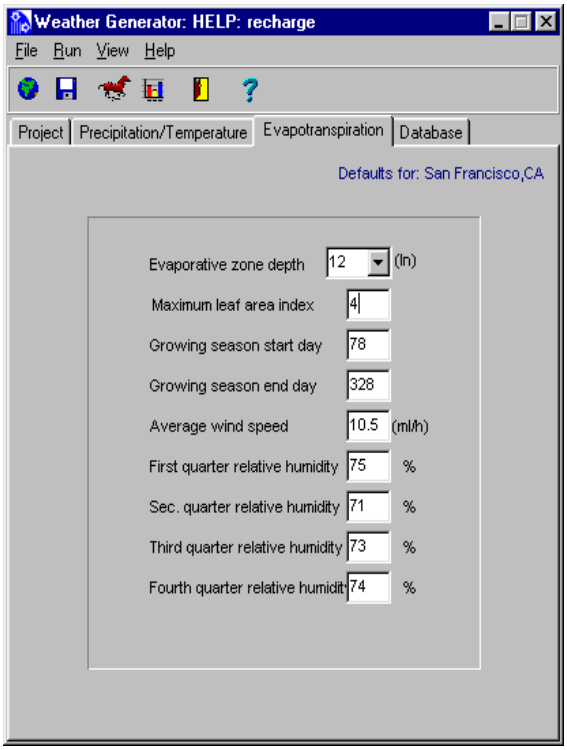
☞ **Evapotranspiration** tab.

In the text box for **Evaporative zone depth**:

type: 12

In the text box for **Maximum leaf area index**:

type: **4**



Let's generate weather data for our site for a 100 year period.

☞ **Project** tab

☞ **Number of Years** text box,

Type: 100, to represent the number of years you wish to build daily weather data for.

To run the Weather Generator,

☞  ,on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  ,to save the generated files

Viewing Weather Data

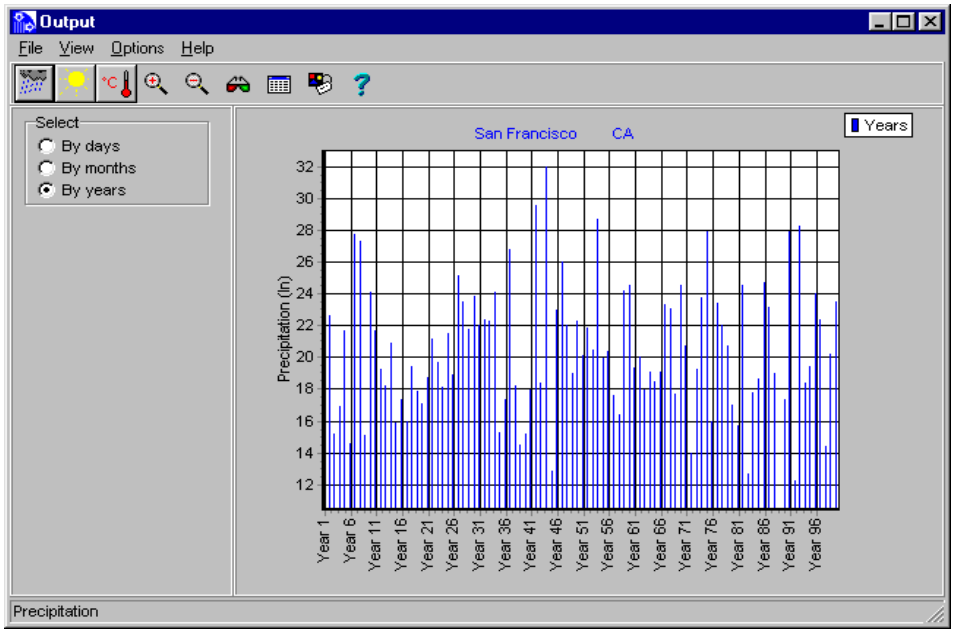
To view the results for the weather data you just generated,

☞  ,on the toolbar to view the **Output** dialog box.

☞  ,on the toolbar to view generated precipitation values

 Copyright 2000 Waterloo Hydrogeologic, Inc.

By default annual totals will be presented, as shown below.

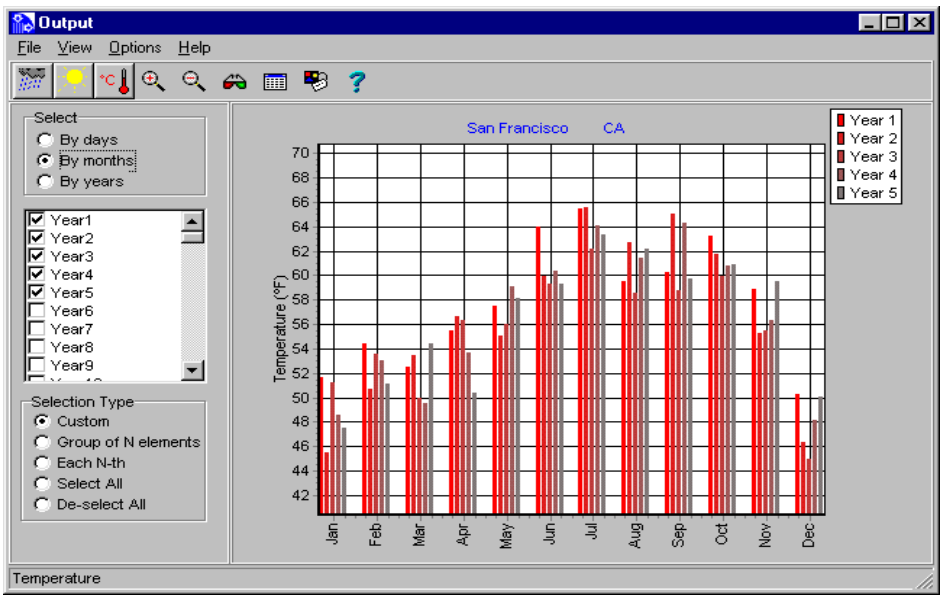


☞  ,to view the generated solar radiation data.

☞  ,to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  ,to view the generated raw data in table format.

☞  ,to close the table.

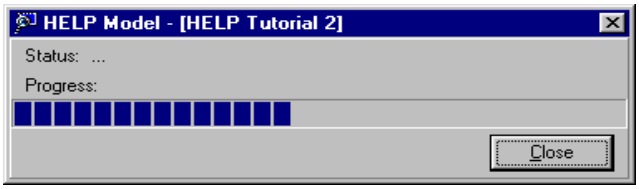
☞  ,to close the graphics.

☞  ,to exit the Weather Generator.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

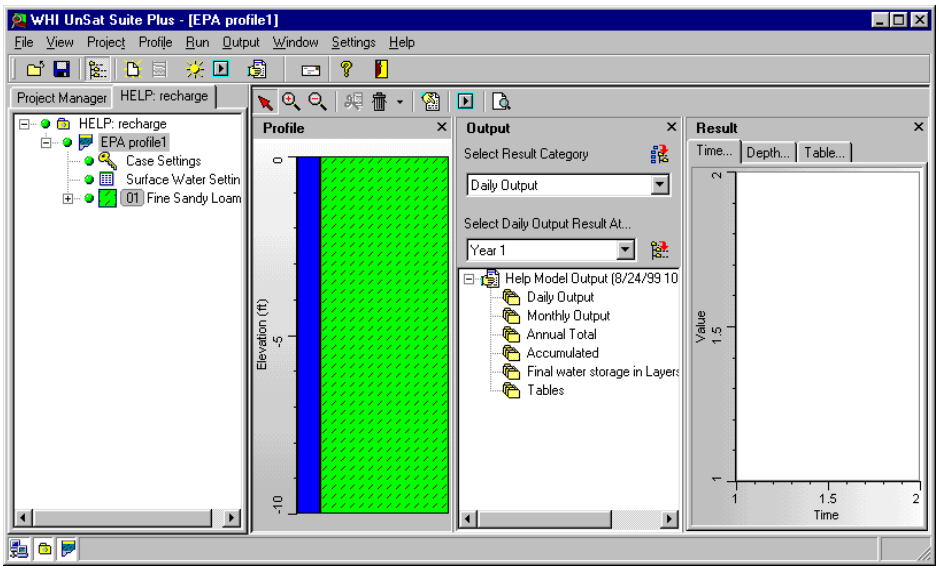


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

After the model has successfully run, Visual HELP will display the Output View and Result View windows.

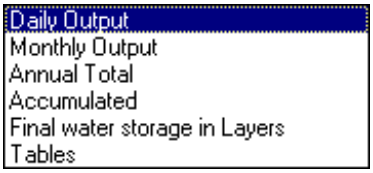


☞  ,to enlarge the graphs viewing area

☞ **X** ,in the Profile View to close the window.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



You may select the category you wish to view, however for this exercise, we recommend viewing your results with an **Annual Total** category. This plot will show you the annual amounts of water that were drained during the simulation period and assess volumes of other water balance constituents.

☞ **Annual Total**

The result for the **Annual Total** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox or

☞ **Select Annual Total Result at...** You will see: rate, volume, percent

Rate means the annual rate of the balance constituent, **volume** means the volume for the area represented by the profile, and **percent** means the percent of the amount of water entering the soil surface.

☞ **rate**

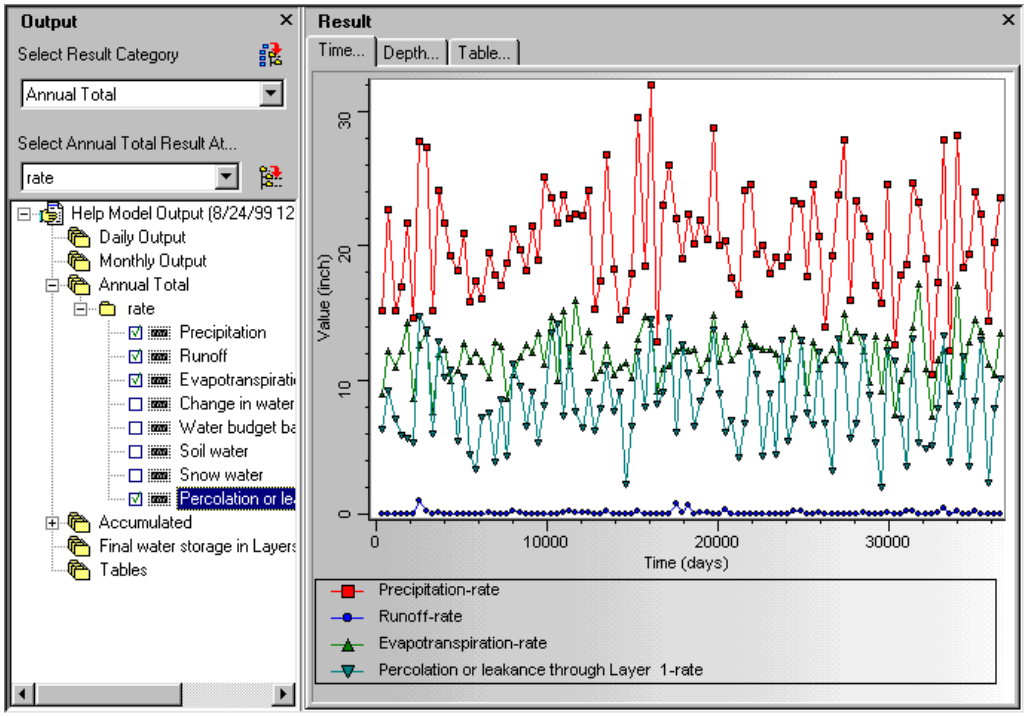


To view all the available results for this specific type of balance, click the icon to the right of the **Select Annual Total Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



This plot illustrates the following:

- At the end of first year, approximately 6.3 inches of recharge (labeled as percolation in the time series plot) is predicted to reach the groundwater.
- Peak recharge is seen to occur in year 7 of the simulation with an annual recharge rate of approximately 14.8 inches for the year.
- Annual recharge can be as low as 2.0 in. (year 81).
- A strong correlation exists between the amount of precipitation and recharge. This can be seen in the similar shapes of the precipitation and recharge plots over the 100-year simulation.
- The level of evapotranspiration activity affects the recharge rate. The recharge rate is higher when periods of high precipitation and low evapotranspiration activity correspond.
- Surface runoff does not play an important role in the profile water balance.

To get information about the principal proportion of the profile water balance constituents, lets examine the output data presented in **Accumulated** category.

From the top menu:

☞ **Output**

☞ **Clear Display Results**

☞ in the **Select Result Category** drop-down listbox.

☞ **Accumulated** from the drop-down list:

☞ in the **Select Annual Total Result at...** drop-down listbox.

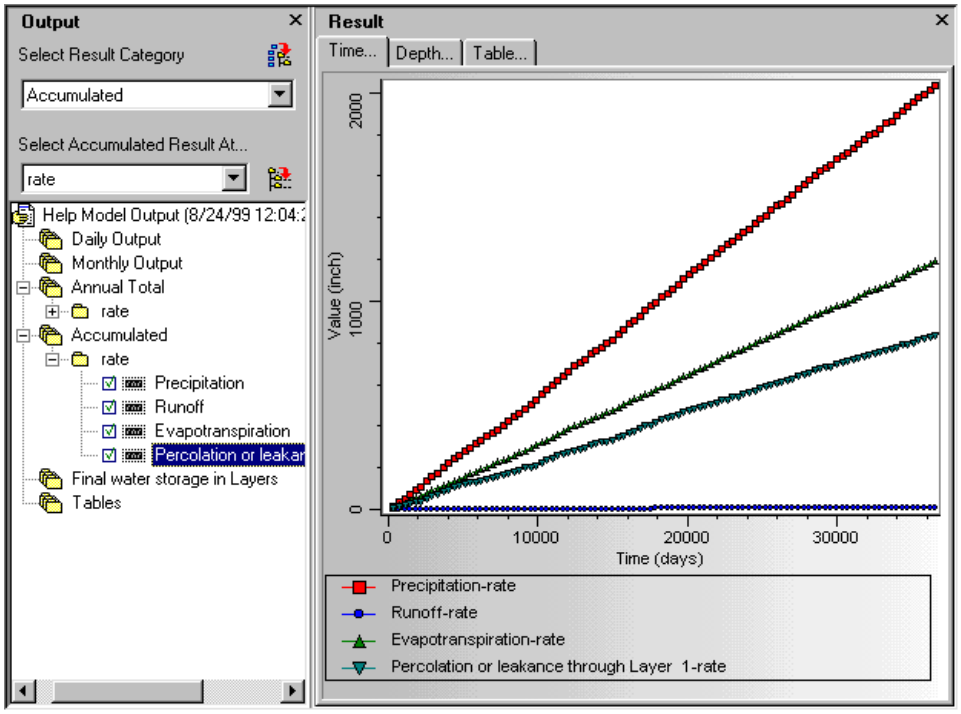
☞ **rate**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

To view all the available results for this specific type of balance, click the icon to the right of the **Select Annual Total Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1** (percolation through the bottom).



This plot illustrates that recharge is growing gradually totalling around 840 inches in 100 years.

For specific values, the profile balance constituents may be viewed in table format.

Viewing Tables

To view Visual HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Annual Totals rate**, from the **Select Table Result at..** lower listbox.

☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name **Annual Totals rate** will appear in the Output Tree.

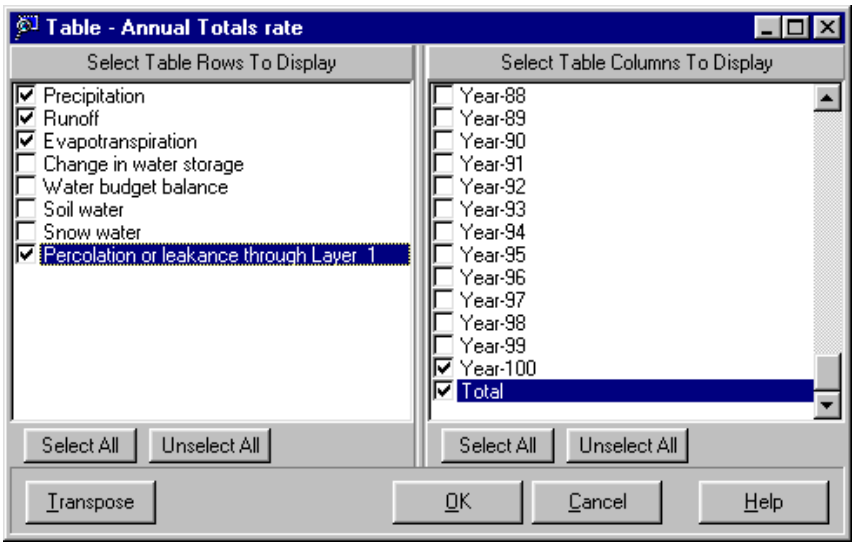
Click the check boxes beside **Annual Totals rate.** The table editing dialog box will appear.

☞ **[Unselect All]** button below the left panel.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Percolation or leakance through Layer 1**

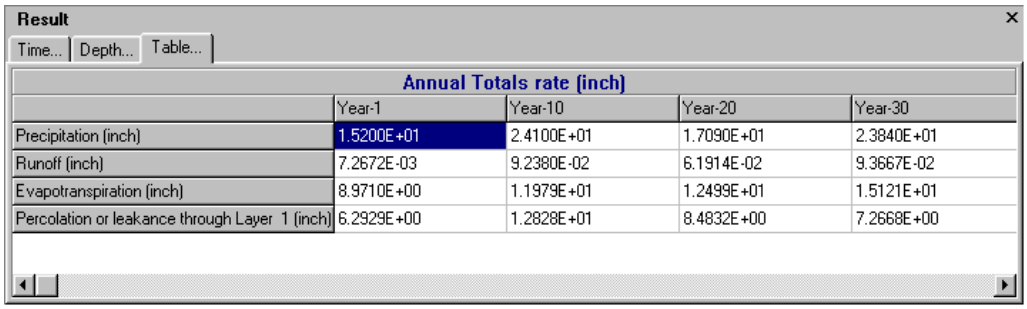
☞ **[Unselect All]** button below the right panel.

Click the check boxes beside the following output times: **Year-1, Year-10, Year-20, Year-30, Year-50, Year-100 and Total.**



☞ **[OK]**

The table will appear in the Output Window:



Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation **2034.6 in**

Runoff **8.56 in**

Evapotranspiration **1192.0 in**

Percolation or leakance through Layer 1 **834.78 in**

Because the values presented above are totals for the 100 year period, the average annual recharge will be equal to 8.34 inches at your project location. This number can be used for long term groundwater simulations. If you want to increase the level of detail in your groundwater model, you may use specific recharge values for individual years. To get these values, prepare the **Annual Totals rate** table for all years. To study the seasonal dynamics of the groundwater, you may use **Monthly** option of the output. These and other options are described in the Visual HELP manual.

Finally, lets prepare the Visual HELP report.

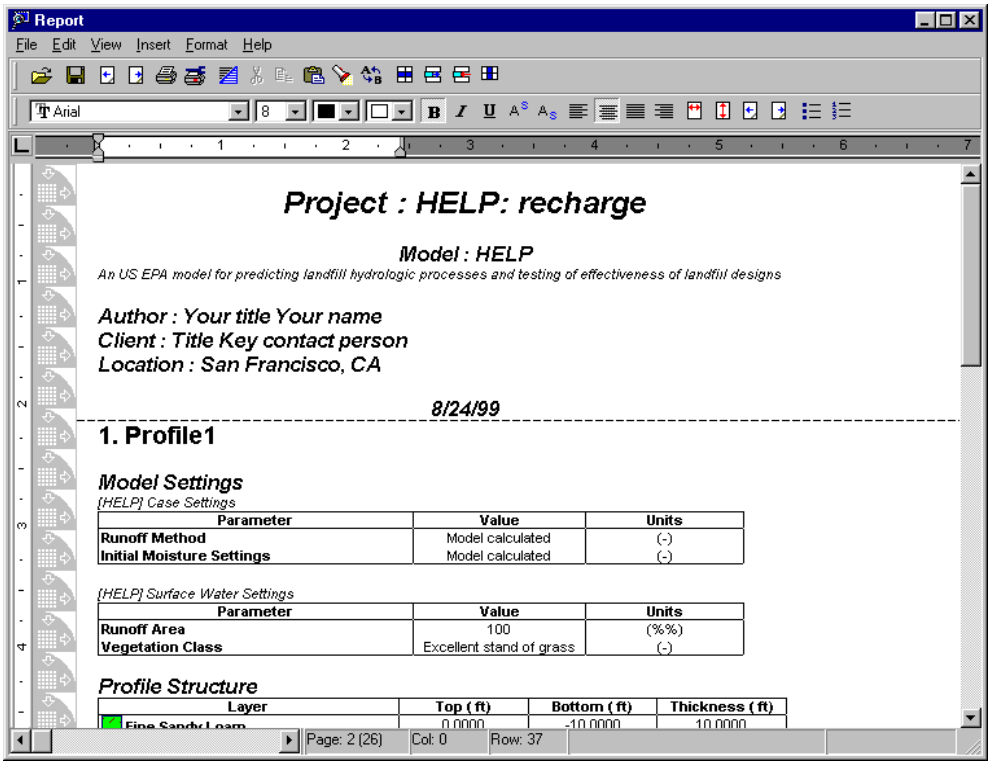
 Copyright 2000 Waterloo Hydrogeologic, Inc.

Creating a Report

As an added feature of Visual HELP, we have developed a time saving, report generator that will help you prepare professional reports of the model simulation.

To create a report and add the project input data, click the  icon from the Operational Icons toolbar.

The Visual HELP Report Generator will display your report in a separate window.



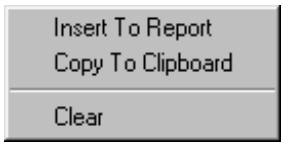
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add the table from the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the table in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The table will be inserted into the report.

To access the report (it will be hidden behind your actual Visual HELP window) in the main menu:

☞ **Project** and then select **[view report]** .

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to, insert headers /footers, change fonts/ letters size, etc.

Once you are satisfied with your report, you may print the report and/or save it for future use.

This concludes the Visual HELP demonstration exercise. Close WHI UnSat Suite Plus by clicking  button in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers for assessment of groundwater recharge. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us at (519) 746-1798 or visit our website at www.flowpath.com.

Conclusions

1) When applied intelligently, the water balance method can be a useful methodology for estimating recharge to groundwater from precipitation.
2) The HELP model (Schroeder, P., et al., 1994) in Visual HELP has been successfully applied by contemporary groundwater practitioners to substantiate historical estimates of groundwater recharge using site-specific data.
3) The results of the 100-year simulation indicate a peak recharge rate of approximately 15-inches will be observed in year 7, and a low of approximately 2-inches is expected in year 81. Average recharge to groundwater will be 8.34-inches.

References

Abraham, B., et al., 1998, A Three Dimensional Groundwater Flow Model of the Peele Dixie Well Field, Including the Florida Reprocessors Superfund Site, Ft. Lauderdale and Davie, Florida, *United States Army Corp of Engineers, Waterways Experimental Station.*

Lerner, D. et al., 1990, Groundwater Recharge: A guide to understanding and estimating natural recharge, *International Contributions to Hydrogeology: Vol. 8, International Association of Hydrogeologists. Heise Publisher, Hannover, Germany.*

Schroeder, Paul R., et al., 19994, The Hydrological Evaluation of Landfill Performance (HELP) Model, Version 3. *Environmental Protection Agency Report EPA/600/R-94/166.*

Simmers, I (ed.), 1997, Recharge of Phreatic Aquifers in (Semi-) Arid Areas, *International Association of Hydrogeologists, International Contributions to Hydrogeology: Vol. 19. A.A. Balkema Publisher, Rotterdam, Netherlands.*

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Introduction to Visual HELP

Modeling Landfill Hydrology with Visual HELP for Windows 95/98/NT/2000

The HELP Model

The **HELP** model (**H**ydrologic **E**valuation of **L**andfill **P**erformance), is a versatile program used to design, evaluate and optimize landfill hydrology and groundwater recharge rates. The HELP model is used and recognized all over the world as the accepted standard for modeling landfill hydrology, and has become an integral component for projects involving landfill operating and closure permits.

The HELP model is a quasi-two-dimensional, multi-layer hydrologic model requiring the following input data for each model profile:

- Weather data (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil properties (porosity, field capacity, wilting point, and hydraulic conductivity)
- Design information (liners, leachate and runoff collection systems, surface slope)

The landfill profile structure can consist of a combination of natural (soil) and artificial materials (waste, geomembranes) with options to install horizontal drainage layers. The HELP model also accounts for the change in slope for different parts of the landfill profile.

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, and leakage through soil, geomembranes, or composite liners.

About Visual HELP

Visual HELP for Windows 95/98/NT/2000 is the most advanced hydrological modeling environment available for designing landfills, predicting leachate mounding and evaluating potential leachate contamination. Visual HELP combines the latest version of the HELP model (v.3.07) with an easy-to use interface and powerful graphical features for designing the model and evaluating the modeling results.

Visual HELP's user-friendly interface and flexible data handling procedures provide you with convenient access to both the basic and advanced features of the HELP model. This completely-integrated modeling environment allows the user to:

- Graphically create several profiles representing different parts of a landfill,
- Automatically generate statistically reliable weather data (or create your own),
- Run complex model simulations,
- Visualize full-color, high-resolution results, ***and***
- Prepare graphical and document materials for your report.

Built-in Database tools include:

- An International Weather Generator for synthetic generation of up to 100 years of daily values of precipitation, air temperature and solar radiation.
- Soil, waste and geomembrane database which contains properties for 42 common landfill materials.

For professional applications in landfill design, Visual HELP is the only software package you will ever need! After running through this demonstration tutorial, we trust you will agree that Visual HELP is now the most complete and easy-to-use modeling tool for designing & optimizing landfill hydrology.

Visual HELP is now available as part of the **WHI UnSat Suite Plus**. This suite of 1-D unsaturated flow models is comprised of Visual HELP, VS2DT, VLEACH, and PESTAN. Now you can model landfill hydrology, predict leachate & contaminant migration, and show degradation of pesticides all with the same easy-to-use software package! For details about the **WHI UnSat Suite Plus,** please call (519) 746-1798 or visit our web page at www.flowpath.com.

New Features of Visual HELP versions 2.1 and 2.2

Profile Viewing and Editing:

- A layer can now be split into two separate layers
- Layers are systematically numbered, and labelled in the Project Tree, which simplifies interpreation of the results

Weather Generating:

- New databases and GIS searching tools have been developed for the major regions of the world (more than 3000 weather stations)
- Data in NOAA format can now be directly imported
- Diagnostics are provided to detect missing records in NOAA files

Output Presentation:

- Output units can be changed without restarting the project
- Water Balance Tables have been added
- New tree-like interface structure allows the user to easily select results
- Report Generator (a new feature of Visual HELP) allows you to display, print and export to Microsoft Word, project input settings and output graphs and tables.

System Requirements

To run Visual HELP, you will need the following minimum system configuration:

- Windows 95/98/2000, or Windows NT 3.5.1 or later;
- IBM PC or compatible,
- Pentium 100 processor;
- CD-ROM drive for software installation;
- 32 Mb of RAM;
- 60 Mb of free hard disk space; and

 Copyright 2000 Waterloo Hydrogeologic, Inc.

- Minimum display resolution of 1024 x 768

The following fonts should be installed on your computer: MS Sans Serif, Arial, and Courier New.

How to Contact Waterloo Hydrogeologic Inc.

To contact Waterloo Hydrogeologic Inc.(WHI), please use the address below:

Waterloo Hydrogeologic, Inc.
180 Columbia Street West, Unit 1104
Waterloo, Ontario, CANADA
N2L 3L3
Phone: (519) 746-1798
Fax: (519) 885-5262

To purchase Visual HELP contact the WHI Sales Department at sales@flowpath.com or visit our web page order form at: http://www.flowpath.com/order_form.html.

Please forward additional comments or suggestions to the Visual HELP development team at:

Mikhail Gogolev, Product Manager mgogolev@flowpath.com

Demonstration Exercise

Learning Objectives

In this demonstration exercise, you will:

- Use Visual HELP as a tool for landfill design and hydrologic simulation;
- Use the Weather Generator to predict meteorological data for a twenty year period and use the predicted data as input to Visual HELP;
- Run the Visual HELP simulation, view the water balance of the landfill, assess the performance of the drainage and leachate collection system, and
- Prepare a report of the simulation data and results.

The instructions for this demo exercise are provided in a step-by-step format that will allow you to choose the features that you are interested in, and examine them without having to complete the entire exercise.

The default landfill profile used in this exercise was designed in accordance with the EPA's recommendations. This design is also used in the official HELP 3.0 User's Manual. The layer properties for the default landfill profile were taken from the HELP database. The simulated landfill is located near Toronto, Ontario, Canada. The area of the landfill is 1 acre.

Terms and Notation

The following terms and notations will be used throughout this tutorial exercise:

type- Type in the given word or value

select- Click the left mouse button where indicated

a - Press the **<Tab>** key

↵ - Press the **<Enter>** key

☞ - Click the left mouse button where indicated

☞☞- Double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click, or values to be entered.

[...] denotes a button to click.

...\...\... denotes a menu selection.

NOTE: All theoretical parts of the demo tutorial are italicized

Starting Visual HELP

To start Visual HELP, you must have it installed on your computer's hard disk. Once installed, click the Windows **Start** button, choose **Programs**, navigate to the folder that contains Visual HELP, and **Visual HELP**.



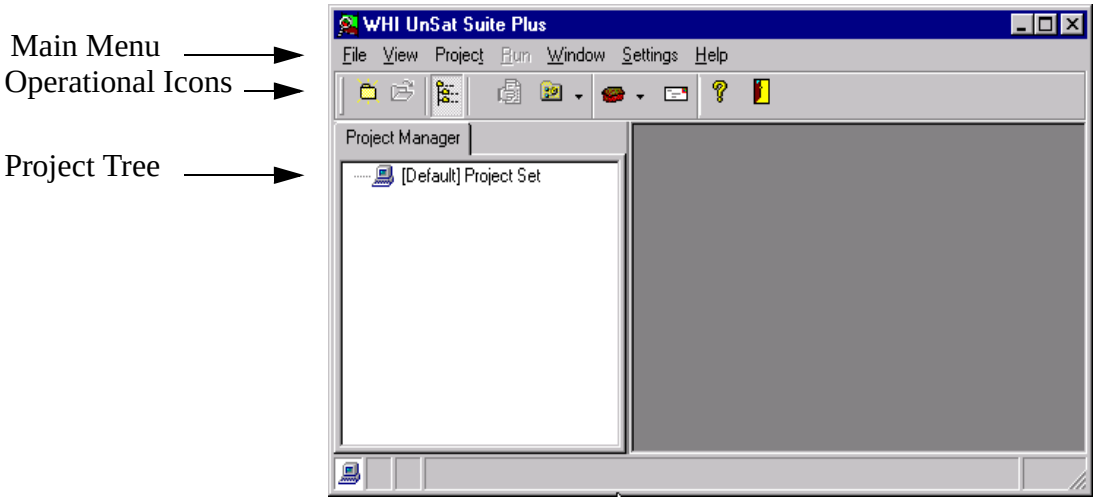
The installation program will also place an icon on your desktop, which you can also double click to start the Visual HELP demo exercise.

When Visual HELP is started an introductory screen will be displayed as shown below.



 Copyright 2000 Waterloo Hydrogeologic, Inc.

After the program is loaded, the main Visual HELP window will appear on your screen as shown in the following figure.



The Visual HELP interface structure has been designed to help you navigate through the program with ease. The operational icons are located directly below the main menu items to allow quick access to additional options. The Project Tree View is located in the left part of the screen and shows the available projects, and the structure of the model profiles in each project.

Module I: Creating a New Project

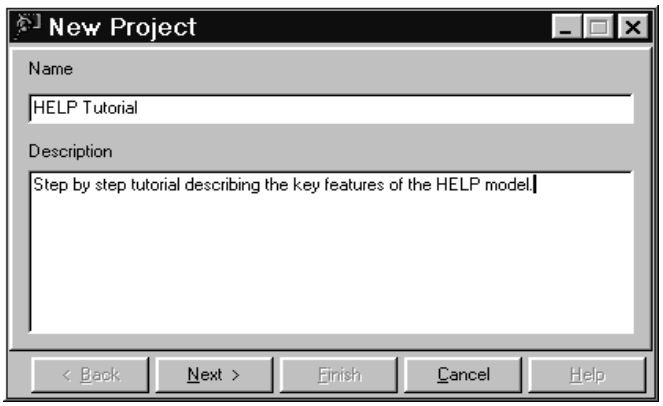
To create a new project,

☞ **File/New Project**, and the **New Project** dialog box will open.

The **New Project** wizard will guide you through the steps required to create a new Visual HELP project.

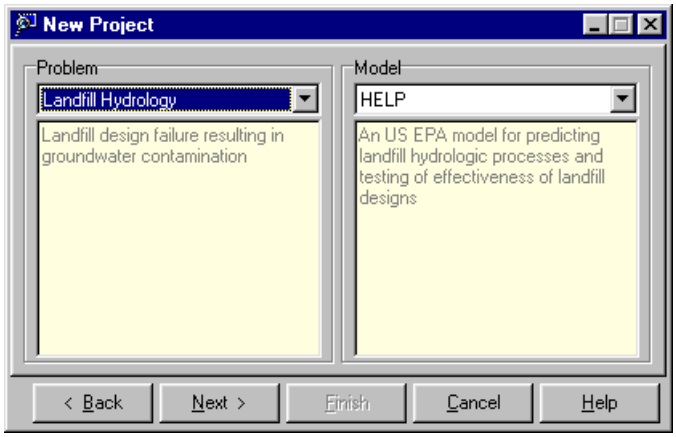
Type: HELP Tutorial, in the **Name** box.

Type: Visual HELP step-by-step tutorial in the **Description** box **.**



☞ **[Next],** to continue with your project setup. A **Problem/Model** dialog window will appear. This allows you to select which type of problem you are working on, and which model you would like to use to solve your problem.

☞ **[Problem]** dropdown menu and select **Landfill Hydrology** from the drop down list. The problem list displays all the problem classes available with the WHI UnSat Suite Plus. Also notice that Visual HELP automatically recognizes the appropriate Model that should be used with the selected Problem type.



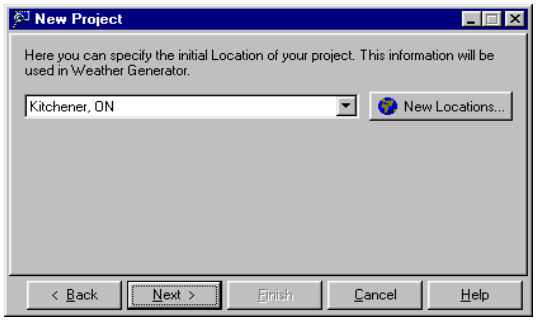
☞ **[Next]**

Once completed, the site location must be defined.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Selecting the Location

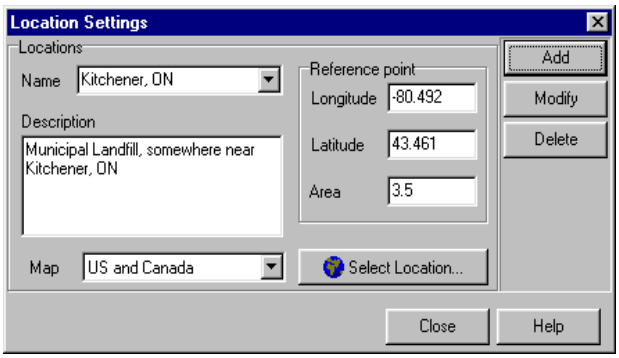
You will now select the location of your project using the GIS Location selection utility. The following dialog box should already appear on your screen.



At this stage, the location list will only contain Kitchener, ON and Buffalo, NY by default. To add a new location,

☞ **[New Locations...]**

The following **Location Settings** dialog box will appear.



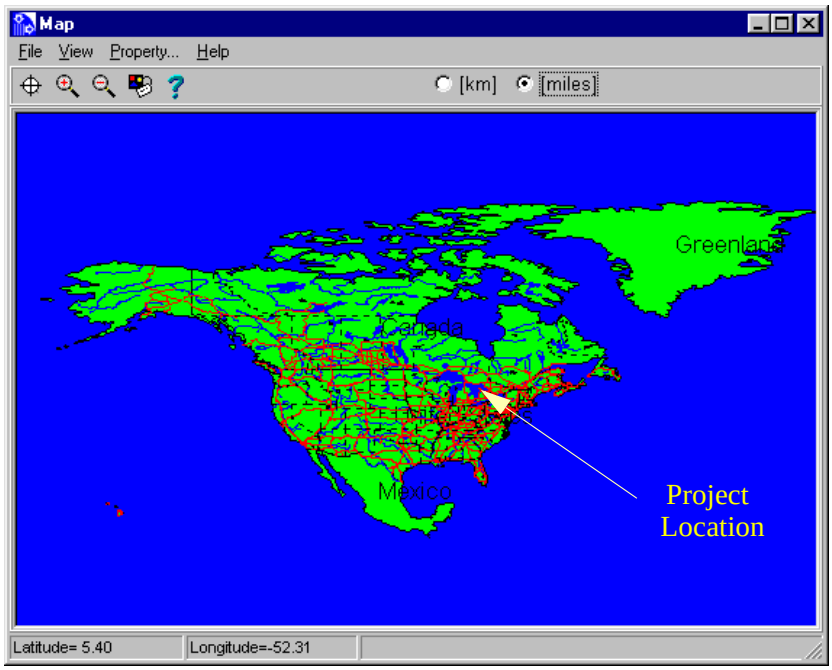
From the **Map** drop-down list,

☞ **North America,** to enable the WHI Locator, developed by WHI. The additional maps found in the list refer to various global regions that will help you to build projects from around the world!



☞ **[Select Location...],** to define our specific project location using the WHI Locator map.

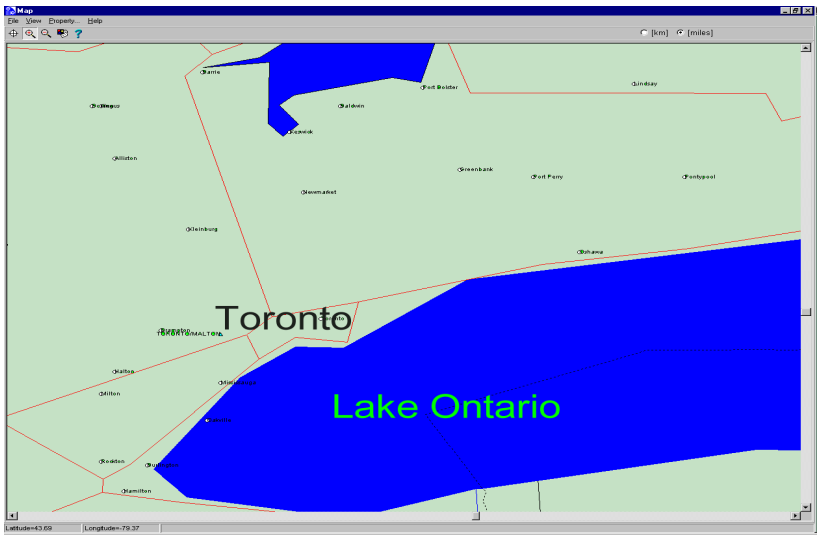
As shown below, a map of USA, Canada, and Mexico appears on your display:.



Now select Toronto, Ontario, Canada as your site location.

☞  , to zoom into the location.

Move the mouse to the right of the region where the landfill project is located (in center of the Great Lakes region). The coordinates, seen in the bottom left of the screen, should read approximately Latitude: 46.8, Longitude:-84.7. Press the left mouse button and stretch a zoom rectangle to the right of the Lake Ontario, release the button (approximate Latitude: 43.3, Longitude:-76.2). After one or two zooms, your screen should resemble the figure below.

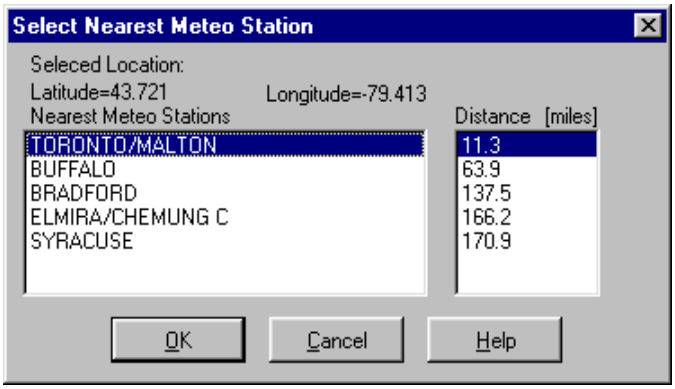


Now select **Toronto** to set the longitude and latitude of the project location.

☞  , to activate the crosshairs.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Move the crosshair to the spot on the map where **Toronto** is located and click on it with the left mouse button. The dialog box showing the five nearest weather stations, including distance will appear.



Note: If you wish to calculate the distance in kilometers, simply click the appropriate option button at the top of the map.

In the "Select Nearest Meteo Station" dialog box,

☞ **[TORONTO/MALTON],** as the nearest weather station.

☞ **[OK]**

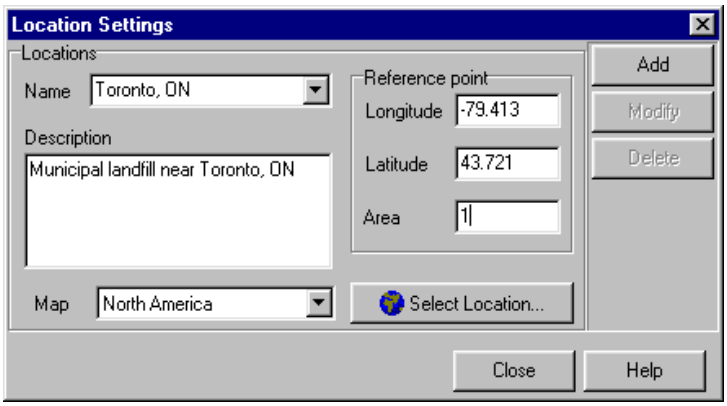
The **Location Settings** dialog box will show the selected weather stations and coordinates of the location you have selected.

Type: Toronto, ON., in the **Name** box.

Type: Municipal landfill near Toronto, ON., in the **Description** text box.

Type: 1 , in the landfill area text box: **1**.

Finally, the **Location Settings** dialog box should appear as follows.



☞ **[Add],** to add the new location to the database for future use.

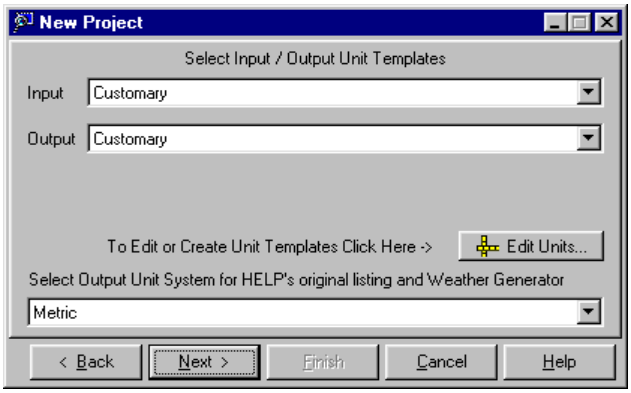
☞ **[Close]**

In the **New Project** dropdown list, select **Toronto, ON**.

☞ **[Next]**

Selecting Units

The next step is to define the units for your project (e.g. Metric or Imperial). This is completed in the "**Select Input/Output Unit Templates**" dialog box. In addition, you may select the units used for the Weather Generator data and original DOS HELP output.



Visual HELP's unit converter capabilities were developed to maximize the flexibility for data input and output unit conversion. To see the range of possibilities, just view the list of available units for hydraulic conductivity by selecting the **[Edit Units]** option.



For this demo exercise we will specify all input units as **Metric**, and output will be in **Customary** units.

From the **Input** drop-down list,

☞ **Metric**

From the **Output** drop-down list

☞ **Customary (in)**.

From the **Select Output Unit System for original listing and Weather Generator** list box,

☞ **Metric**

☞ **[Next]**

Assigning Authors and Clients

Visual HELP allows you to create and maintain your own database of project Authors and Clients for use in future projects. For this exercise we will use the default settings.

☞ **[Next]**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

All the project information will be presented, as seen in the figure below.



☞ **[Finish]** to save the information as a project in the database and add it to the project tree.

After the project settings are specified, the **New Profile Wizard** will open.

Profiles and Multiple Profiles in Visual HELP. *For the purpose of hydrological simulations, a profile represents a part of a landfill that is assumed to have the same cross-section throughout. The profile contains all of the layers of the landfill that it represents. It may also contain details of engineering components such as subsurface drainage, leachate recirculation systems, geomembranes, geonets, and composite liners. The surface of the profile can be sloped, which is typical for the peripheral parts of the landfill, or flat, which is typical for the central part of the landfill. The slope of the landfill usually mimics the shape of the waste body. At the time of landfill closure, the waste layer at the periphery of the landfill is usually sloped and the entire landfill site is capped by several layers, which form the landfill cap. Although drainage pipes are not usually installed in the landfill cap, permeable sand layers in the cap can transport water to the bottom drain or sump that surrounds the landfill. This process can be simulated with Visual HELP.*

A typical landfill can be represented as a set of profiles. This is done by creating several profiles in one project. This is useful because it allows you to use one profile to simulate the middle of the landfill, and several other profiles to simulate the edges, where the cross-section is tapering.

Selecting an Existing Profile

The **New Profile Wizard**, which appears after you finish creating a project, is shown below:

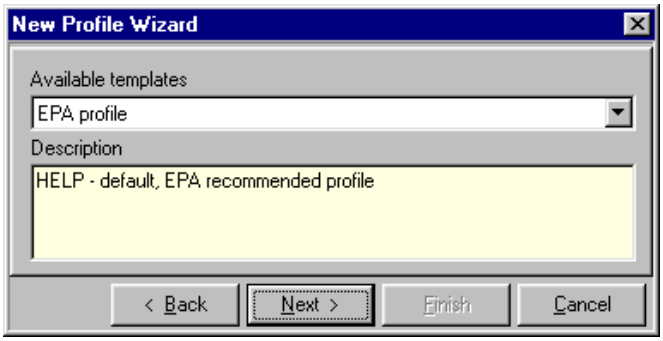


In this dialog box you may select an already existing profile template or you can build a profile 'from scratch'. In this demonstration exercise we will use the default profile template.

☞ **[use existing profile template]**

☞ **[Next]**

The **Available templates** list will appear:



☞ **EPA Profile**

☞ **[Next]**

All the profile information will be presented:.



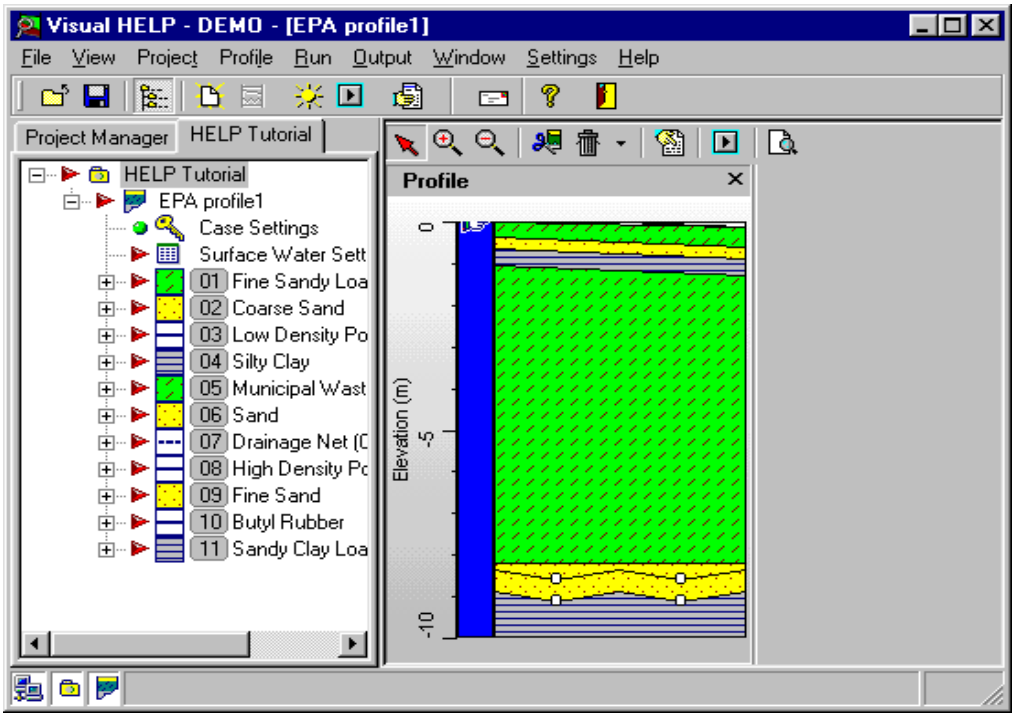
☞ **[Finish]** to load the profile.

The new project details will be added to the project tree. This concludes creating a new project using Visual HELP. We will now continue with designing the landfill profile.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Module II: Designing a Landfill Profile

After you create the project, the following picture will appear on your display.



Viewing the Default Profile

In the Profile View, located in the center part of the Visual HELP window, you will see the loaded default landfill profile with a depth scale.

A description of each landfill layer will appear when the mouse cursor is pointed at the layer.

As you can see, this profile consists of:

- a sloped landfill cap in the depth interval 0.0 - 1.2 m, consisting of fine sandy loam, coarse sand and silty clay
- a 7 m thick waste layer in the depth interval 1.2 - 8.2 m; and
- a double liner leachate collection and removal system in the depth interval 8.2-9.6 m.

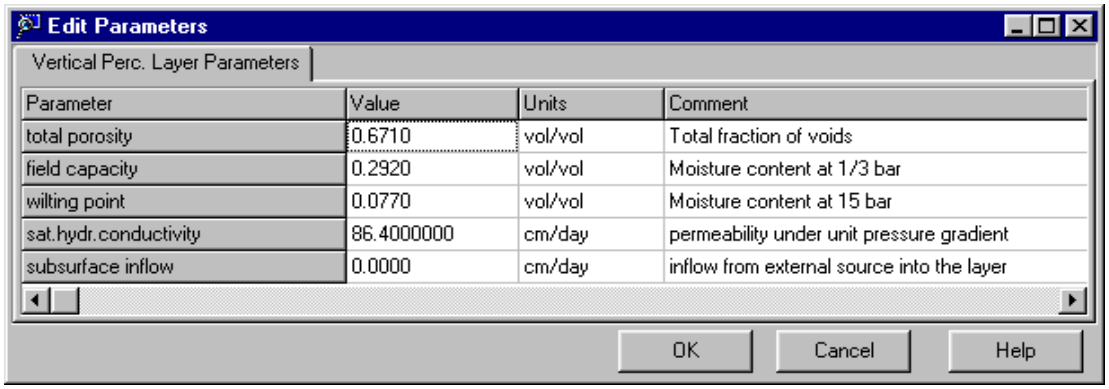
The profile construction can also be seen in the **Project Tree View** in the left part of the screen.

The Project Tree View is also used to view and modify the parameters of specific layers.

☞ **'+'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

☞ ☞ **Vertical Percolation Layer Parameters**. The **Edit Parameters** dialog box will appear.

The **Edit Parameters** menu allows you to modify the values of the parameters associated with the selected layer.



☞ **[Cancel],** in the **Edit Parameter** dialog box to exit.

☞ **'-'** sign in the box to the left of the **Municipal Waste (312 kg/cub.m)** icon.

As you can see, the Project Tree View can be used to easily access specific Profiles, Profile Layers, and Layer Parameters.

Now that you have become familiar with some viewing functions of Visual HELP, you will learn how to edit the profile. In this part of the exercise you will:

- learn how to edit the layer structure of the landfill profile
- learn how to edit parameters of individual layers
- learn how to set initial moisture conditions and runoff parameters of the profile

Editing the Layer Structure of the Landfill Profile

To assist you in designing a landfill, Visual HELP offers many convenient tools for resizing, splitting, inserting and deleting layers. Visual HELP is a graphical user interface for the US EPA HELP model and, as such, obeys all conventions of the original HELP model.

The layers in the landfill are classified by the hydraulic function that they perform. Four categories of layers are available in original HELP:

- *vertical percolation layers,*
- *lateral drainage layers,*
- *barrier soil liners, and*
- *geomembrane liners.*

The topsoil and waste layers are generally vertical percolation layers. Sand layers above liners are typically lateral drainage layers; compacted clay layers are typically barrier soil liners. Geomembranes are classified as geomembrane liners. Composite liners are modeled as two layers. In the original HELP, geotextiles are not considered as layers unless they perform a unique hydraulic function. In Visual HELP, taking into account that this class of landfill material is growing extensively, geotextiles and geonets are defined as a separate category.

Although HELP allows a wide range of layer combinations, there are some basic rules and restrictions for the arrangement of layers in a profile that should be followed. These rules are obeyed by Visual HELP and you will get a warning if you try to delete or insert a layer incorrectly.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Resizing Layers

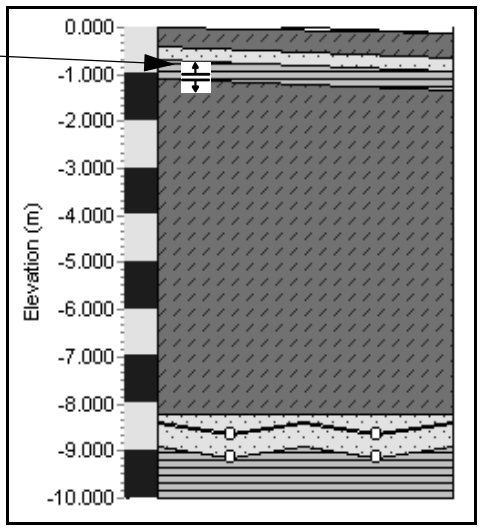
In Visual HELP, the layers are either resizable or non-resizable.

Resizable Layers *Layers built from soil and waste, grouped in the Vertical Percolation, Lateral Drainage, Barrier Soil design categories, are resizable.*

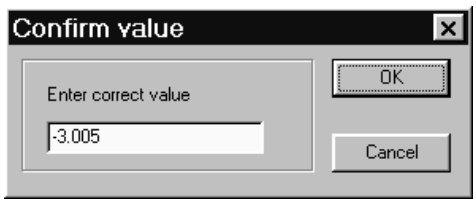
Non-Resizable Layers *Layers built from industrially produced materials with fixed thickness, grouped in the Geomembrane Liner and Geotextile and Geonet categories, are non-resizable.*

To test the resizing feature, move the mouse arrow on the **Municipal Waste (312 kg/cub.m)** layer in the profile view and move the arrow upwards to the top of the layer. As soon as the cursor encounters the layer's boundary, the cursor symbol changes:

New Cursor Symbol



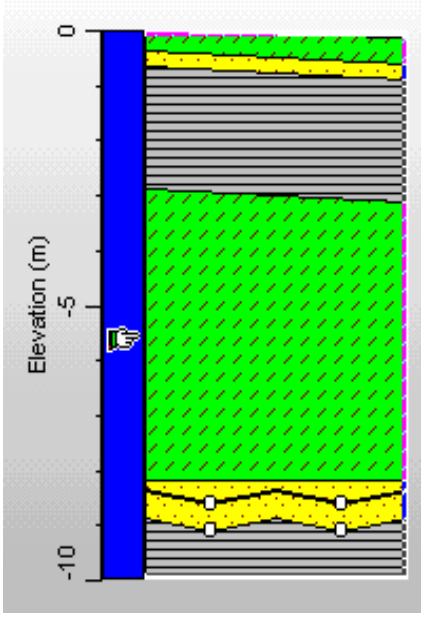
Press the left mouse button and, holding the mouse button, move the cursor downwards to approximately **-3.000** and release the mouse button. A **Confirm Value** dialog box will appear.



Type: -3.000 in the **Enter correct value** box.

☞ **[OK]**

The profile will change according to the new setting specified.



Note: Moving the layer's boundary resizes both layers which are separated by this boundary, however it does not change the total thickness of the profile. You can also change the thickness of an individual layer.

Now resize the municipal waste layer back to its original size.

Place the mouse arrow on the **municipal waste** layer in the profile view, and move the arrow up until the cursor symbol changes.

Click and drag the boundary to approximately -1.200.

Type: -1.2, in the **Enter correct value** text box.

☞ **[OK]**

Deleting a Layer

We will now examine how to delete a layer.

<Right click> the layer's name **Silty Clay**, in the Project Tree View.

.



Click the right mouse button. A shortcut menu will appear

☞ **Delete**

The layer **Silty Clay** will disappear from the Profile View. Notice an 'X' beside the **Silty Clay** layer to represent a deleted layer.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Restoring a Layer

Now let us restore the **Silty Clay** layer.

<Right click> the layer's name **Silty Clay** in the Project Tree View.

☞ **Restore** and the original layer returns.

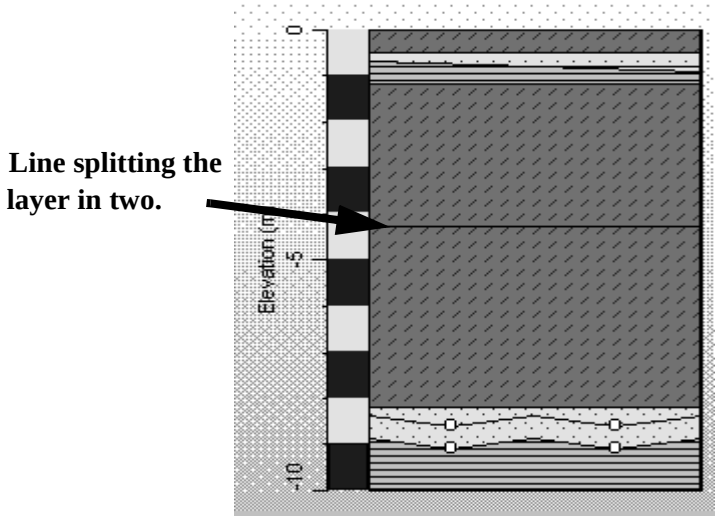
Splitting a Layer

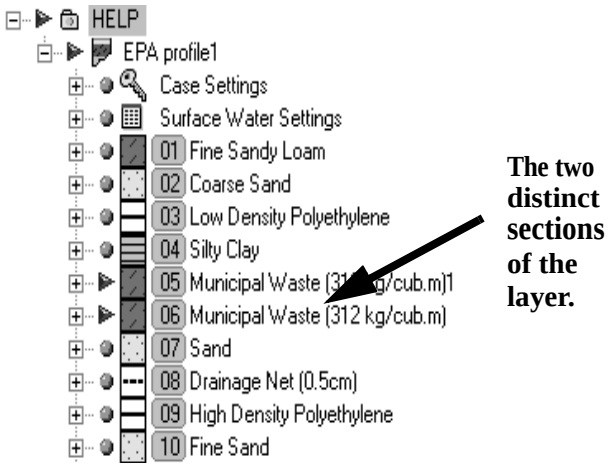
You can split a layer up into multiple sections and substitute materials for each section or assign different values of parameters for each section.

<right click> on the **Municipal Waste (312 kg/cub.m)** layer in the profile view.

☞ **Layer/Split**

A line will appear through the layer at the cursor position and a new layer will appear in the project tree. Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.



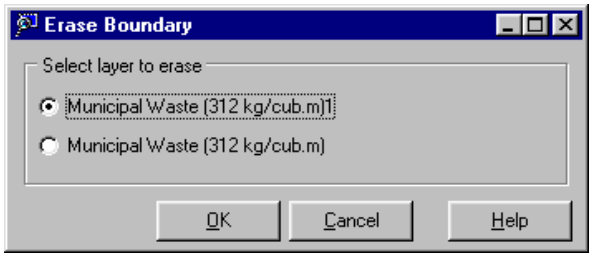


You may also substitute a material in the new layer or insert another layer between two separate parts of the layer. To return to the original profile construction, simply merge the two layers.

Place a cursor at the boundary between two parts of a layer and **<right click>**,

☞ **Merge Layers**.

In the **Erase Boundary** dialog box select the layer to erase: **Municipal Waste (312 kg/cub.m)1**



☞ **[OK]**.

Inserting a Layer

<right click> on **Sand**, in the Project Tree View

The following shortcut menu will appear



:

☞ **[Insert Layer]**

An empty **Profile Material Properties** dialog box will appear. You can now specify the new material to insert.

Note: The new layer will always be inserted above the current layer.

Note: To insert the new layer inside the existing layer, split the layer first with the split function.

Material Category Click a category from the **Material Category** list. The list will be limited to those categories that are specified by the HELP Model only.

☞ **[HELP] Barrier Soil Liner.**

Material Click the appropriate material from the **Material** list. Each layer category is associated with several textures that can be used in the design of the layer. The texture number in Visual HELP corresponds directly to the texture number in the original DOS HELP.

☞ **[Clay Loam**]

Once the material is selected, the top and bottom settings are activated.

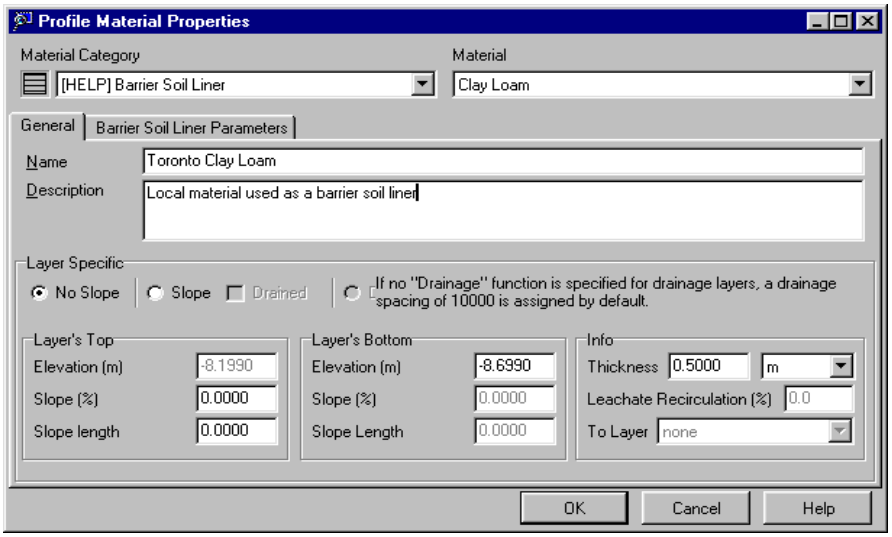
Name **Type: Toronto Clay Loam,** as the unique name for the material.

Description **Type: Local material used as a barrier soil liner**, in the comments field.

Type: 0.5 in the **Thickness** box.

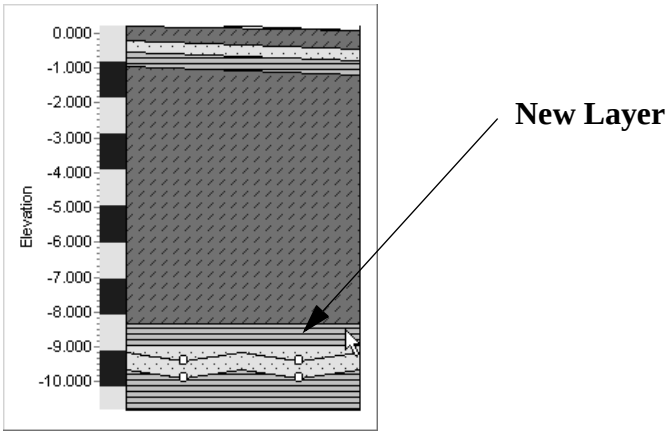
Once completed, your **Profile Material Properties** dialog box will appear as shown in the figure below:

 Copyright 2000 Waterloo Hydrogeologic, Inc.



☞ **[OK]**

The new layer is now defined:



The **Toronto Clay Loam** has now become part of the profile and you can modify it like the rest of profile layers.

To return to the original profile construction, which will be used for the rest of this tutorial, delete the recently inserted layer.

<right click> on **Toronto Clay Loam,** in the Project Tree View.

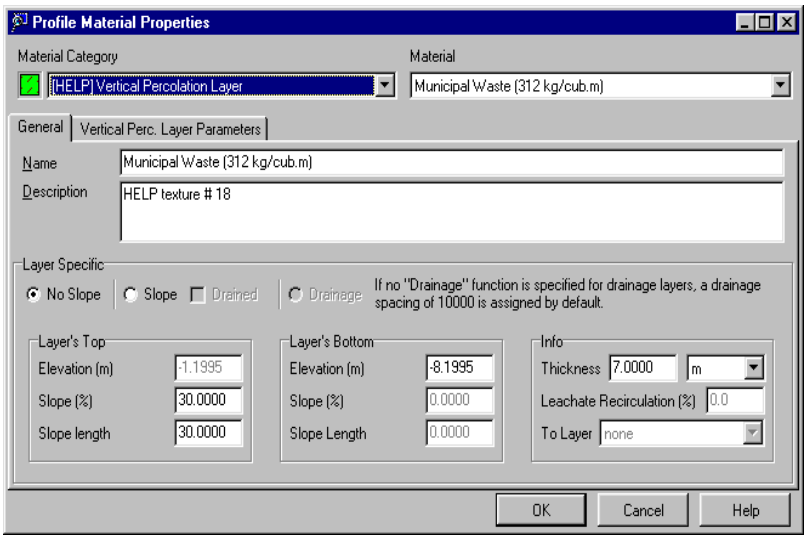
☞ **[Delete]**.

Editing Layer Parameters

The properties of a landfill material depend on the material category.

☞ ☞ the layer name **Municipal Waste (312 kg/cub.m),** in the Project Tree.

The **Profile Material Properties** dialog box will appear.



In this dialog box, you can view and edit information about the layer including;

category, material texture, slope, top or bottom elevation, and thickness.

☞ **Vertical Perc. layer Parameters** tab to edit the material properties of particular layer.

☞ **Value** beside **total porosity**

Type: 0.62

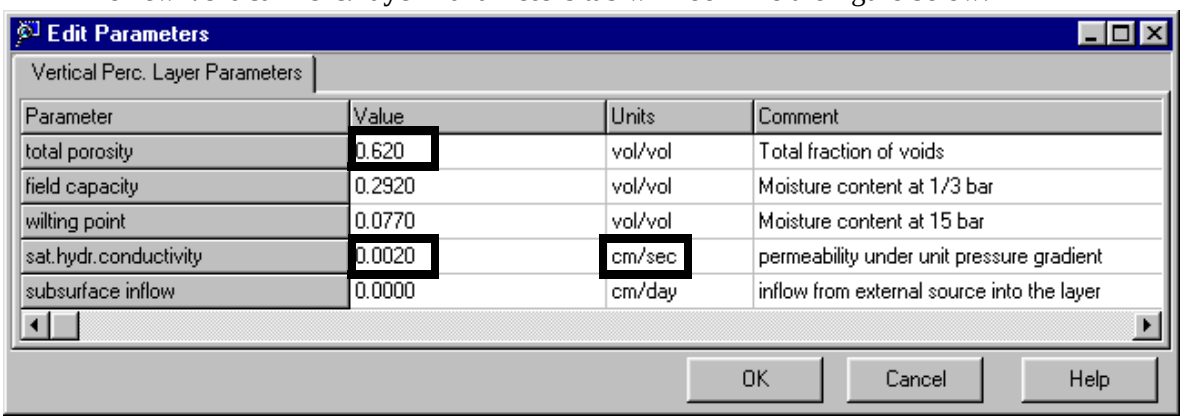
☞ **Units** beside the **sat. hydr. conductivity**, a drop-down arrow will appear.

☞ the drop-down arrow

☞ **cm/sec** from drop-down list of available units

Type: 0.002, in the **Value** field.

The new **Vertical Perc. layer Parameters** tab will look like the figure below:



☞ **[OK]**

☞ **[-] ,**beside **Municipal Waste (312 kg/cub.m)** in the Project Tree.

Now edit the properties of the Lateral Drainage layer. The layers from this category have the same hydraulic parameters as Vertical Percolation layers. In addition, the drainage and leachate recirculation parameters may be specified.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

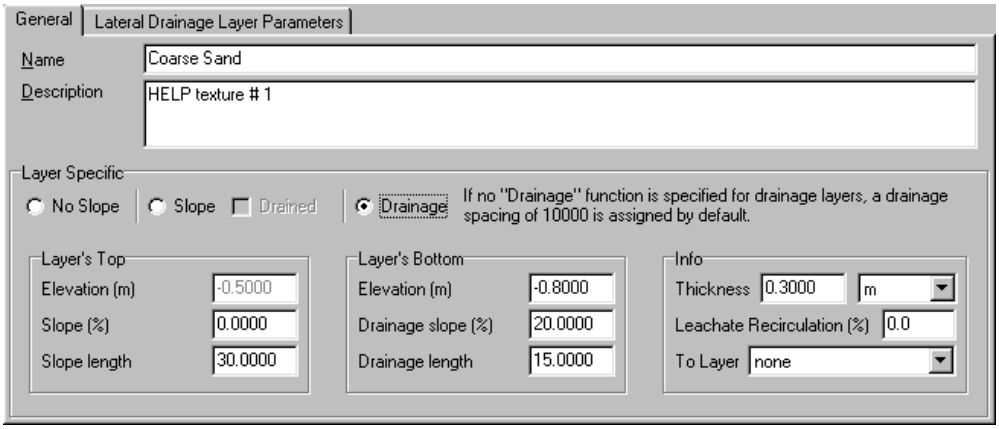
☞☞ **Coarse Sand** in the Project Tree. By default, the **Slope** function is selected for this layer.

Select **Drainage** by clicking the appropriate radial button to set the Drainage function.

Type: 20 (%) for **Drainage Slope**

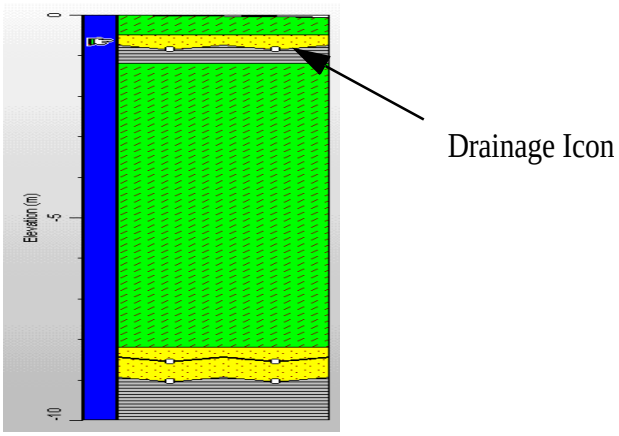
Type: 15 (m) for **Drainage Length**

Once you have made these changes, the properties for the **Coarse Sand** material should be similar to the figure below:



☞ **[OK]**

The Profile View will change to reflect the change of the **Coarse Sand** layer's status.



☞ **[-] ,**beside **Coarse Sand** in the Project Tree.

Finally, you will edit parameters of a Geomembrane Liner.

☞ **[+],** beside the **Low Density Polyethylene**

☞☞ **[Geomembrane Liner Parameters]**

☞ **Units,** beside the **sat. hydr. conductivity**

☞ the drop-down arrow

☞ **[cm/sec],** from the drop-down list.

Type: 7.00E-13, in the **Value** field.

☞ **[Units]**, beside the **pinhole density.**

☞ the drop-down arrow

☞ **[# / acre],** from the drop-down list.

Type: 10, in the **Value** field.

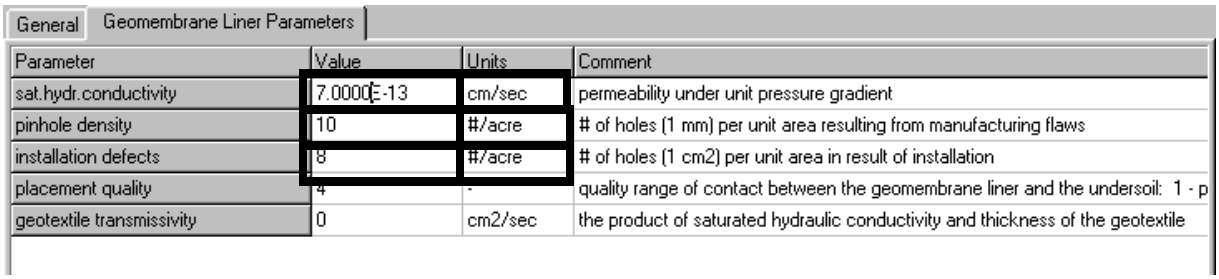
☞ **Units,** beside the **installation defects.**

☞ the drop-down arrow

☞ **[# / acre],** from the drop-down list.

Type: 8, in the **Value** field.

The **Vertical Perc. layer Parameters** tab for LDPE liner will look like the figure below.



☞ **[OK]**

☞ **[-],** beside **Low Density Polyethylene** in the Project Tree.

With this action completed, you have finished editing the parameters of individual layers of your profile.

These sample actions provide only a brief overview of the many profile design features provided by Visual HELP. Once you learn how to fully utilize all of the graphical data manipulation tools in Visual HELP, you will begin to truly appreciate the benefits of this powerful modeling tool.

Setting Initial Moisture Conditions & Runoff Parameters

Finally, you will define our landfills moisture conditions and runoff parameters. This can be completed through the **Case Settings** parameter group located in the Project Tree View.

As with the original HELP model, Visual HELP gives you two options for setting the initial moisture storage and surface water on top of the soil. You may use:

- *the model simulated values, or*
- *the user specied values.*

Depending on the selection made, runoff can be:

- *calculated by the model which will account for the type of material, slope, slope length and type of land cover,*
- *specified by the user,*
- *specified by the user and further adjusted to the slope and slope length.*

In the first case, which is the default, the model will assign realistic values for the initial moisture storage and simulate one year of landfill life. The values of moisture storage obtained from this simulation will be used as initial values.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

For this specific exercise, the runoff curve number and initial moisture content will be calculated by Visual HELP. These functions are selected by default. To view other available options,

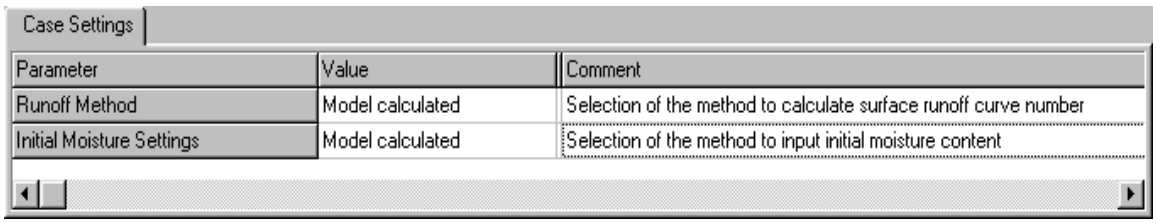
☞☞ **Case Settings** (found in the Project Tree)

☞ in the **Value** field beside **Runoff Method,**

☞ the drop-down arrow. Three options are available in the drop-down list: **User specified, User modified and Model calculated** (see the product Manual for explanation of these functions).

☞ **Model calculated.** The program will automatically pick up the slope and slope length values for the top layer and use them in the runoff calculation.

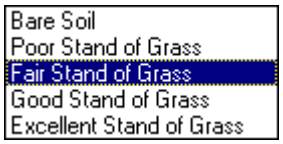
Leave the **Initial Moisture Settings** as **Model Calculated**



☞ **[OK]**

To specify additional parameters for surface runoff,

☞☞ **Surface Water Settings** from the Project Tree View.



To set the various vegetation classes, click in the **Value field**, found beside the **Vegetation Class.**

☞ the drop-down arrow. The drop down list of available types of the land cover will appear.

☞ **Fair Stand of Grass**

Note: Surface Slope and Slope Length which regulate the runoff, may be edited in the General Tab of the top profile layer.

☞ **[OK]**

The next step is to generate your site specific weather using the USDA Weather Generating program that synthetically generates daily values for precipitation, temperature and solar radiation.

NOTE: Visual HELP is also an effective tool for generating aquifer recharge values that can be used in other models such as Visual MODFLOW. To calculate accurate recharge rates for your project, simply set your model profile to represent your site specific hydrogeological conditions and run!

This completes the data preparation process. With a fully functional version of the WHI UnSat Suite you would be able to run the model. Since this is the Demo version which does not allow you to save modifications, you must now close the current project and open the project **HELP tutorial 1**, which was specifically developed for you and contains all changes you have made so far.

☞ **Project Manager** in the **Project Tree View.** The current project will close and you will get access to the list of available projects.

☞ ☞ project **HELP tutorial 1.**

You are now ready to generate the required weather data and, subsequently, run Visual HELP.

Module III: Generating Weather Data, Running the Model, Viewing Output, and Reporting

We will now create site specific weather data that will be considered when modeling our landfill profile. As an added feature, Visual HELP also allows you to easily input historical daily data in NOAA and Canadian Climate Centre formats.

IMPORTANT: The HELP model requires three different types of meteorological data that must be provided as daily values:

- *Precipitation (rain or snow),*
- *Solar radiation, and*
- *Mean air temperature.*

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- *Calculate the volume of water flowing into the landfill,*
- *Simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and*
- *Simulate surface storage, snowmelt, runoff and infiltration during cold periods.*

Visual HELP includes a built-in Weather Generator for synthetic generation of daily values of precipitation, mean temperature, and solar radiation. The WHI International Weather Generator includes a global database with data from more than 3000 stations and a GIS feature for locating the nearest stations globally

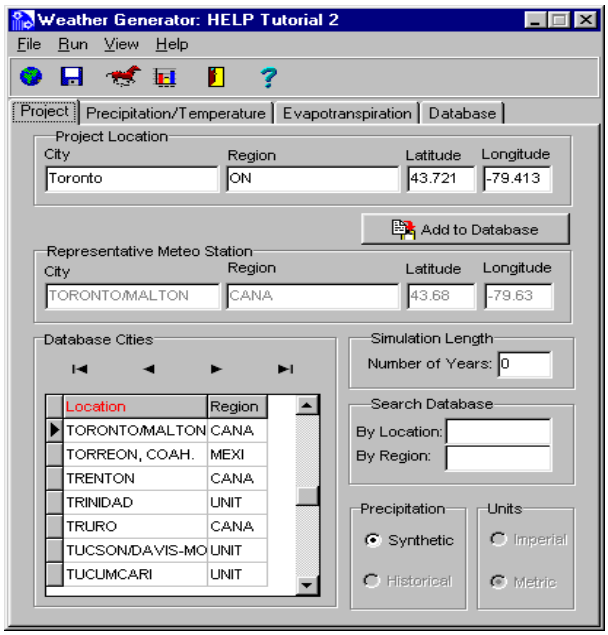
To import weather data not found in the database, you must modify the format of your data so that it meets the standards of Visual HELP. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. Customers in the U.S.A. may automatically import data in the NOAA format. Visual HELP checks NOAA files for missing daily and monthly records and informs the user about the times, for which data are missing to make the correction process easy.

☞ **Run** from the top toolbar

☞ **Weather Generator**

 Copyright 2000 Waterloo Hydrogeologic, Inc.

After an introductory splash screen, the Weather Generator dialog box will appear.



For this tutorial, you will be using data for the TORONTO/MALTON weather station. The TORONTO/MALTON station was selected among five stations located nearest to our landfill site.

Parameters for TORONTO/MALTON appear automatically in the text boxes throughout the **Weather Generator** dialog boxes after it starts (if you have time, click **Precipitation/ Temperature, Evapotranspiration and Database** tabs to see the options to customize the weather parameters).

Now you will generate weather data for your site for the next 20 years.

☞ **Number of Years** text box,

Type: 20, to represent the number of years you wish to build weather data for.

To run the Weather Generator,

☞  on the **Weather Generator** toolbar. The Weather Generator will begin computations.

☞  to save the generated files

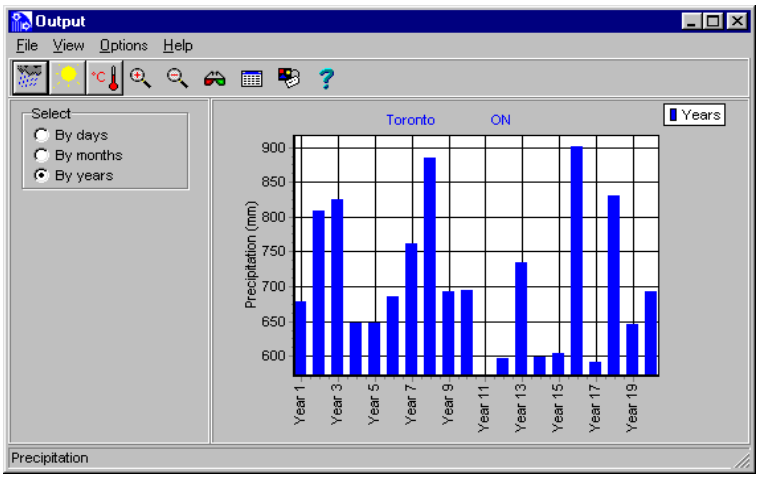
Viewing Weather Data

To view the results for the weather data you just generated,

☞  on the toolbar to view the **Output** dialog box.

☞  on the toolbar to view generated precipitation values

By default annual totals will be presented, as shown below.

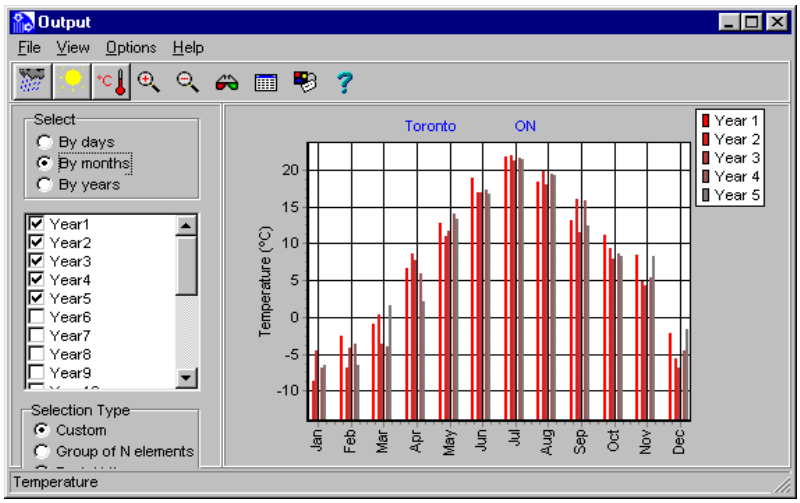


☞  to view the generated solar radiation data.

☞  to view the generated air temperature data.

☞ **By months** to view monthly outputs.

By default, monthly results for the first five years will be displayed:



☞  to view the generated raw data in table format.

☞  to close the table.

☞  to close the graphics.

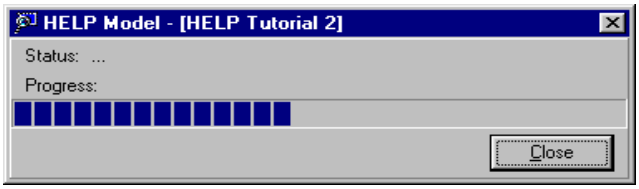
☞  to exit the Weather Generator.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

Running Visual HELP

☞  in the main toolbar to run the HELP simulation.

The program will collect input files and run the HELP model. A progression bar will appear showing you the status of your model run.

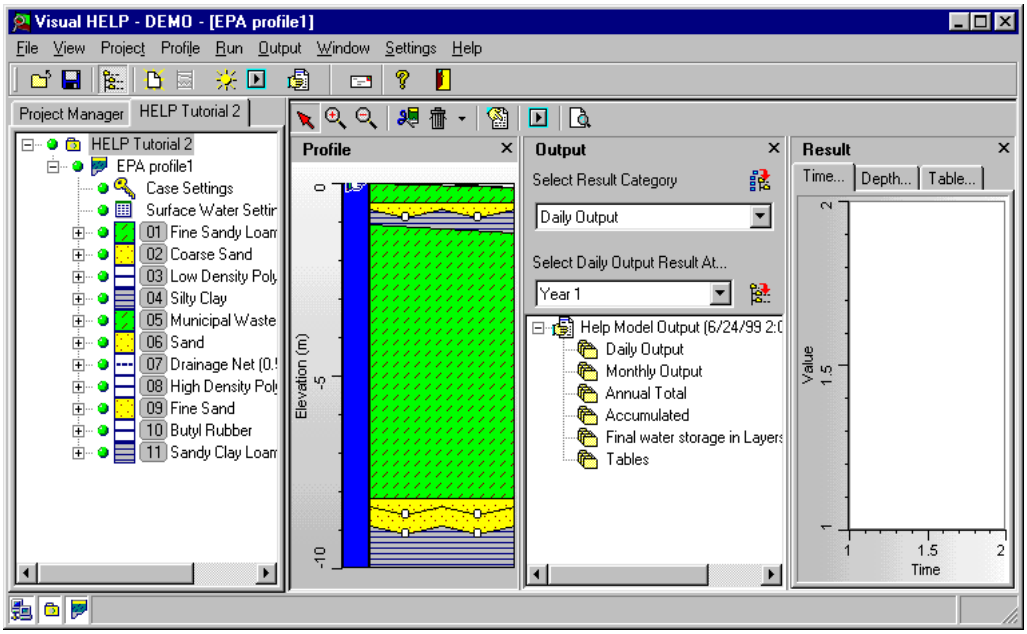


Viewing Results

Although you can easily view and print the original DOS HELP reports, we will only discuss graphical output using the Visual HELP interface.

Viewing the Output Graphs

After the model has successfully run, Visual HELP will display the Output View and Result View windows.



☞  from the Operational Icons bar, to close the Project Manager and enlarge the graphs viewing area

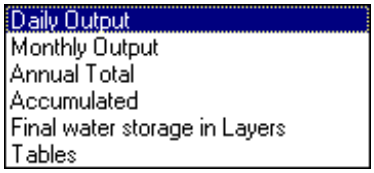
☞ **X** in the Profile View to close the window.

The **Output** View window contains two drop-down fields:

Select Result Category (used to select general result category)

Select Specified (Category Name) Result At.... (used to select specific result category)

To select the general output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



For this exercise, we recommend viewing your results with an **Accumulated** category. This plot will show you the total volumes of water drained during the simulation period. This allows you to examine the total volume of leachate that percolated through the landfill bottom during the simulated time and assess the total volumes of other water balance constituents.

☞ **Accumulated**

The results for the the **Accumulated** category will appear in the listbox below. To view all available results, click the arrow in the drop-down listbox.

☞ **Select Accumulated Result at...** , to view the available balance types. ie) rate, volume

Rate means the annual rate of the balance constituent, and **volume** means the volume for the area represented by the profile.

☞ **rate**

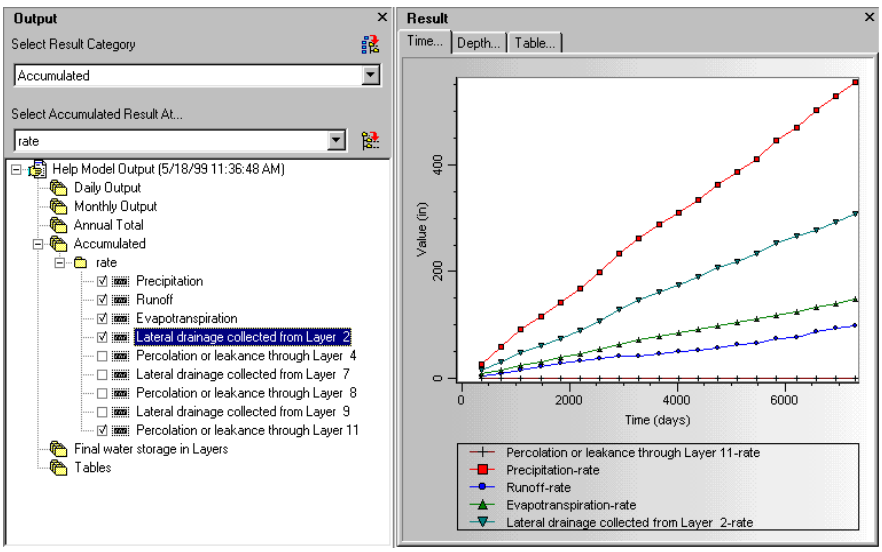


To view all the available results for this specific type of balance, click the icon to the right of the **Select Accumulated Result at...** box.

The list of available balance constituents will open in the Result Tree.

Click the check boxes beside the following variables: **Precipitation, Runoff, Evapotranspiration, Lateral Drainage collected from Layer 2** (the first drainage layer), **Percolation or leakance through Layer 11** (percolation through the bottom).

The graph of the variables will appear in the Result View window.



To erase a variable from the Result View window, deselect the corresponding check box in the Result Tree.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

From the graph, one may conclude that Runoff, Evapotranspiration, and Lateral Drainage collected from Layer 2 play an important role in the landfill balance while Percolation through Layer 11 (the landfill bottom) is equal or close to 0.

Viewing Tables

To view HELP tables,

☞ **Tables,** from the **Select Result Category** drop-down listbox.

☞ **Accumulated rate**, from the **Select Table Result at..** lower listbox.

☞  **Add to Output Tree** icon to the right of the lower output listbox

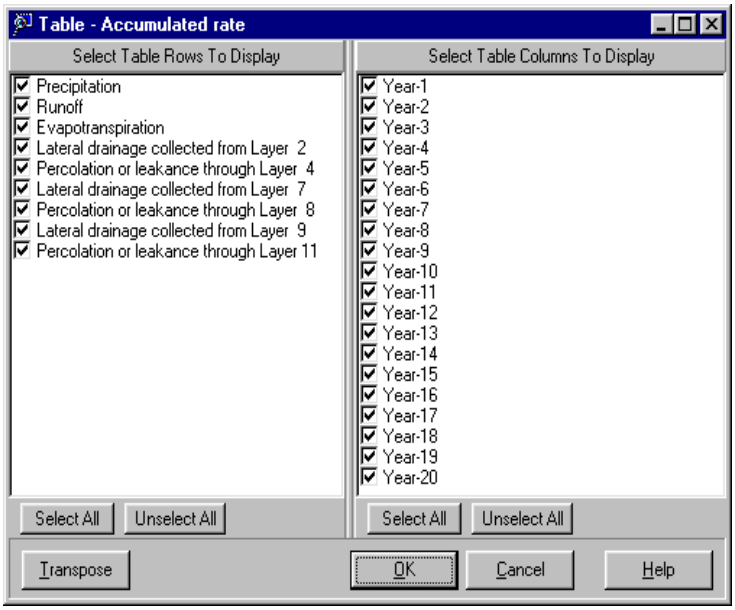
The table name **Accumulated rate** will appear in the Output Tree. For future use, let's place the table results into a new Result Window.

To do this, **<right click>** the name, **Accumulated rate** in the Output Tree. The following menu will appear:



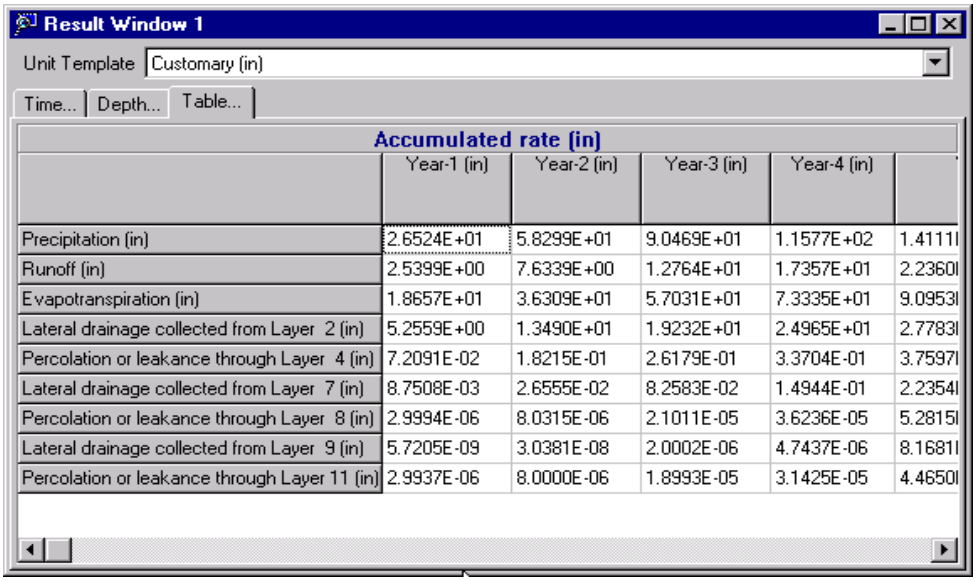
Choose **New Result Window**.

The New **Result Window 1** will open in the background and the table editing dialog box will appear.



☞ **[OK]**

The table will appear in **Result Window 1**:



	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)	
Precipitation (in)	2.6524E+01	5.8299E+01	9.0469E+01	1.1577E+02	1.4111I
Runoff (in)	2.5399E+00	7.6339E+00	1.2764E+01	1.7357E+01	2.2360I
Evapotranspiration (in)	1.8657E+01	3.6309E+01	5.7031E+01	7.3335E+01	9.0953I
Lateral drainage collected from Layer 2 (in)	5.2559E+00	1.3490E+01	1.9232E+01	2.4965E+01	2.7783I
Percolation or leakance through Layer 4 (in)	7.2091E-02	1.8215E-01	2.6179E-01	3.3704E-01	3.7597I
Lateral drainage collected from Layer 7 (in)	8.7508E-03	2.6555E-02	8.2583E-02	1.4944E-01	2.2354I
Percolation or leakance through Layer 8 (in)	2.9994E-06	8.0315E-06	2.1011E-05	3.6236E-05	5.2815I
Lateral drainage collected from Layer 9 (in)	5.7205E-09	3.0381E-08	2.0002E-06	4.7437E-06	8.1681I
Percolation or leakance through Layer 11 (in)	2.9937E-06	8.0000E-06	1.8993E-05	3.1425E-05	4.4650I

Scroll to the end of the table to see the final values for accumulated volumes.

Precipitation	**553.98 in**
Runoff	**74.84 in**
Evapotranspiration	**359.14 in**
Lateral Drainage collected from Layer 2	**119.43 in**
Percolation or leakance through Layer 11	**0.00 in**

These values show that the default EPA landfill design (slightly modified by you during the exercise), provides good ground water protection for our project location. Having made such an optimistic conclusion, lets prepare the Visual HELP report.

☞  to close Result Window 1.

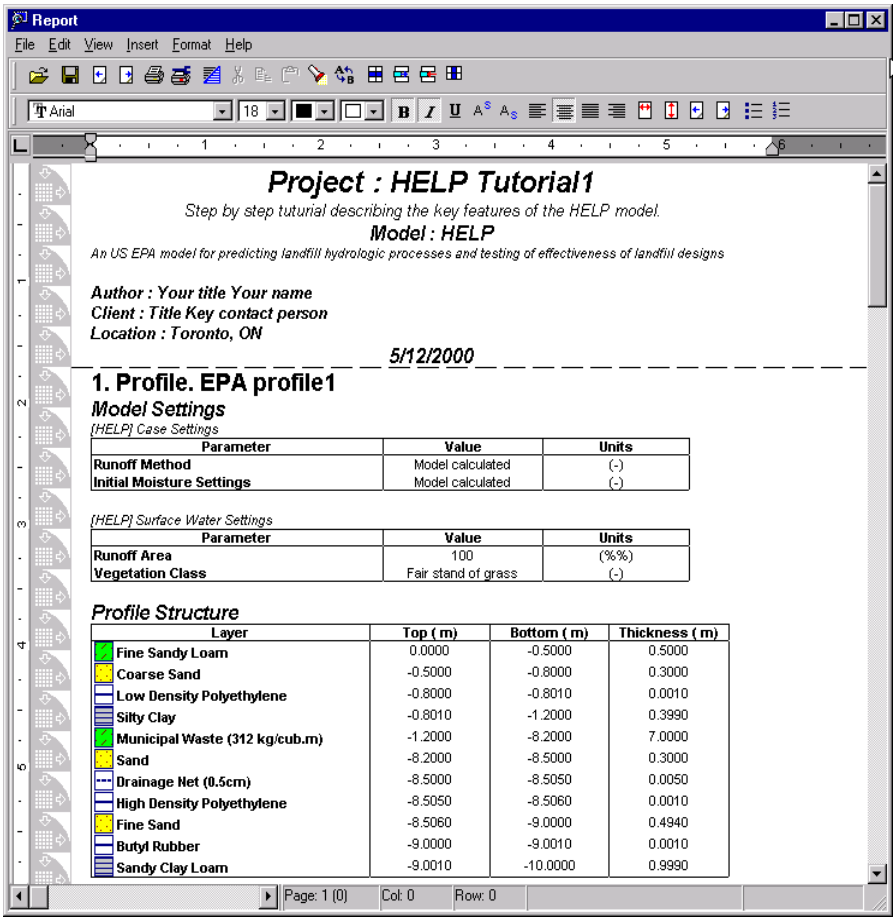
Creating a Report

As an added feature of Visual HELP, we have developed a time saving report generator that will help you prepare professional reports of the model simulation.

 To create a report and add the project input data, click the icon from the Operational Icons toolbar.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The Visual HELP Report Generator will display your report in a separate window:



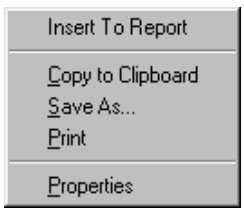
By default, the Report Generator lists all input data for your project. In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by Visual HELP.

Note: The graphs and tables will be placed at the insertion point.

To add a graph from the Result View to the report:

In the **Report** window scroll the cursor to the end of the report

<right click> the graph in the Visual HELP Result View window. The following menu will appear:



☞ **Insert To Report**. The graph will be inserted into the report.

Go to the "Project" in the main menu and select **[view report]** to continue editing. To re-size the graph you just inserted, click the graph in the **Report** window and stretch it until it reaches the proper size.

To add a table from the **Result Window 1** to the report:

In the **Report** window place the cursor below the imported graph.

<right click> the table in the **Result Window 1**

3) ☞ **Insert To Report**. The table will appear in the report. Again, go to the "Project" in the main menu and select **[view report]** to continue editing or to view your report at any time.

The table may be larger than the Report window allows. In this case, the table will be automatically wrapped to improve the general appearance of the report. The report is fully customizeable and allows you to, insert headers /footers, change fonts/ letters size, etc..

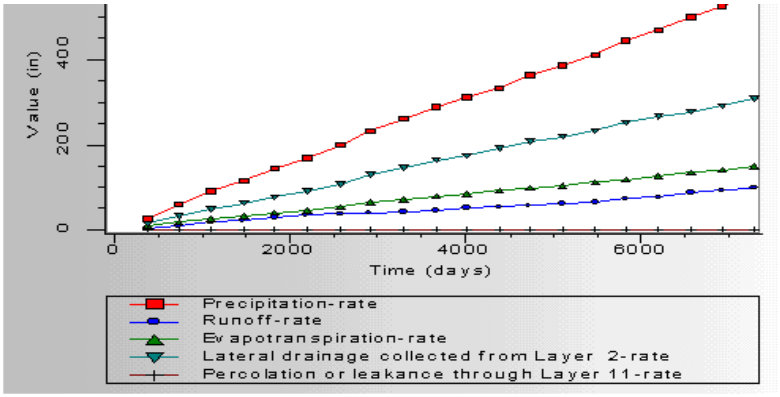


Fig.1 Accumulated balance constituents.

Table 1. Accumulated rate (in)

	Year-1 (in)	Year-2 (in)	Year-3 (in)	Year-4 (in)
Precipitation	2.6669E+01	5.8476E+01	9.0929E+01	1.1642E+02
Runoff	3.9219E+00	9.5287E+00	1.6193E+01	2.2387E+01
Evapotranspiration	8.2591E+00	1.5974E+01	2.4371E+01	3.1858E+01
Lateral drainage collected from Layer 2	1.4484E+01	3.1264E+01	4.8366E+01	6.2130E+01
Percolation or leakance through Layer 4	3.7569E-03	6.5516E-03	9.2846E-03	1.2289E-02
Lateral drainage collected from Layer 7	3.7532E-03	6.5417E-03	9.2717E-03	1.2271E-02
Percolation or leakance through Layer 8	3.7459E-06	8.9520E-06	1.2983E-05	1.7380E-05
Lateral drainage collected from Layer 9	3.5588E-06	8.4902E-06	1.2188E-05	1.6448E-05
Percolation or leakance through Layer 11	1.8707E-07	4.6209E-07	7.9524E-07	9.3185E-07

(continued)

	Year-5 (in)	Year-6 (in)	Year-7 (in)	Year-8 (in)

Once you are satisfied with your report, you may print the report and/or save it for future use. Let's take a look at additional output features available with Visual HELP.

Peak Daily Values

To assess the landfill hydrologic performance in extreme conditions, it is important to know when the peak hydrologic events occurred. The **Peak Daily Values** table picks up the dates of such events.

☞ **[Tables]** from the **Select Result Category** drop-down listbox.

☞ **[Peak daily values]** from the **Select Table Result at..**



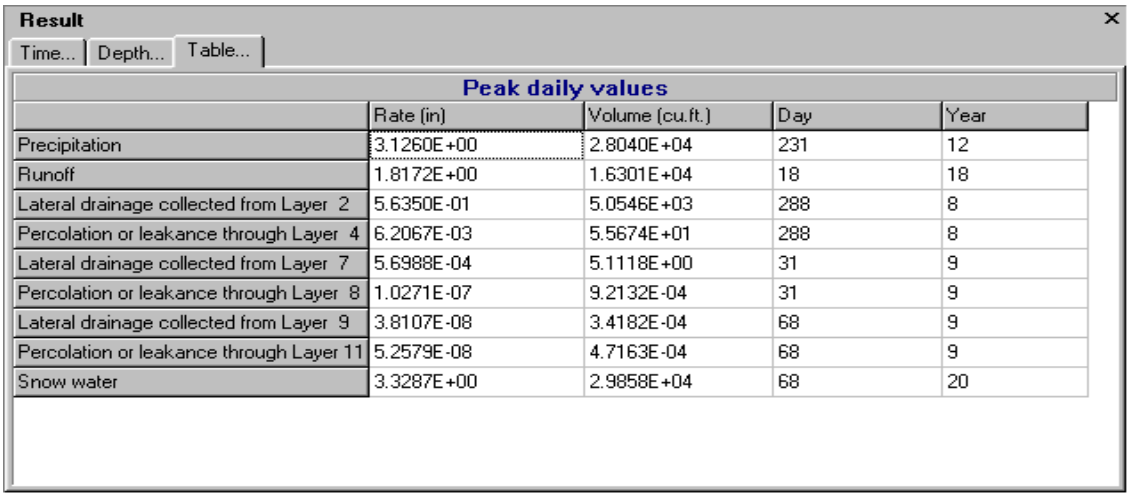
☞ **Add to Output Tree** icon to the right of the lower output listbox.

The table name "**Peak daily values**" will appear in the Output Tree. Click the check box beside it.

☞ **[OK]** in the table editing dialog box.

 Copyright 2000 Waterloo Hydrogeologic, Inc.

The following table will appear:



Peak daily values

	Rate (in)	Volume (cu.ft.)	Day	Year
Precipitation	3.1260E+00	2.8040E+04	231	12
Runoff	1.8172E+00	1.6301E+04	18	18
Lateral drainage collected from Layer 2	5.6350E-01	5.0546E+03	288	8
Percolation or leakance through Layer 4	6.2067E-03	5.5674E+01	288	8
Lateral drainage collected from Layer 7	5.6988E-04	5.1118E+00	31	9
Percolation or leakance through Layer 8	1.0271E-07	9.2132E-04	31	9
Lateral drainage collected from Layer 9	3.8107E-08	3.4182E-04	68	9
Percolation or leakance through Layer 11	5.2579E-08	4.7163E-04	68	9
Snow water	3.3287E+00	2.9858E+04	68	20

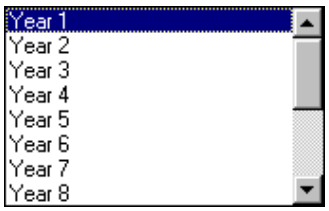
You may conclude that no leakage occurred through the bottom of the landfill (Peak daily percolation/leakage through layer 11). You may also find that the peak precipitation (3.12 in) happened on day 231 of year 12 and that peak runoff (1.81 in) happened on day 18 year 18.

To learn more about the circumstances of these events, Daily Output graphs may be studied.

Daily Output

☞ **[Daily Output]** from the **Select Result Category** drop-down listbox.

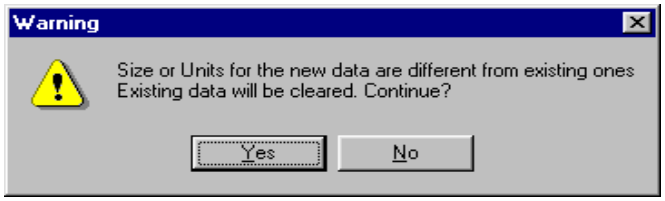
Next select the year for which you would like to have the results to be displayed from the listbox.





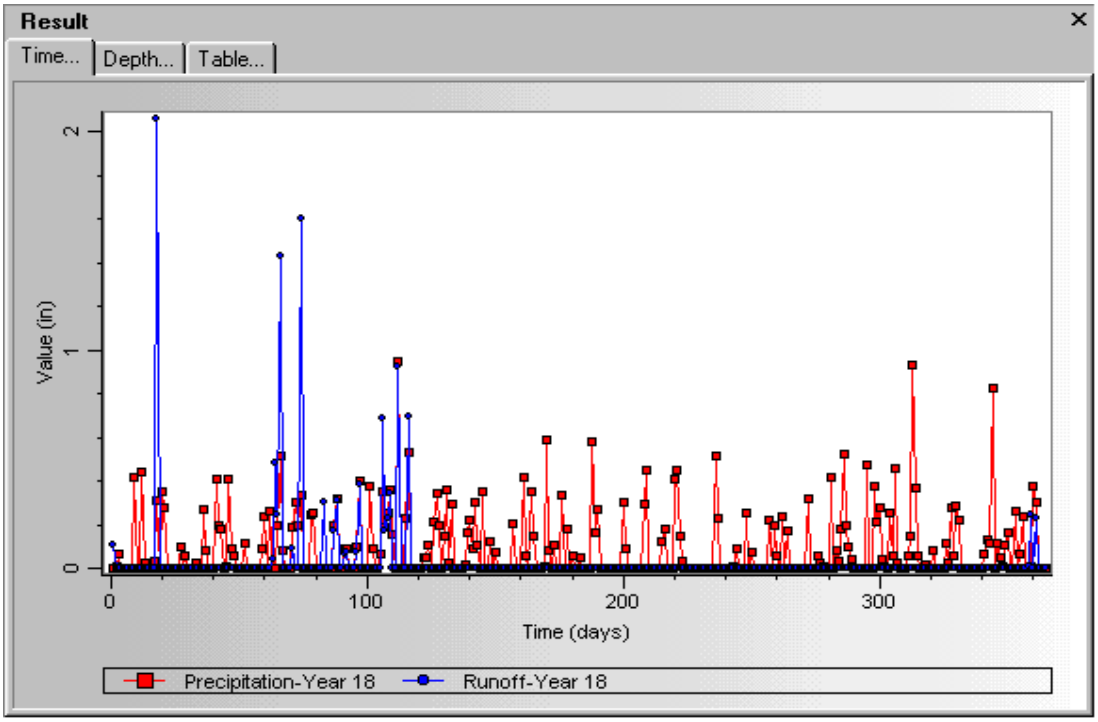
Scroll the list of years, and select the desired year: **18**. To view the available results for this specific type of balance, click the **Add to Output Tree** icon to the right of the lower output listbox.

The tree of available variables for year 18 will appear. Select **Precipitation** and **Runoff** by clicking the appropriate check boxes. This means that you are going to place these graphs to the Main Result Window. A warning will appear:



☞ **[Yes]**

The graph for the daily values of these variables will appear:



This graph shows that the maximum runoff event was not preceded by very intensive precipitation. However, after reviewing the weather data, it appears that day 18 was the first warm day with an average temperature of 5 degrees C (41 Fahrenheit) after a long span of frosty days. The peak runoff was caused by the intensive thaw of snow accumulated during the previous cold period.

Click to close the graph.

This concludes the Visual HELP demonstration exercise. Close Visual HELP by clicking in the top right corner.

We hope these instructions have provided you with a good understanding of the capabilities and tools Visual HELP offers. If you have time, we would encourage you to go back and re-examine some of the other powerful features and analysis capabilities which were not covered by this demo exercise. If you have any questions about the functionality, capabilities, or features of this software, please do not hesitate to contact us (519) 746-1798 or visit our webpage, www.flowpath.com.

 Copyright 2000 Waterloo Hydrogeologic, Inc.