Getting Started with SoilCover– A step-by-step tutorial

Introduction

This tutorial presents a simple hypothetical analysis designed to highlight many of the features available in SoilCover and to introduce you to SoilCover's EXCEL user-interface for input and data reduction. It does not include a review of detailed theory regarding each parameter/feature, nor does it identify what range of values would be considered reasonable to obtain a realistic solution. Please refer to the Reference Manual installed with SoilCover for more theoretical information.

Launching SoilCover

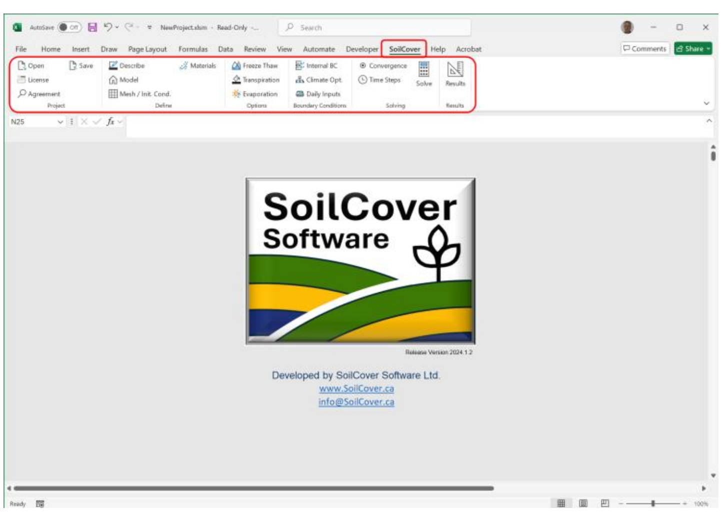
When you install SoilCover it will create a shortcut icon on your desktop.

You can start the program using this shortcut,

or you can open Excel and navigate to

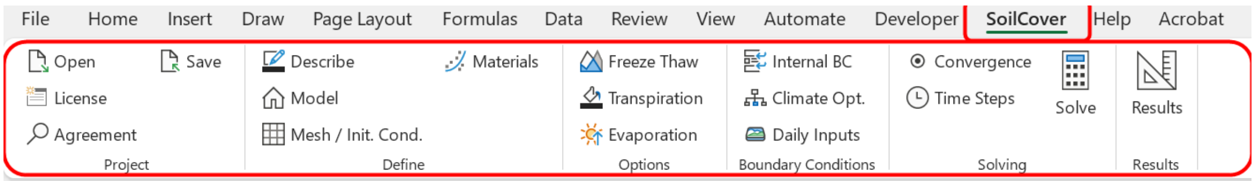
C:\SoilCover\SCResources\NewProject.xlsm

When the program opens, you will see the screen shown below. It is a Read Only Excel file but it has a new toolbar and menu item as shown.



The new toolbar is where you will execute all SoilCover operations. In general, as you develop a new model, you will access every menu item by working left to right.

Menu Groups



You can open, save, check the license etc. using the items in the **Project** group.

At this point, you should use the Save command and give your project file a unique name and folder location.

As you use the software more often and develop more project files, it will save you some time if you open an existing project file and make modifications to it including assigning a new name.

Use the items in the **Define** group to describe your project location and solver duration, set up material properties, define the model geometry and assign materials and initial conditions, and then create a finite element mesh.

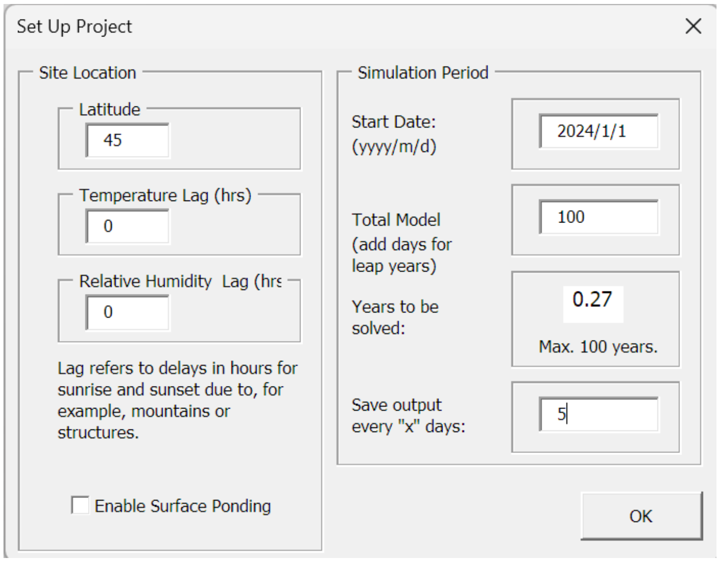
Further, using the **Options** group, you can specify to include freeze/thaw, plant transpiration, and evaporation computation preferences.

After that, you will assign internal boundaries such as source or sink fluxes and their locations, how you want climate data computed and applied, and then you must specify climate and base and surface **boundary conditions** for each day in the analyses.

Prior to **solving**, you specify convergence and time step controls, and then you can start the solver. The Solver command launches a Fortran executable file which reads in all your model parameters, solves each day of the analyses, and saves daily and cumulative solution date to text files that are then later reimported to the Excel file using the **Results** command.

Step by Step Example

As noted above, generally you should use all menu command buttons in a left to right direction to ensure you have a fully defined mode. Using the Describe command, set up a key project parameters as shown below.

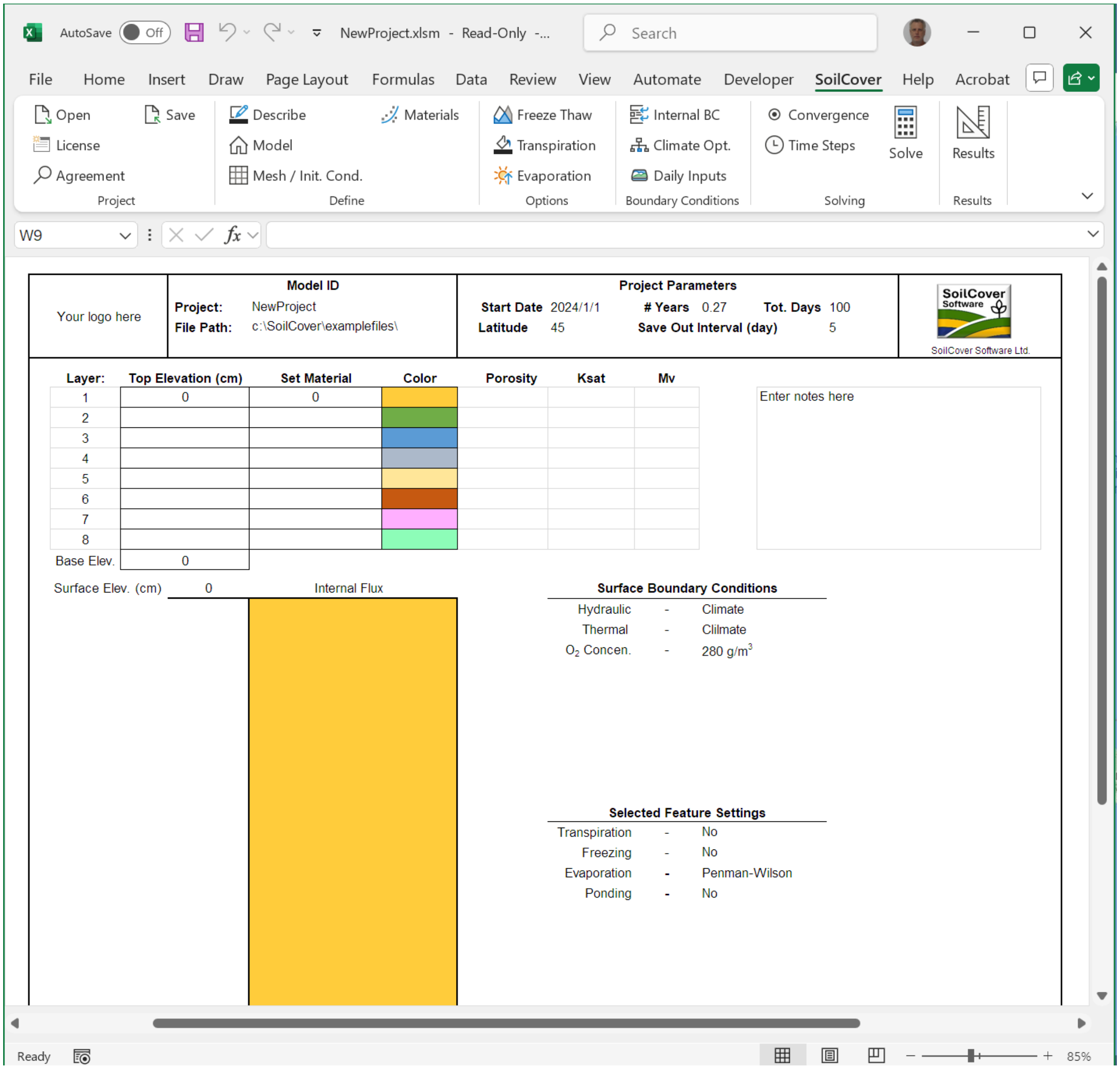


In this example, the latitude is set as 45 deg. north, there is no lag in temperature or relative humidity caused by proximity to mountains or oceans, we are not enabling surface ponding (which would store non infiltrated snow or rain as a pond with a total head boundary condition until such time as the pond infiltrated or evaporated), the start date is January 1, and the model will solve for 100 days with data written to the results file every 5th day. The model estimates it is solving 0.27 years.

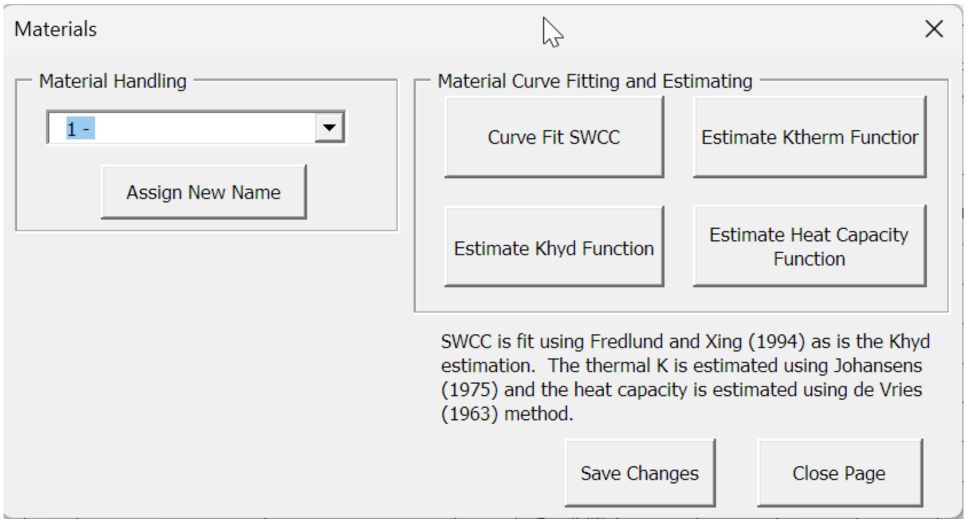
The first time you use the Describe command and close the dialogue box, you may get an Image Update error. This can be ignored. It is telling you that not all model inputs are entered yet for generating a full model image, but this will resolve itself as you enter material properties and define the geometry.

The following image is the Model view. It includes the project name, file location, main parameters, and various soil layers and assigned material properties. Prior to entering any soil layer information, you should first define some material properties. Once that is complete, then you can return to this page and enter the soil layer information and assign a material.

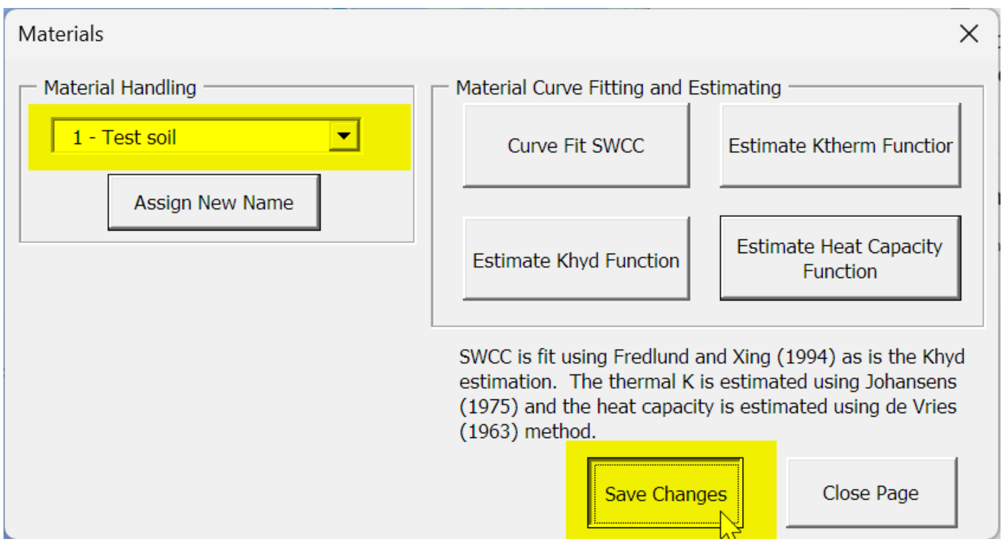
Other information on the Model view includes boundary conditions, other feature settings, and the location of an internal “flux section” you can tell the model to accumulate internal flow data at. The model will automatically accumulate all boundary fluxes for later presentation.



Prior to building the soil layers, you must input material properties. Use the Materials command and you will see the following dialogue box.



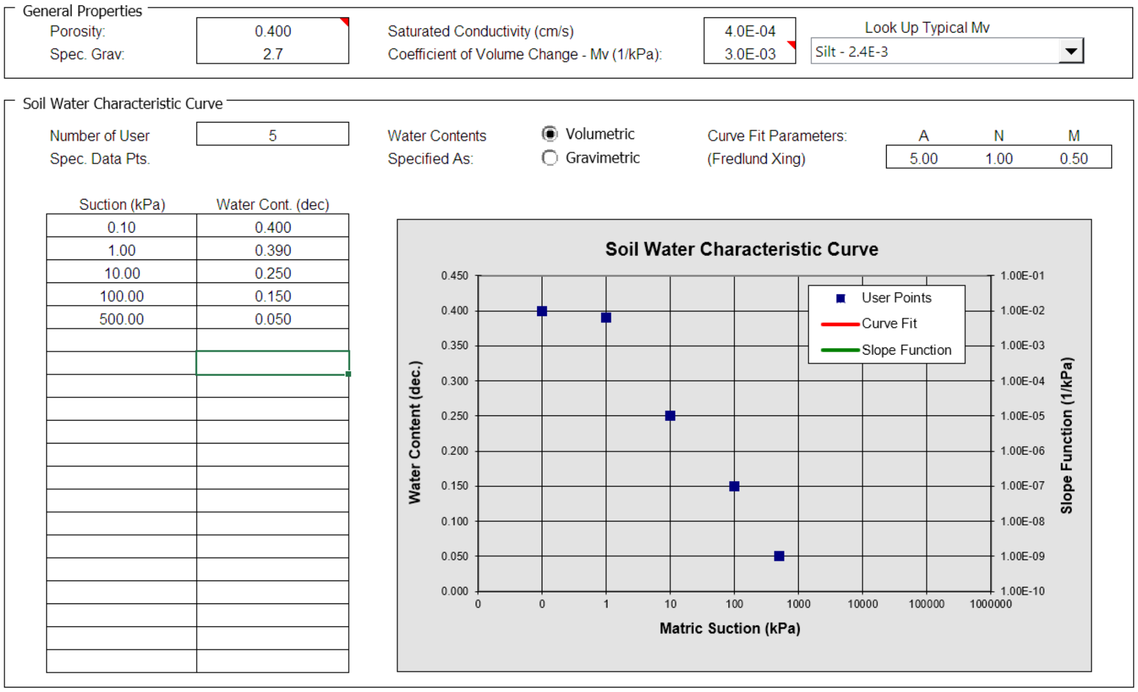
Start by assigning a name. Do not Save Changes until after you have defined each material function within a uniquely named soil.



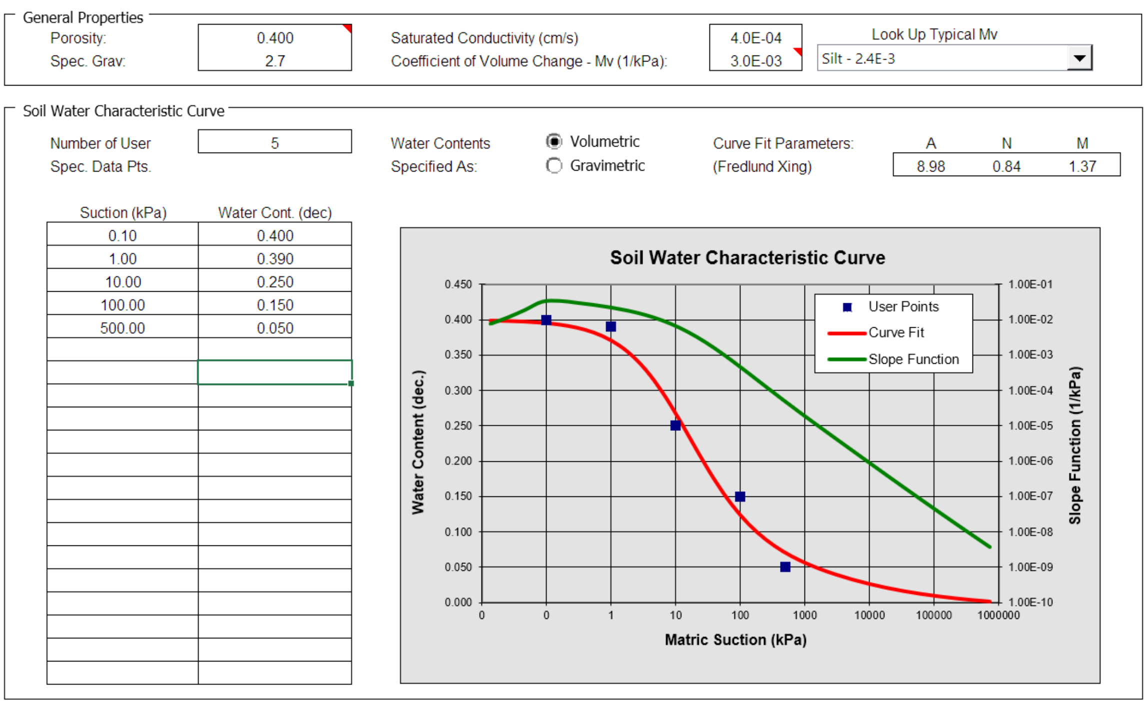
You must use each of the four buttons in the upper right to define materials for saturated / unsaturated water and heat flow. Once that is done, you save the changes prior to closing the page.

Later on when you use the drop down list for Material Handling, you can select a prior defined soil to view its properties and change them (don't forget to save changes if you make any changes) or choose an empty material name to define a new material.

The first material function you MUST create is the Soil Water Characteristic Curve (SWCC) and it is accomplished by entering data in the blank cells as shown. The other material functions use the SWCC in part to complete their estimation algorithms. The SWCC relies on user input data points which are then curve fit.



In the example above, we have specified a porosity of 0.4, a specific gravity of 2.7, and saturated conductivity and Mv as shown. After these are entered use the Curve Fit SWCC button to fit a curve through your data points and to show you the slope of that curve. The curve fit parameters A, N and M will be generated. Note that it is the slope function of the SWCC that is used in the finite element solver so ensure that the slope is a nice smooth shape. Enter more user data points and re fit the curve to fine tune the curve fit and slope function.

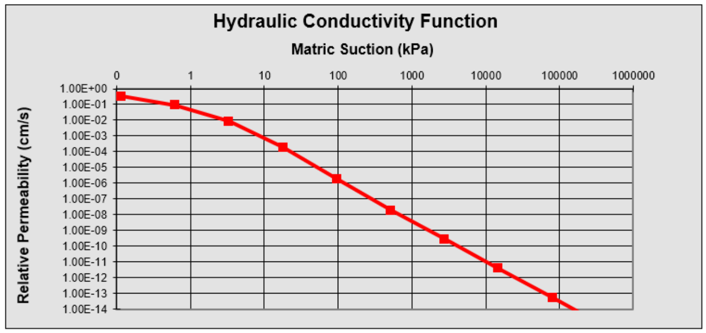


Once the SWCC is created, you can simply use the Estimate Khyd and thermal functions to generate the remaining curves.

Unsaturated Permeability Function

Data Points: 10

Suction (kPa)	Relative Conductivity (cm/s)
0.12	3.34E-01
0.62	8.73E-02
3.34	8.61E-03
17.96	1.93E-04
96.48	1.81E-06
518.25	1.88E-08
2783.81	2.69E-10
14953.49	3.92E-12
80324.03	5.49E-14
431467.91	7.69E-16

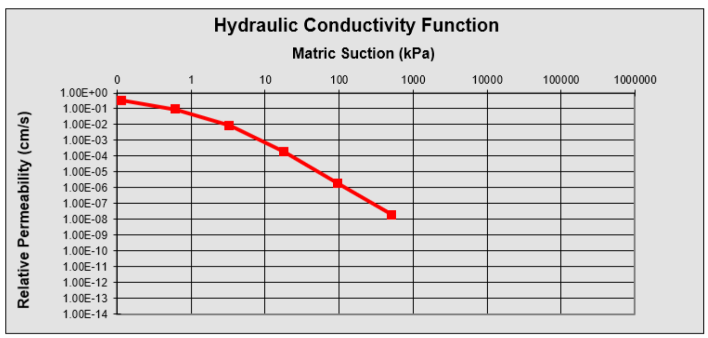


The above image shows the estimated RELATIVE Hyd K function. Notice that it has very low conductivities at high suctions. In this case, you should consider removing the last four data points and updating the # Data Points cell. Removing these high suction, very low conductivity values will help the solver accommodate infiltration into very dry soils without impacting the overall water balance of the model during drier intervals.

Unsaturated Permeability Function

Data Points: 6

Suction (kPa)	Relative Conductivity (cm/s)
0.12	3.34E-01
0.62	8.73E-02
3.34	8.61E-03
17.96	1.93E-04
96.48	1.81E-06
518.25	1.88E-08



Note that this function is the **RELATIVE** hydraulic conductivity function. The value from this function is multiplied by the Ksat value you entered above to arrive at the actual value used in the solver.

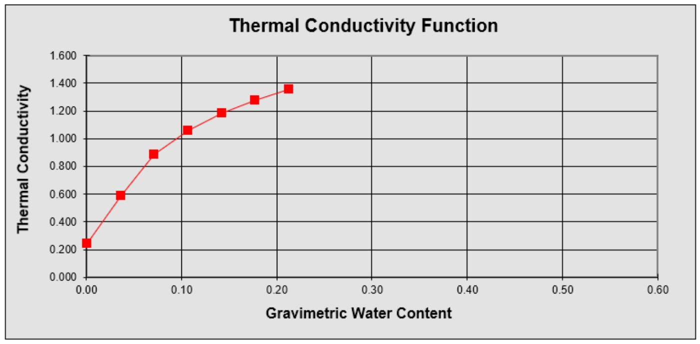
For the thermal properties, you must enter the quartz content and mass specific heat of the soil solids as shown below, prior to using the Estimate Thermal Conductivity or Estimate Heat Capacity commands.

Thermal Conductivity Function

Data Points: 7

% Quartz (dec.) 0.2

Gravimetric Water Content	Thermal Conduct. (W/mC)
0.001	0.246
0.036	0.590
0.072	0.885
0.107	1.059
0.142	1.183
0.177	1.279
0.213	1.358

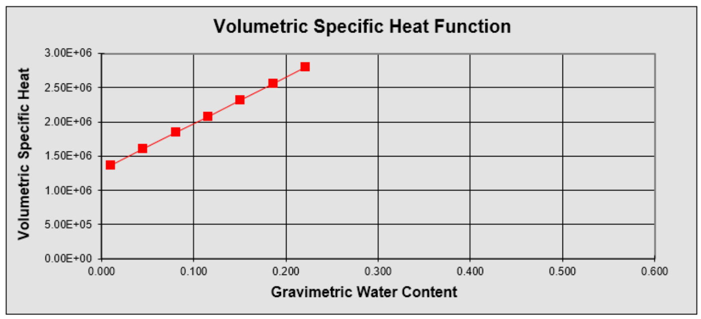


Volumetric Specific Heat Function

Data Points: 7

Solids Specific Heat (J/kg-C) 800

Gravimetric Water Content	Volumetric Sp. Ht. (J/m3-C)
0.01	1.36E+06
0.05	1.60E+06
0.08	1.84E+06
0.12	2.08E+06
0.15	2.32E+06
0.19	2.56E+06
0.22	2.80E+06



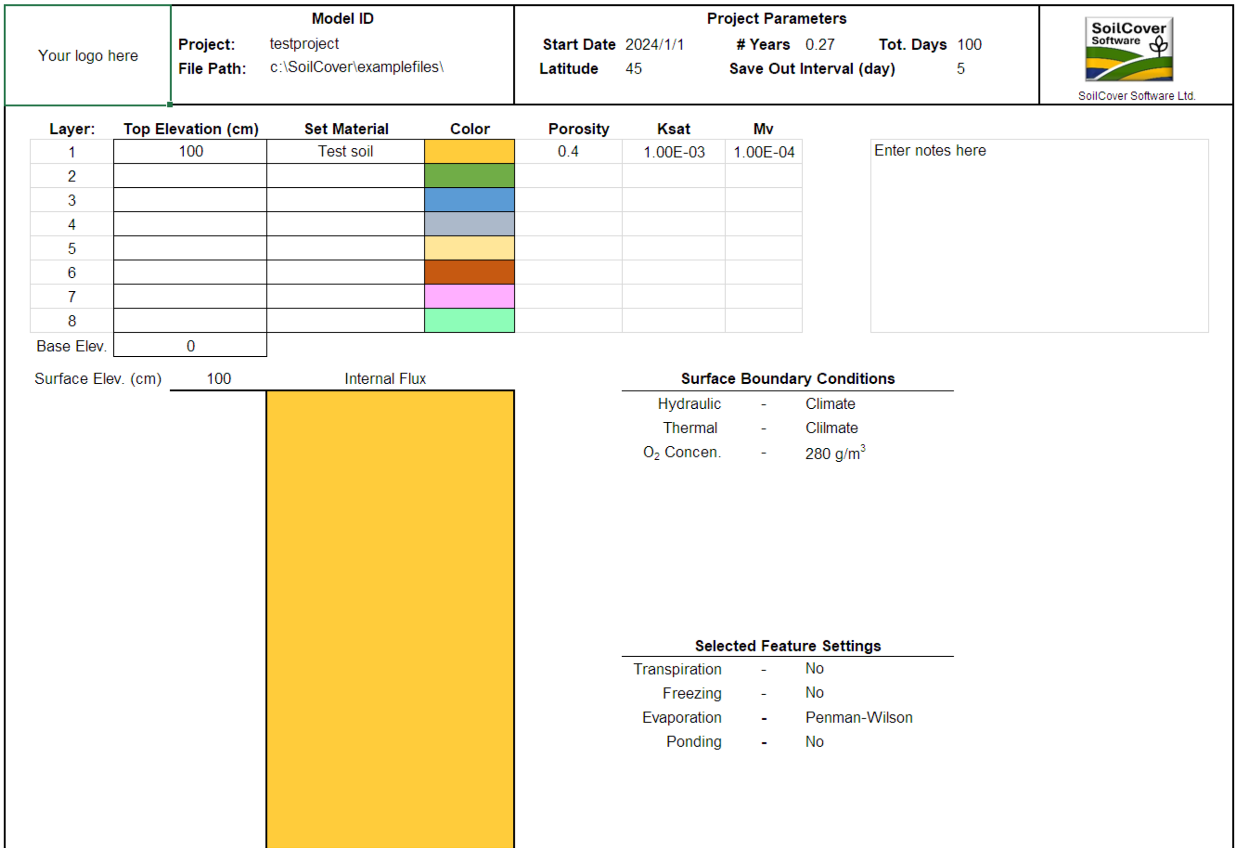
When you close the material page, you will be taken back to the Model page where you can specify the soil layer geometry and assign the defined material properties.

You must specify the top elevation of each layer and use the drop down list to specify the material that exists at that layer. The previously defined porosity and Ksat and Mv will be populated in the upper right cells of the geometry table.

The model image should update also. As you change other model parameters the other information on this model page will automatically update.

You can add your company logo to the upper left Header cell and you can add some model notes in the designated box too.

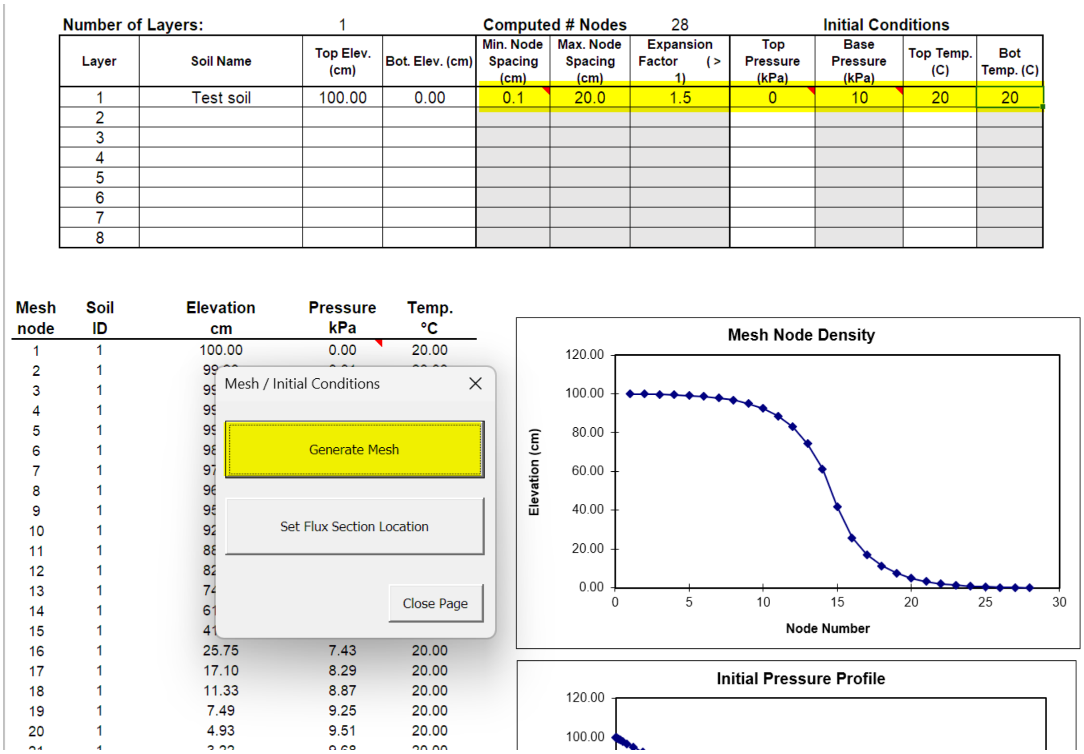
The model ID and Define parameters are updated in the Header also. By now, you should have used the SAVE command to give a project name and file location. This Model view page will confirm you have done that.



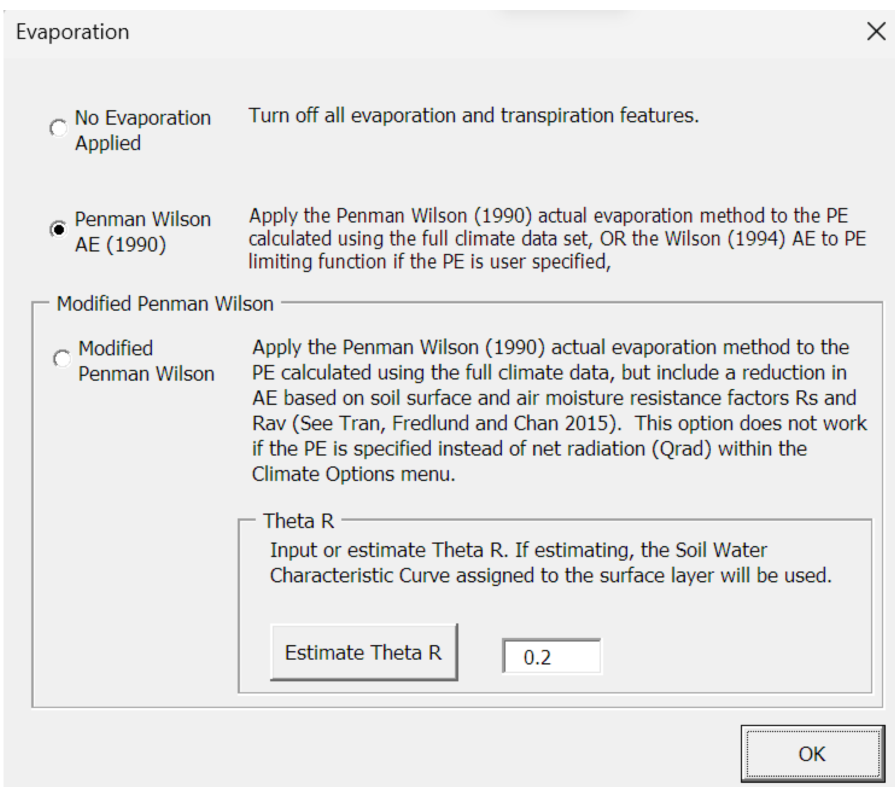
Next you must define the FEM mesh. You need to enter the mesh spacing and initial pressure and temperature conditions for each layer as shown and then use the Generate Mesh command to automatically generate the mesh node locations.

A graphic of the node spacing will appear. It shows that more nodes are located near the surface and base boundary which is good practice as this is where the boundary conditions are generally imposed.

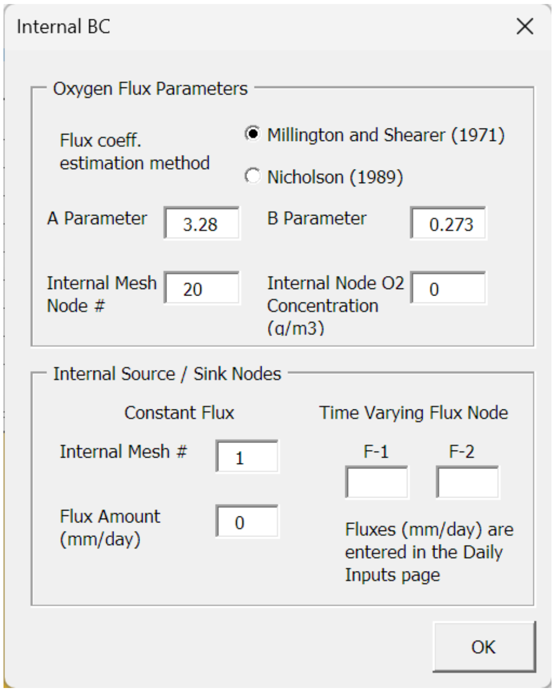
You can also use the Set Flux Section location command. This will specify a specific Node number in the FEM mesh where the software will automatically cumulate water flow volumes at. Typically you would specify the Node number corresponding to the base of your composite cover layer profile.



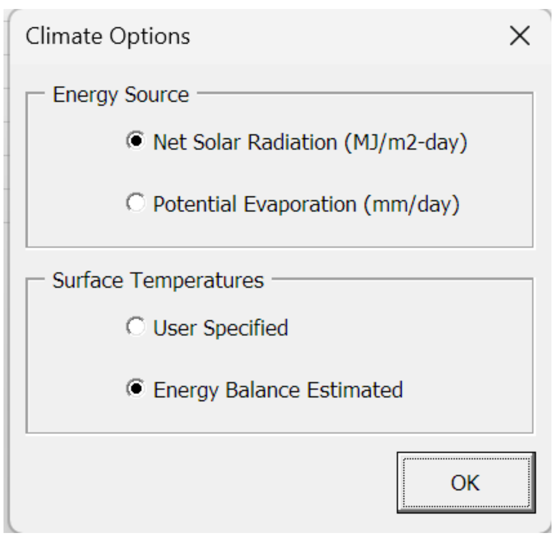
For this model, we will not specify freeze / thaw or vegetation. We must still however decide what evaporation model to apply. Here we will accept the default Penman Wilson 1990 method.



There is an option to specify internal boundary conditions and oxygen flux parameters as shown below. We will not consider these in this test model. However, if you have an internal source or sink fluxes you can specify the location and daily constant amount here, or, you can specify a location for one or two transient flux values here and then specify the flux magnitude in the daily input data page.



You must next specify how you want the surface energy balance to be estimated. You can choose options as noted below. These are described in in the Reference manual.



The most intensive data input page is the daily data page which consists of the following table.

Daily Climate Data and Solver Parameters

Hydraulic BC TYPES and value UNITS may be:
0 - Pressure (kPa) 1 - Flux (mm/day) 2 - Unit Gradient (bottom only)

		Weather Data Section									Boundary Conditions Section				Other Daily Inputs Section			
Date	Write Out Day	Max. AirTemp (C)	Min. AirTemp (C)	Net Radiation (MJ/m2/day)	Max. RH (dec)	Min. RH (dec)	Wind Speed (km/h)	Daily Precip. (mm/day)	Rain Start (Hrs)	Rain Stop (Hrs)	Bottom Hyd. BC (Type)	Bottom Hyd. BC (Value)	Top Temp (C)	Bottom Temp (C)	Root Depth D-1 (cm)	Root Depth D-2 (cm)	Inner Flux F-1 (mm/day)	Inner Flux F-2 (mm/day)
Initial	1	0.00	0.00	0.00	0.00	0.00	0.00	0.0	0	0	0	0.0	0.0	0.0	0	0	0.0	0.0
2024-01-01		0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-02	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-03	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-04	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-05	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-06	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-07	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-08	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-09	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-10	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-11	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-12	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-13	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-14	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-15	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-16	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-17	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-18	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-19	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-20	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-21	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-22	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-23	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-24	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-25	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-26	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-27	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-28	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2024-01-29	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0

The date column is generated automatically and there are as many rows of data as the duration of the model you specified. In this example, the "write out" days are every 5th day as you specified earlier. You must now enter date in the other appropriate columns. Note that the Top Temp column is greyed out. This is because you selected to have top temperatures estimated from surface energy balance calculations.

For the bottom boundary conditions you can set a Type as pressure, flux, or unit gradient. You then set the value with the units as noted in the pink information box at the top of the table.

For a simple model it is easiest to enter data on the second row (not the first greyed out row) and then copy / paste the data to the 100th row. As shown below. Be careful NOT to copy the first two columns of data as these are set internally by date and write out data frequency.

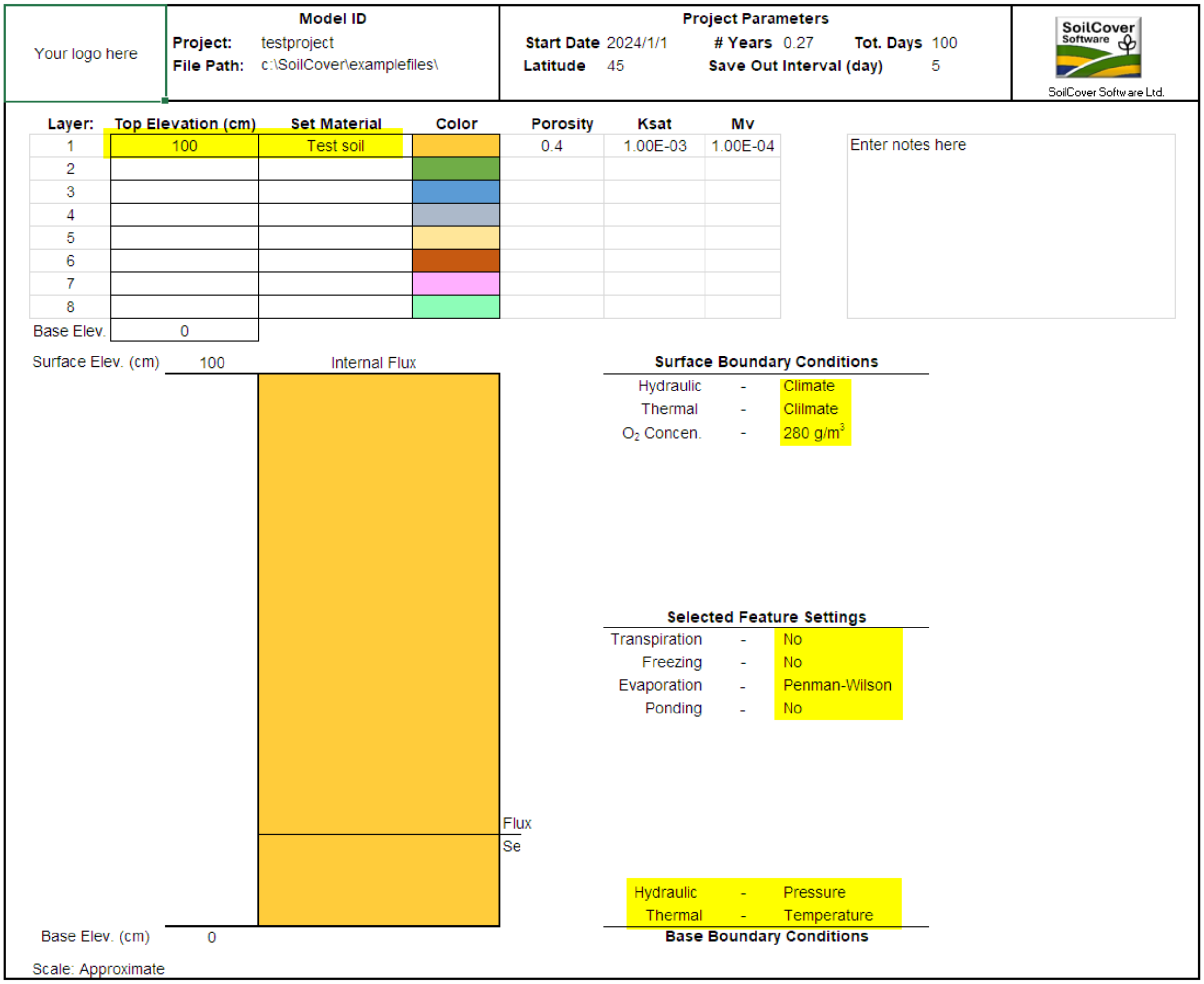
Daily Climate Data and Solver Parameters

Hydraulic BC TYPES and value UNITS may be:
0 - Pressure (kPa) **1** - Flux (mm/day) **2** - Unit Gradient (bottom only)

		Weather Data Section									Boundary Conditions Section				Other Daily Inputs Section			
Date	Write Out Day	Max. AirTemp (C)	Min. AirTemp (C)	Net Radiation (MJ/m2/day)	Max. RH (dec)	Min. RH (dec)	Wind Speed (km/h)	Daily Precip. (mm/day)	Rain Start (Hrs)	Rain Stop (Hrs)	Bottom Hyd. BC (Type)	Bottom Hyd. BC (Value)	Top Temp (C)	Bottom Temp (C)	Root Depth D-1 (cm)	Root Depth D-2 (cm)	Inner Flux F-1 (mm/day)	Inner Flux F-2 (mm/day)
Initial	1	0.00	0.00	0.00	0.00	0.00	0.00	0.0	0	0	0	0.0	0.0	0.0	0	0	0.0	0.0
2024-01-01	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-02	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-03	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-04	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-05	1	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-06	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-07	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-08	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-09	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-10	1	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-11	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-12	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-13	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-14	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-15	1	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-16	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-17	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-18	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-19	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-20	1	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-21	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-22	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-23	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-24	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-25	1	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-26	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-27	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-28	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0
2024-01-29	0	20	10	12	0.8	0.6	2	0	0	0	0	10	0	20	0	0	0	0

The convergence and time step control data are typically left at their default values, however they can be adjusted if you need to tighten up the parameters to help with convergence.

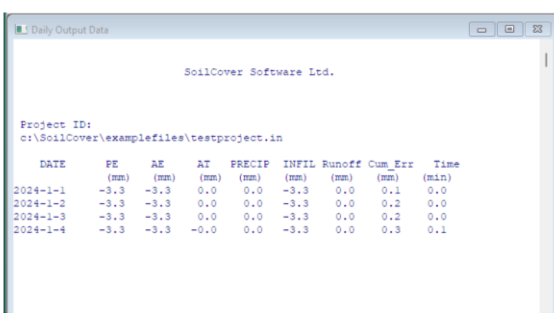
The final step before solving is to check over the model page, confirm soil properties are applied as desired, check the type of boundary conditions and special feature settings etc.

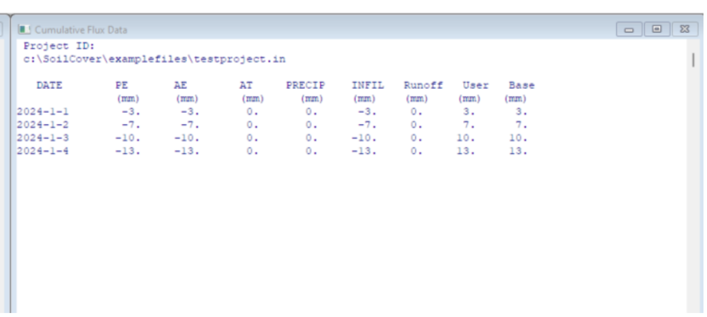


Prior to solving, it is a good idea to SAVE the file one more time.

When you next press the SOLVE command button, some data checks will occur internally and error messages provided if any.

The solver will then launch and a daily and cumulative flux table will start to develop on screen as shown below.





You must wait until the solver completes, or, if there are problems with the data shown, you can cancel the solver by closing the windows and then proceed to check all inputs and model parameters.

Before you close the solver screen, it is a good idea to check the following computed values.

DATE	PE (mm)	AE (mm)	AT (mm)	PRECIP (mm)	INFIL (mm)	Runoff (mm)	Cum_Err (mm)	Time (min)
2024-3-30	-3.4	-3.4	-0.0	0.0	-3.4	0.0	8.9	0.7
2024-3-31	-3.4	-3.4	-0.0	0.0	-3.4	0.0	9.0	0.7
2024-4-1	-3.3	-3.3	-0.0	0.0	-3.3	0.0	9.1	0.7
2024-4-2	-3.3	-3.3	-0.0	0.0	-3.3	0.0	9.3	0.7
2024-4-3	-3.3	-3.3	-0.0	0.0	-3.3	0.0	9.4	0.8
2024-4-4	-3.4	-3.4	0.0	0.0	-3.4	0.0	9.5	0.8
2024-4-5	-3.3	-3.3	0.0	0.0	-3.3	0.0	9.6	0.8
2024-4-6	-3.4	-3.4	-0.0	0.0	-3.4	0.0	9.8	0.8
2024-4-7	-3.3	-3.3	-0.0	0.0	-3.3	0.0	9.9	0.8
2024-4-8	-3.4	-3.4	0.0	0.0	-3.4	0.0	10.0	0.8
2024-4-9	-3.3	-3.3	0.0	0.0	-3.3	0.0	10.2	0.8

Completed. Use the project workbook to analyse your results.

Total run time = 0.80 minutes

2024-3-29	-295.	-295.	0.	0.	-295.	0.	296.	287.

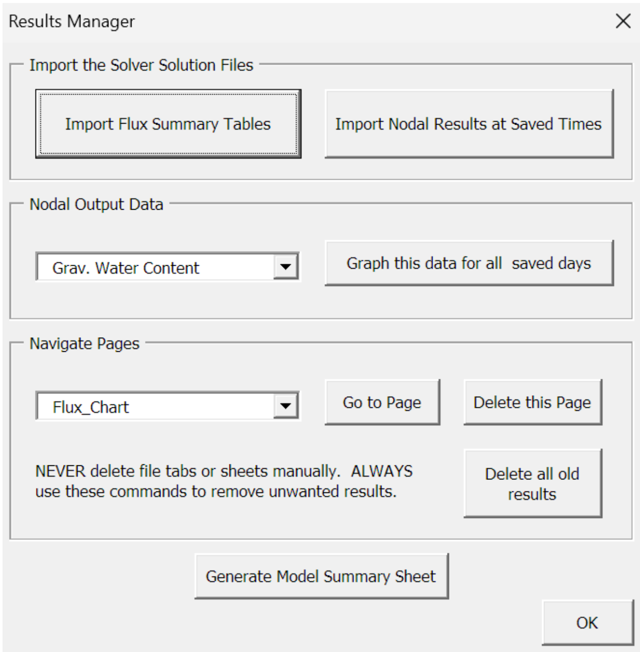
Project ID:
c:\SoilCover\examplefiles\testproject.in

DATE	PE (mm)	AE (mm)	AT (mm)	PRECIP (mm)	INFIL (mm)	Runoff (mm)	User (mm)	Base (mm)
2024-3-30	-299.	-299.	0.	0.	-299.	0.	299.	290.
2024-3-31	-302.	-302.	0.	0.	-302.	0.	302.	293.
2024-4-1	-305.	-305.	0.	0.	-305.	0.	306.	296.
2024-4-2	-309.	-309.	0.	0.	-309.	0.	309.	299.
2024-4-3	-312.	-312.	0.	0.	-312.	0.	312.	303.
2024-4-4	-315.	-315.	0.	0.	-315.	0.	316.	306.
2024-4-5	-319.	-319.	0.	0.	-319.	0.	319.	309.
2024-4-6	-322.	-322.	0.	0.	-322.	0.	322.	312.
2024-4-7	-325.	-325.	0.	0.	-325.	0.	326.	316.
2024-4-8	-329.	-329.	0.	0.	-329.	0.	329.	319.
2024-4-9	-332.	-332.	0.	0.	-332.	0.	332.	322.

Here in this example the cumulative water balance error is about 10mm, and the net cumulative infiltration is about -332 mm (negative at surface is water leaving the system), with a total base boundary flux of 322 m (positive is water entering the system). The difference between the two in this case, is about 10mm, which suggests that the saturated initial conditions of the 1m profile along with no assigned infiltration and daily evaporation of about 3.3mm, result in evaporation drawing water up into the base. Over the 100 day simulation about 3.2 % of that water was not accounted for. Of a total of 332 mm evaporation, there is an error of 10.2mm which is 10.2 / 332 = 0.032 or 3.2%. This error could be reduced by adjusting the FEM mesh spacing, smoothing the SWCC curve some more, adjusting the Mv value to be more appropriate for this soil etc.

This data can be viewed in the Results section of the user interface as well. You can close the two Solver table windows and proceed to use the internal Results tools.

All of the results investigations are initiated in SoilCover using the following Results Manager dialogue box.



The first step is to import the two output files written by the FEM solver. One file is a Flux Summary Table, and the other is Nodal parameter results at every save out timestep.

Typically you will first import the flux summary tables and generate a Simulation Summary Page using the upper left command. This will provide the following information.

SoilCover Simulation Summary Page

1. Project Name **testproject**

2. Project Directory **c:\SoilCover\examplefiles**

3. Run Parameters

Transpiration - No
Freezing - No
Evaporation Method - Penman-Wilson

Start Date - 2024/1/1
Years - 0.27
Model Days: 100

Start Day in Yr - 1

4. Mesh Information

Convergence Criteria

Max. Iterations	Max.Change Suction (%)	Max.Change Temperature (%)	Suction Dampening (%)	Temperature Dampening (%)
10	1	1	0	0

Time Step Control

Max.Change Suction (%)	Max.Change Temperature (%)	Minimum Time Step (secnds)	First Time Step (secnds)	Maximum Time Step (secnds)
1	1	10	10	3600

Soil Profile Data

Number of Nodes	Number of Layers	Internal BC Node	Internal BC Value (mm/day)
28	1	1	0

5. Soil Property Summary

Soil Name	Porosity	Spec. Grav.	Mv (1/kPa)	Ksat (cm/s)
Test soil	0.4	2.7	1E-04	0.001

6. Boundary Conditions

First date of run each year	2024/1/1
Total run days/year	100
Top thermal BC type	Cllmate
Bottom thermal BC value (C)	0
Day 1 top BC type	Climate
Day 1 bottom BC type	Pressure
Day 1 bottom BC value	0.00

7. Transpiration Summary

Moisture limiting point (kPa)
Moisture wilting point (kPa)
Grass quality

First date of growing season
Last date of growing season
First day root depth (cm)

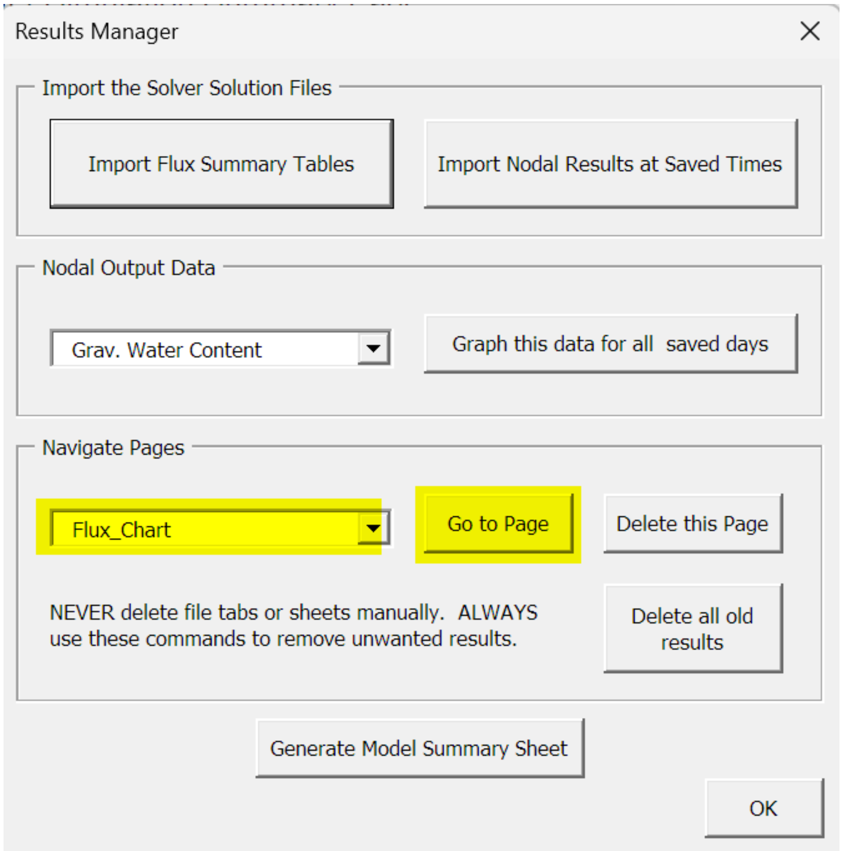
8. Run Output Summary

Cumulative precipitation (mm)	0
Cumulative bottom flux (mm)	322
Cumulative PE (mm)	-332.15
Cumulative PT (mm)	0
Cumulative ET (mm)	-332.2
Cumulative user source / sink flux (mm)	0

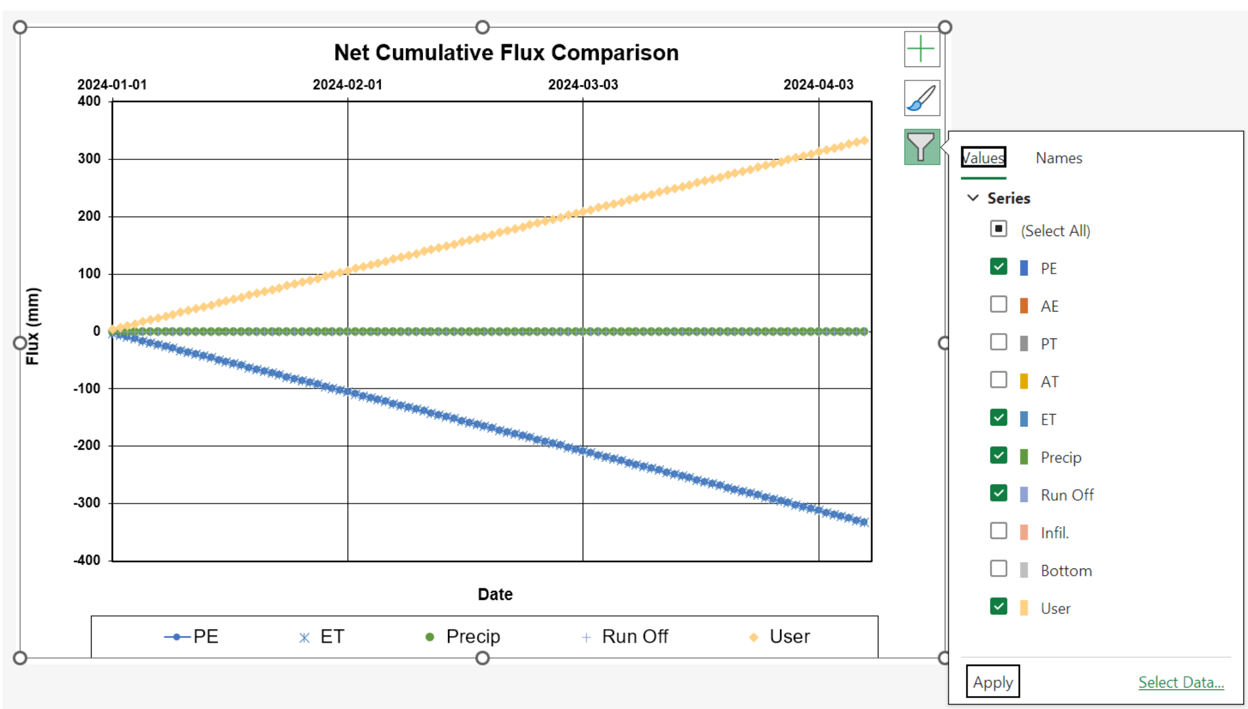
Cumulative infiltration (mm)	-332.2
Cumulative runoff (mm)	0
Cumulative AE (mm)	-332.2
Cumulative AT (mm)	0
Cumulative flux section (mm)	332.4
Flux section node	17
Flux section elevation (cm)	17.10

This will automatically also create a flux summary chart which you can access using the Navigate Pages drop down menu. This menu is how you will navigate to any results pages or charts you generate.

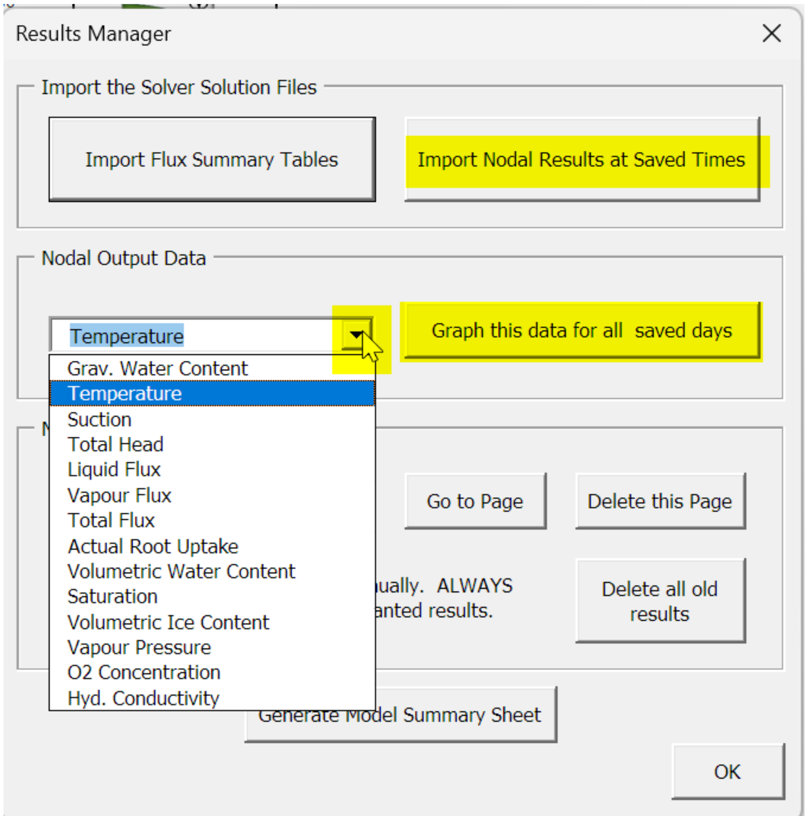
Choose the Flux_Chart page and hit the Go To Page button to view the flux summary chart.



You can double click the vertical and horizontal axis to change the grid size and then use the Filter drop down list to choose which data to view on the chart.

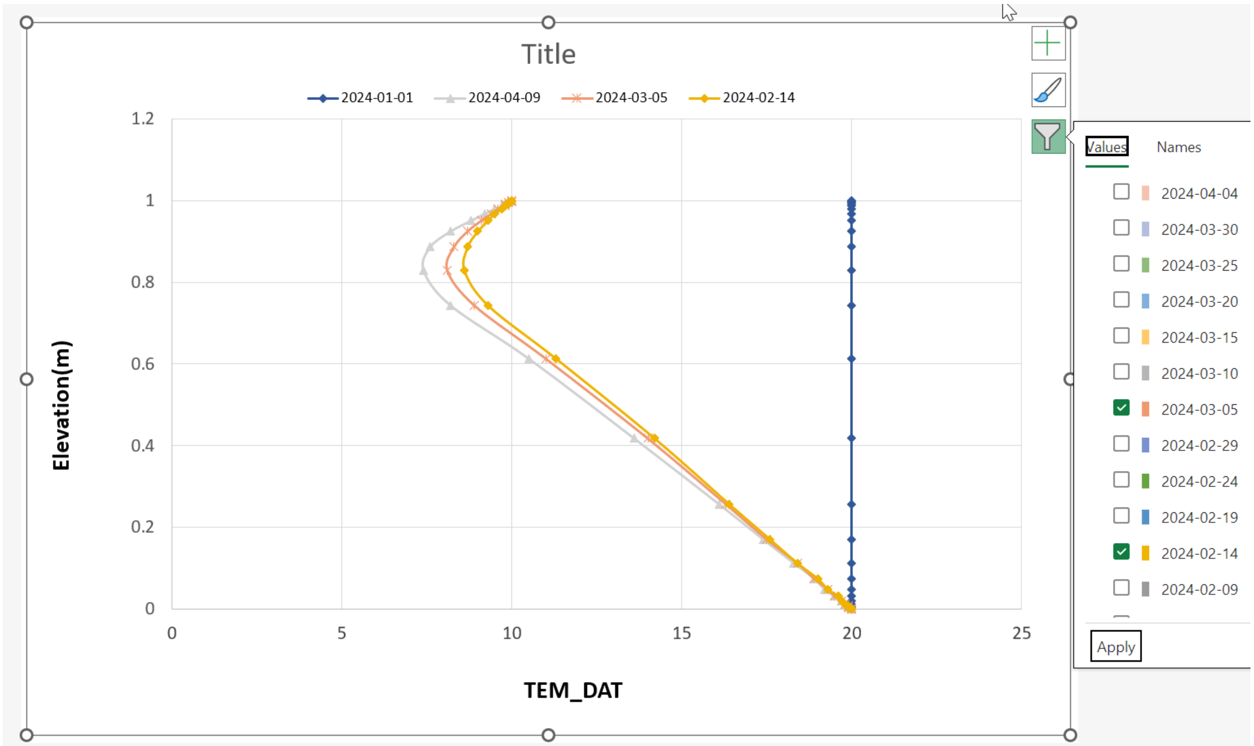


You can also use the Nodal output data commands to view profile graphs of any parameter on any day AFTER you have **imported the nodal results at saved times** using the command in the upper right corner.



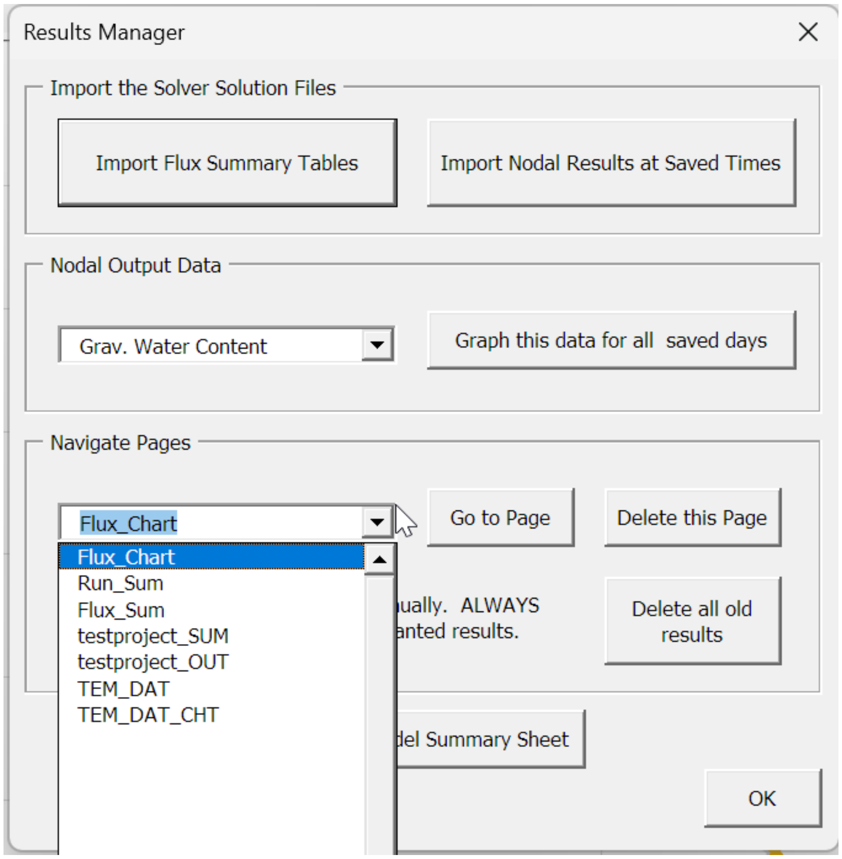
All of the available nodal data is listed in the Nodal Output Data drop down list. In this example, we have highlighted Temperature to generate a profile of all temperatures at each model Save date.

You must use the Filter option to choose which dates to view and you can also use the Excel features to edit the title and change the axis grids etc.



As you create more pages you may need to manage them, delete some etc. You can view the pages you have created and deleted the selected pages using the commands shown below.

You can also delete all the charts in one step using the Delete All Old Results command button.



If you want to see the raw solver output data, navigate to the Run_Sum or Flux_Sum pages. Explore the other pages as well.

That's about it. The best way to use the software is to explore options and understand the theoretical basis for the model that is provided in the Reference Manual.

Please note that there are some error notices on occasion. While we have tried to find them all we will have missed some. If you find one that is preventing you from completing your model, please email us at support@soilcover.ca.