

# Get Started

- To bring up the graphical user interface (GUI) of the ecosys program, you can either double click over the "EcosysGUI.exe" icon on your desktop, or click Start-->All Programs-->EcosysGUI.exe. The GUI is shown below:

The screenshot displays the Ecosys Graphical User Interface (GUI) with the following components:

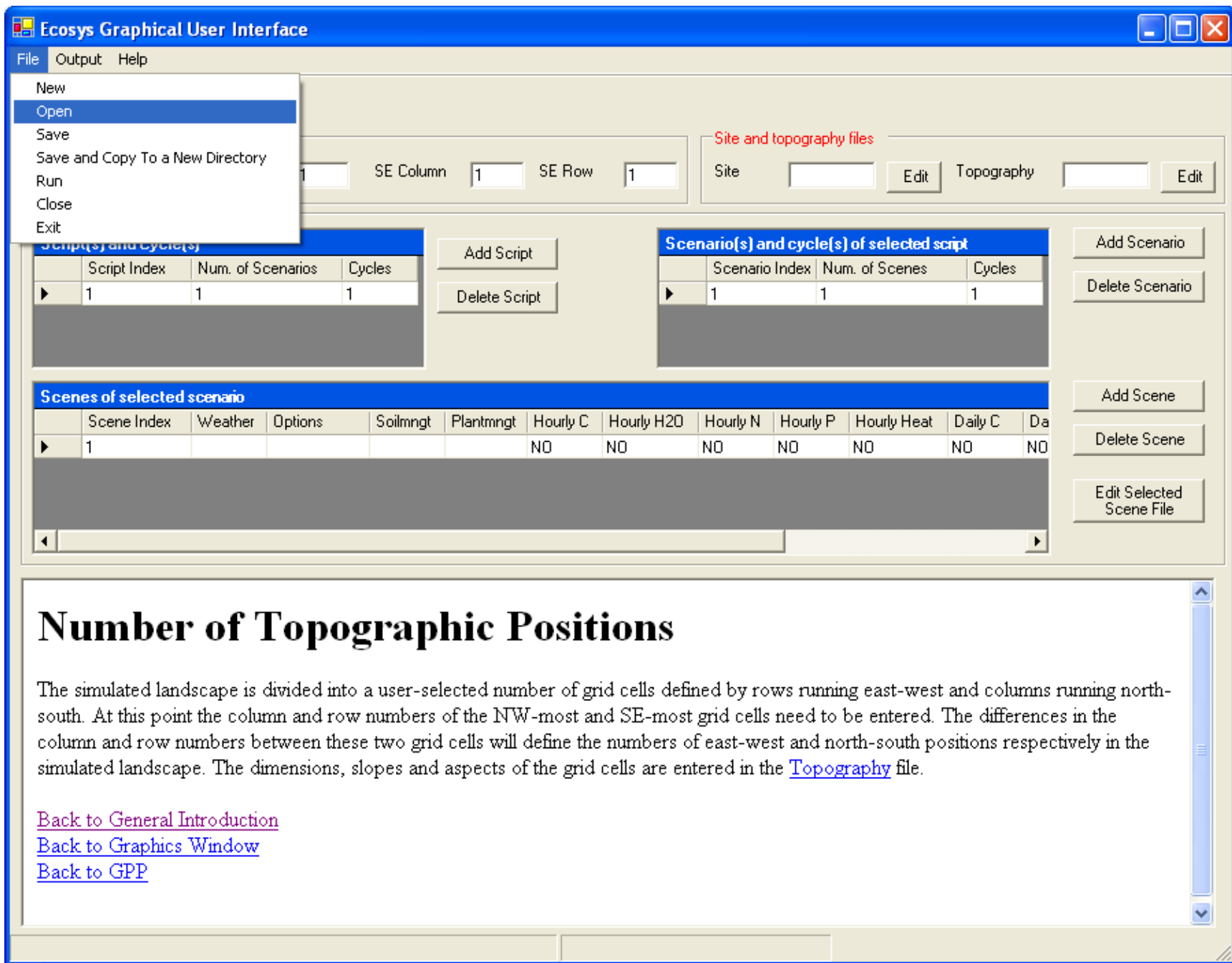
- Location of Landscape:** NW Column (1), NW Row (1), SE Column (1), SE Row (1).
- Site and topography files:** Site (text field), Edit, Topography (text field), Edit.
- Script Information:**
  - Script(s) and Cycle(s):** Table with columns: Script Index, Num. of Scenarios, Cycles. Row 1: 1, 1, 1.
  - Buttons:** Add Script, Delete Script.
- Scenario(s) and cycle(s) of selected script:** Table with columns: Scenario Index, Num. of Scenes, Cycles. Row 1: 1, 1, 1.
- Buttons:** Add Scenario, Delete Scenario.
- Scenes of selected scenario:** Table with columns: Scene Index, Weather, Options, Soilmngt, Plantmngt, Hourly C, Hourly H2O, Hourly N, Hourly P, Hourly Heat, Daily C, Da. Row 1: 1, (empty), (empty), (empty), (empty), NO, NO, NO, NO, NO, NO, NO.
- Buttons:** Add Scene, Delete Scene, Edit Selected Scene File.

**Number of Topographic Positions**

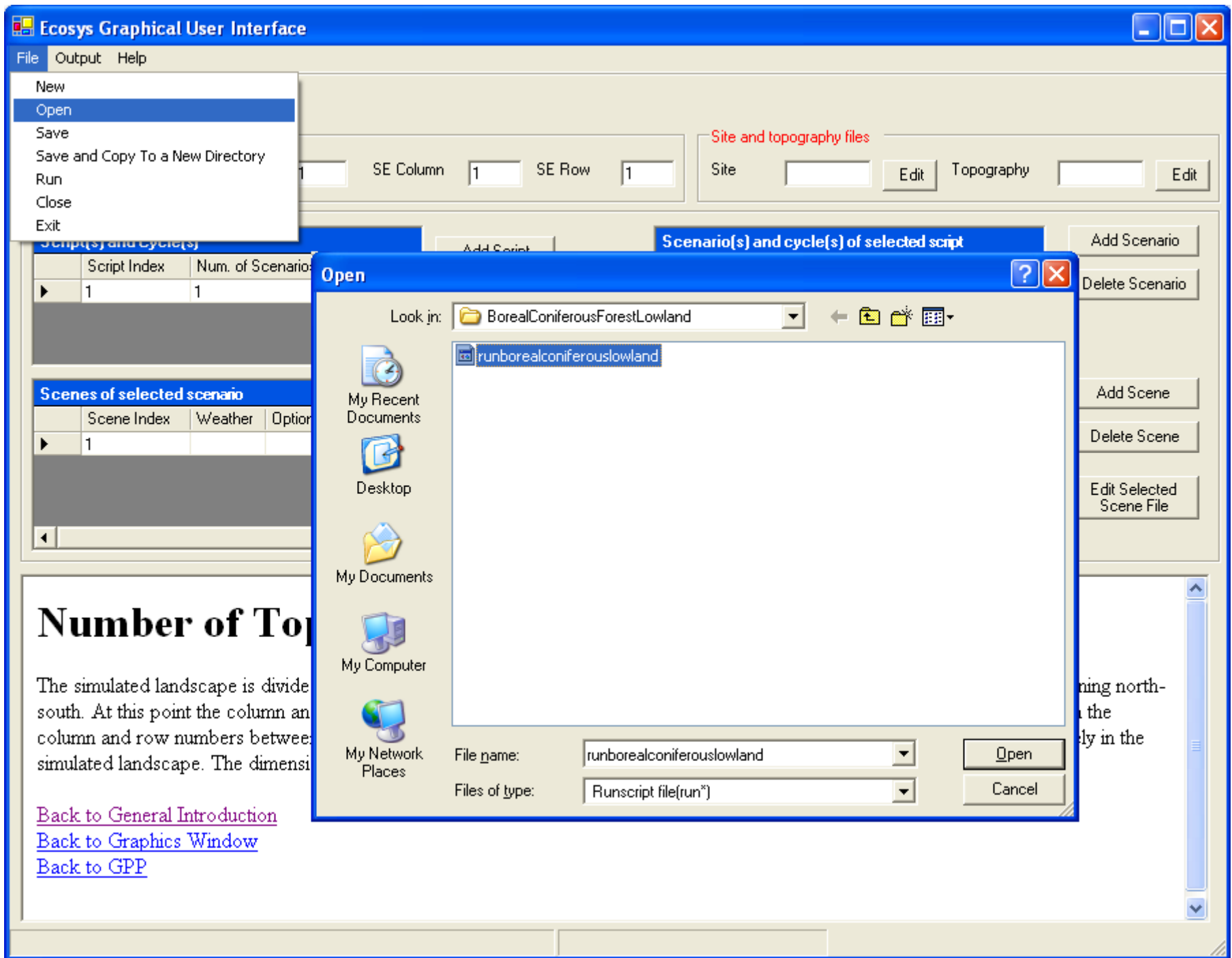
The simulated landscape is divided into a user-selected number of grid cells defined by rows running east-west and columns running north-south. At this point the column and row numbers of the NW-most and SE-most grid cells need to be entered. The differences in the column and row numbers between these two grid cells will define the numbers of east-west and north-south positions respectively in the simulated landscape. The dimensions, slopes and aspects of the grid cells are entered in the [Topography](#) file.

[Back to General Introduction](#)  
[Back to Graphics Window](#)  
[Back to GPP](#)

- Click File menu on the above window, you will see the following window



- Click Open on the above window, you will see the following window which allows you to open a run script file



- After open a run script file, you will see a window with input data. You can click over "First Scenario", "Next Scenario", "Previous Scenario", "Last Scenario" to navigate each scenarios in this run script. When you navigate the scenario, the scenes associated with the current scenario are updated too. In addition, you can delete a scenario, add a new scenario, delete a scene, add a new scene, and edit a scene file by clicking corresponding button on the window.

**Ecosys Graphical User Interface**

File Output Help

Location of Landscape: NW Column  NW Row  SE Column  SE Row

Site and topography files: Site  Edit Topography  Edit

Script Information

Script(s) and Cycle(s)			
Script Index	Num. of Scenarios	Cycles	
▶ 1	2	1	

Add Script Delete Script

Scenario(s) and cycle(s) of selected script			
Scenario Index	Num. of Scenes	Cycles	
▶ 1	1	1	
2	11	10	

Add Scenario Delete Scenario

Scenes of selected scenario												
Scene Index	Weather	Options	Soilmngt	Plantmngt	Hourly C	Hourly H2O	Hourly N	Hourly P	Hourly Heat	Daily C	Da	
▶ 1	o196w	ob80y2	NO	ob80p	NO	NO	NO	NO	NO	sccd1	scv	

Add Scene Delete Scene Edit Selected Scene File

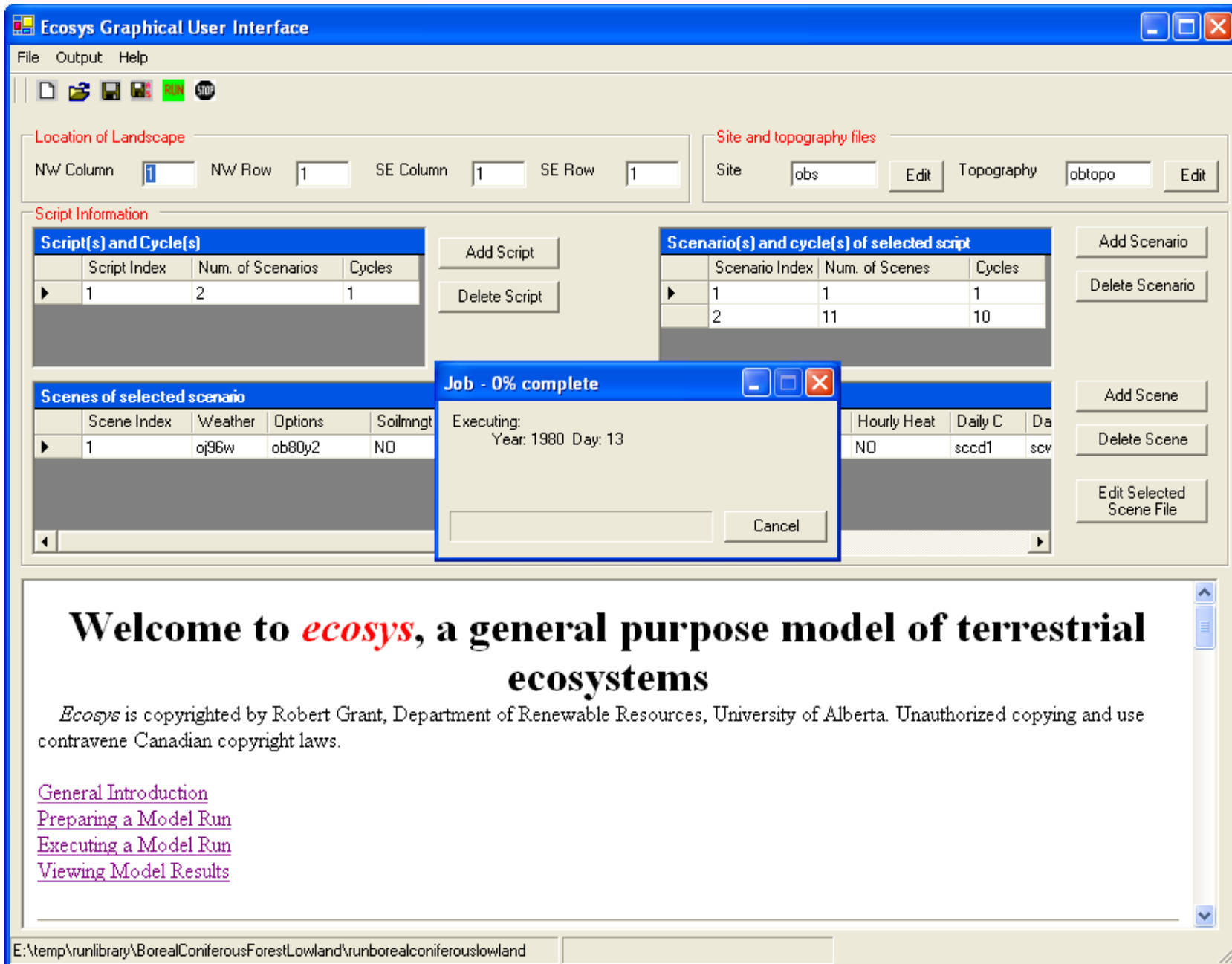
## Welcome to *ecosys*, a general purpose model of terrestrial ecosystems

*Ecosys* is copyrighted by Robert Grant, Department of Renewable Resources, University of Alberta. Unauthorized copying and use contravene Canadian copyright laws.

[General Introduction](#)  
[Preparing a Model Run](#)  
[Executing a Model Run](#)  
[Viewing Model Results](#)

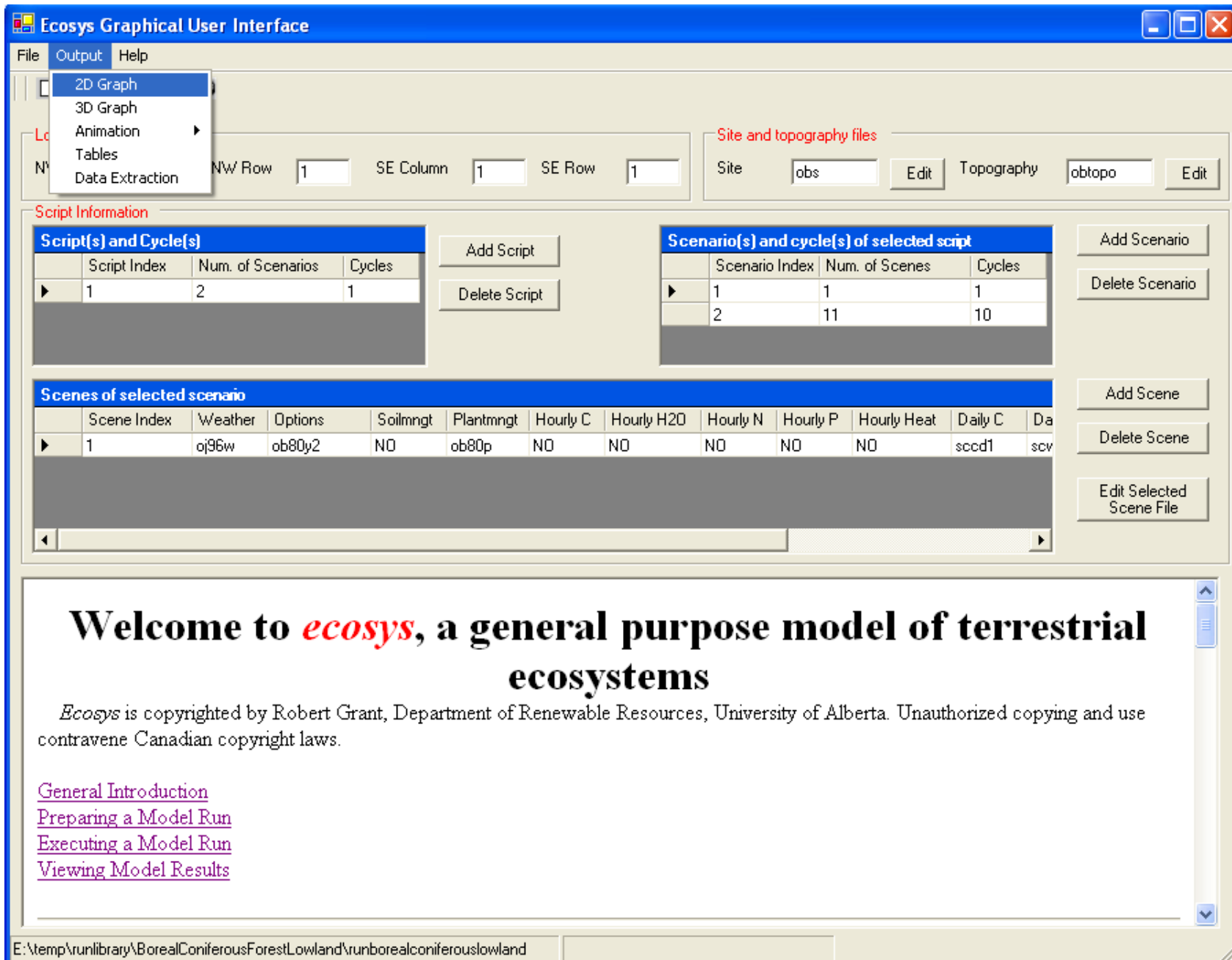
E:\temp\runlibrary\BorealConiferousForestLowland\runborealconiferouslowland

- Click "Run" button, if there is nothing wrong in the run script file, then you will see the following window showing the progress of the execution of the ecosys program.

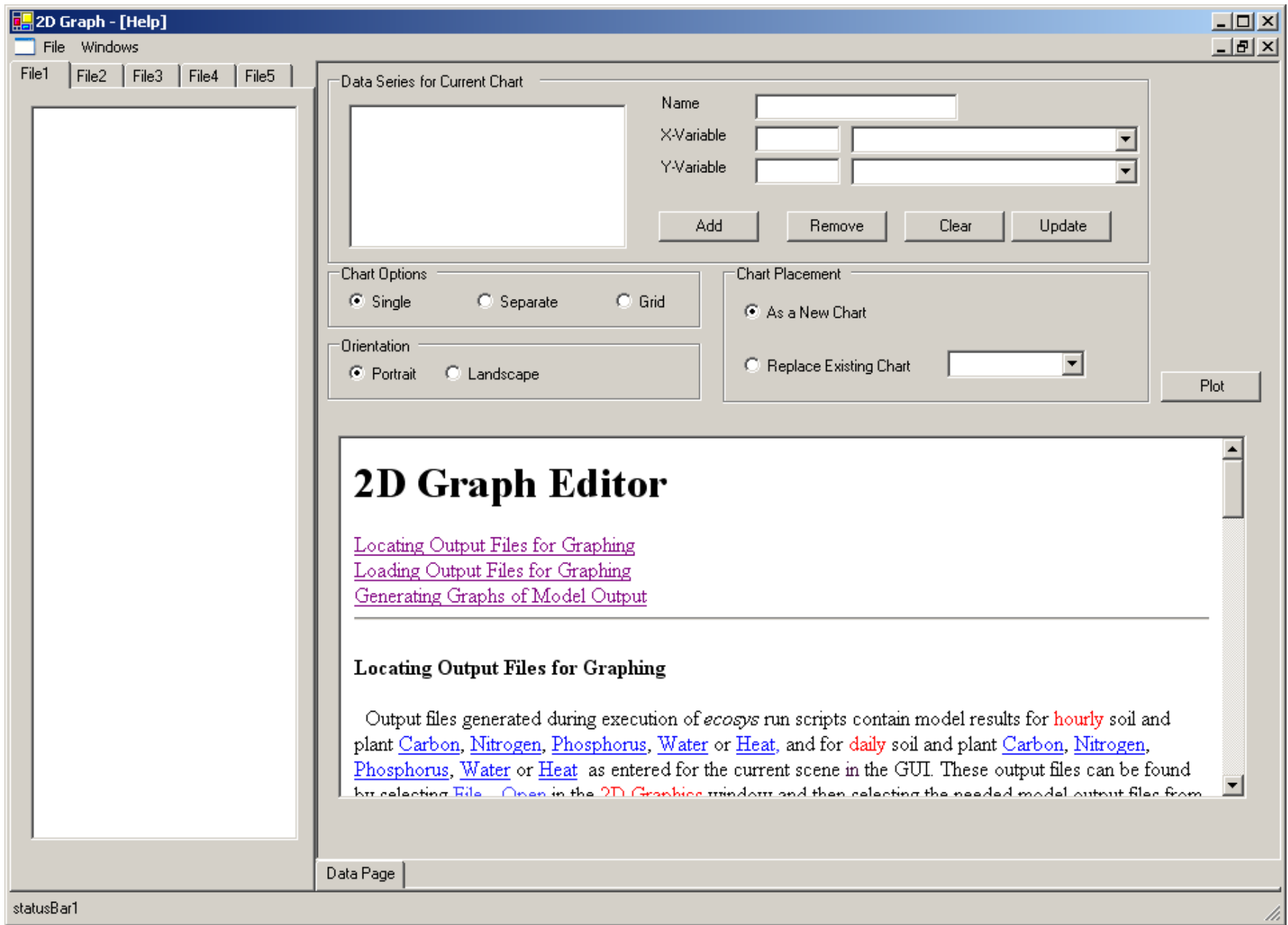


To visualize the output window, you need to do the followings:

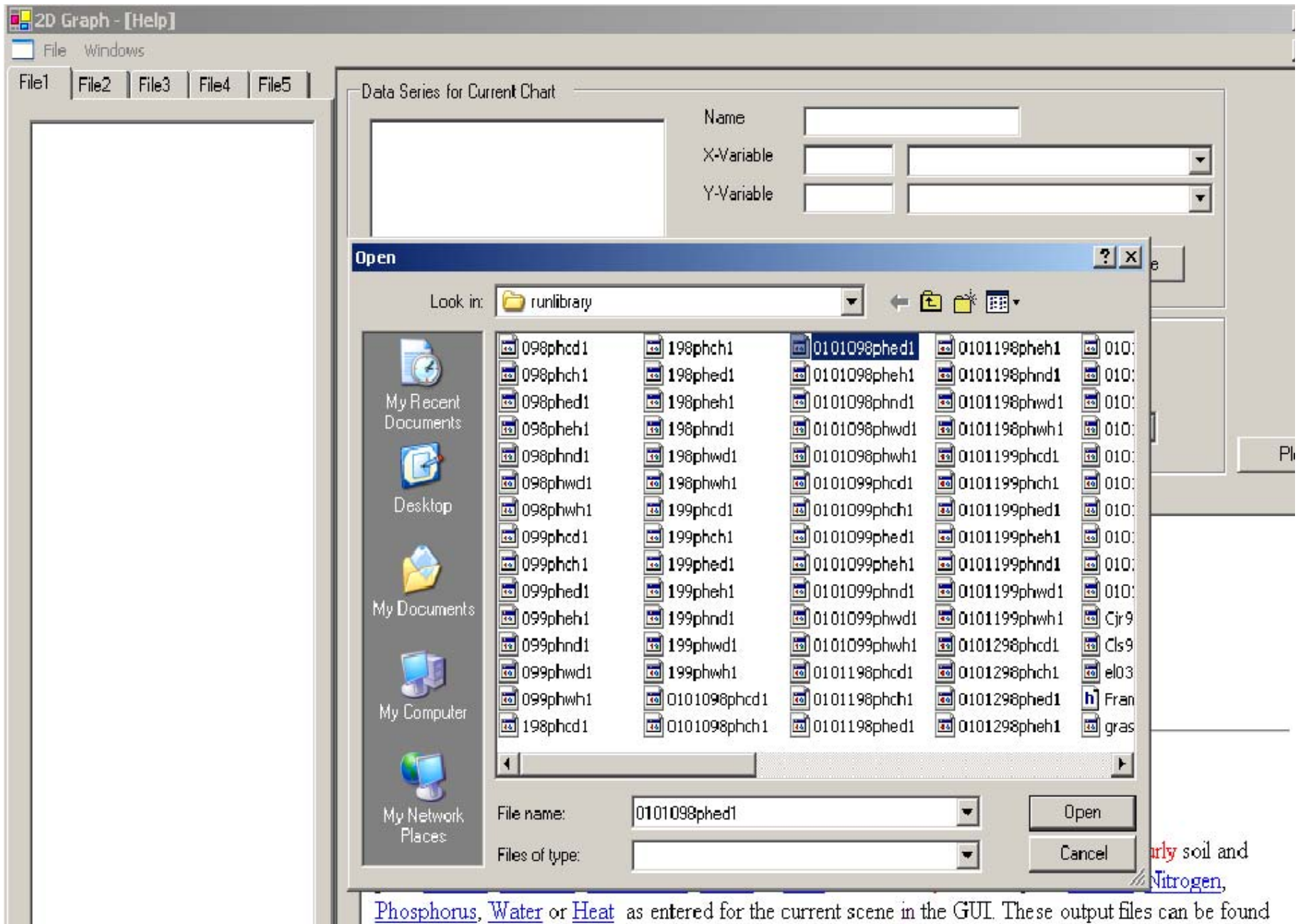
- Go to the main window, and click "Visualize" menu and select "2D Graph", which is shown below:



- After clicking 2D Graph from the above window, you will see:

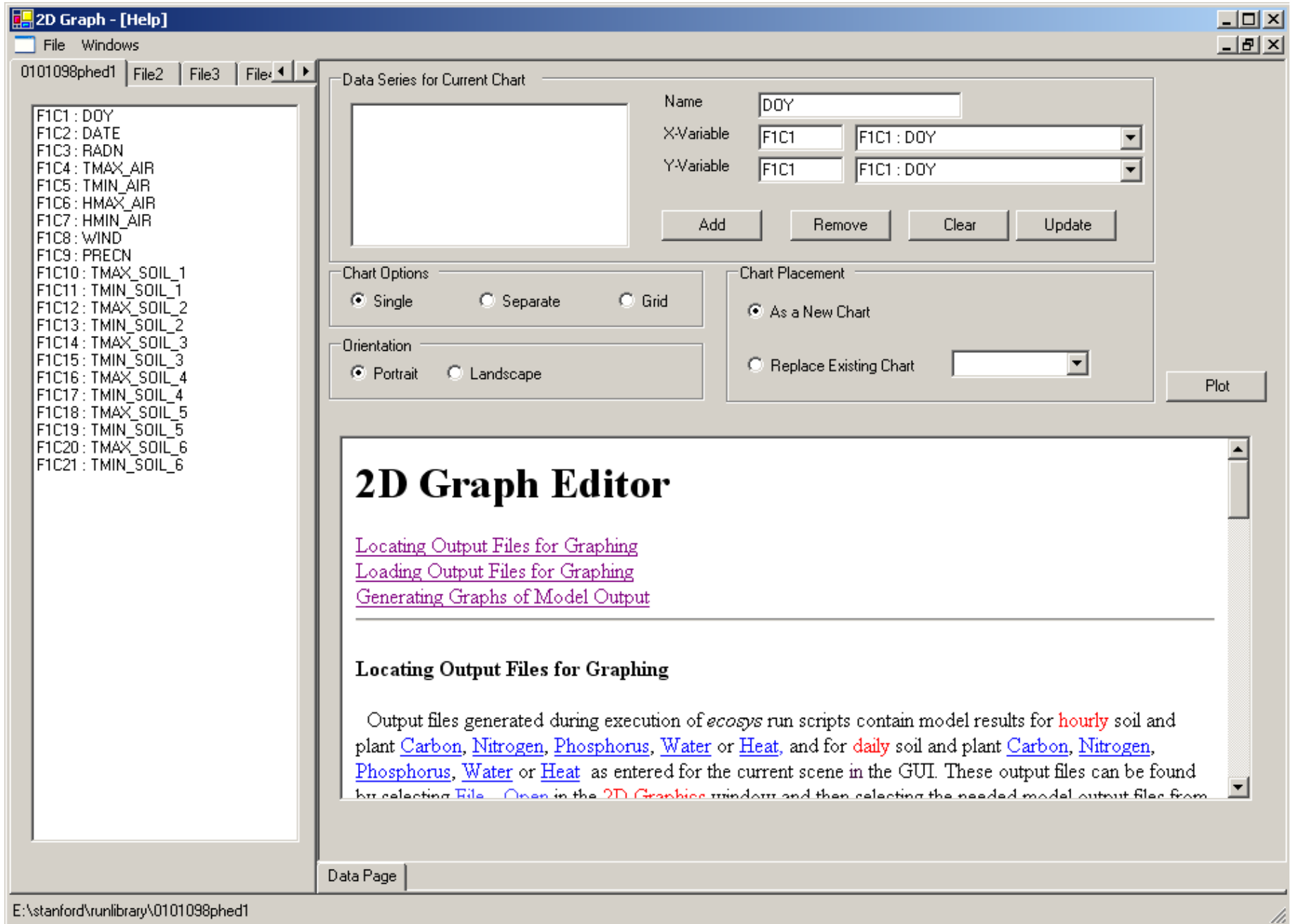


- From the above window, select File menu followed by Open, then you will see a File Open Dialog which allows you select output file to visualize. Please navigate to the folder where you execute the ecosys program. All output files are stored there. All output file names begin with 7 digits. Please refer to the help document to learn about meanings of these digits.





- After open a output file, the window looks like below. In the following window, you can open up to five output files. In the left panel, "F1C1" means Column 1 of File 1, similary, "F2C10" means Column 10 of File 2



- To create a plot, always choose column 1 (i.e., F1C1, F2C1, FnC1...) as x axis. Y axis can be any columns, say, F1C5, F2C4 etc. Please select columns of interest to you to plot from the right panel of the following window. As an example, I chose F1C5 as my y data. After selection, click Add button.

**2D Graph Editor**

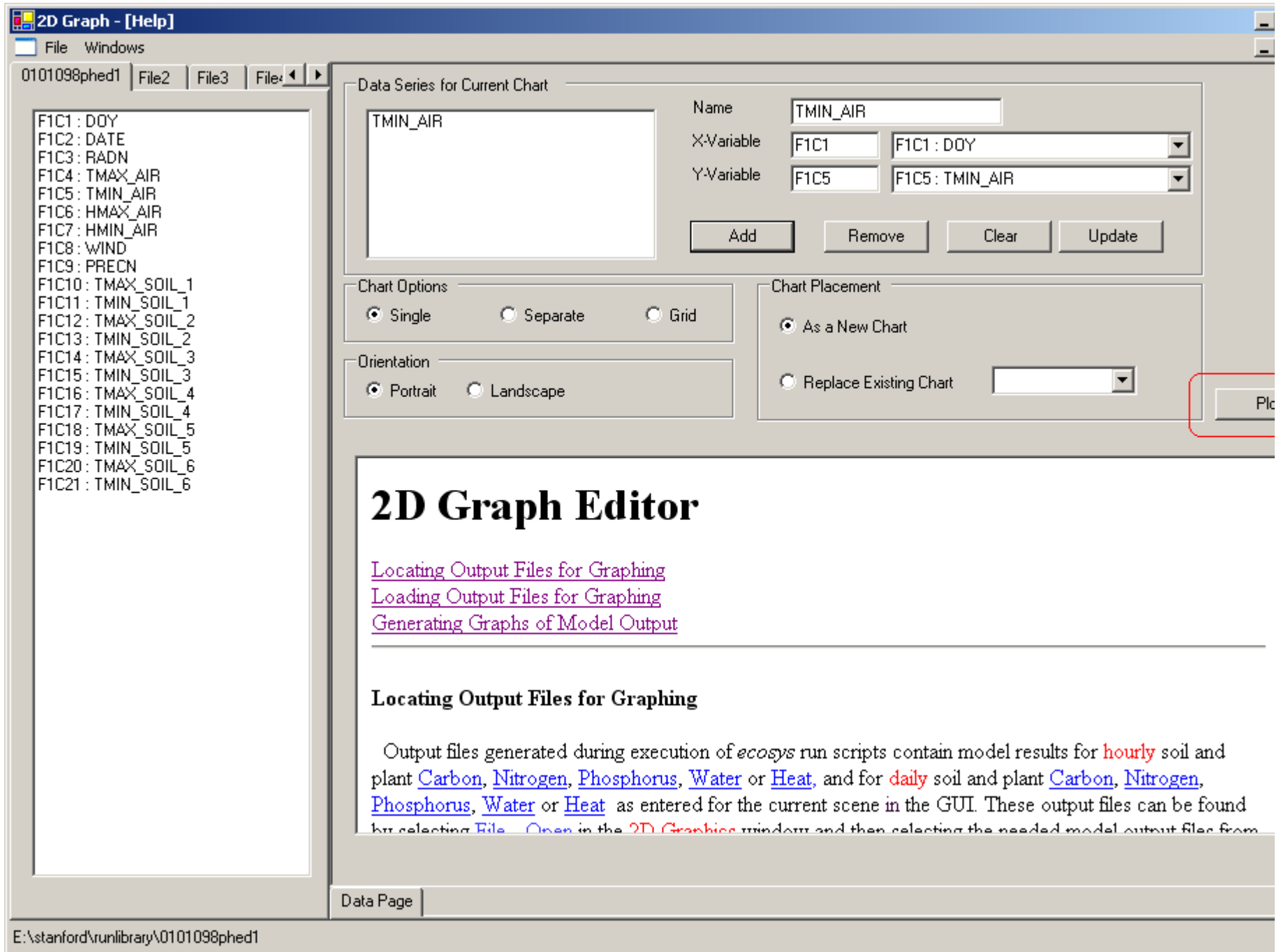
[Locating Output Files for Graphing](#)  
[Loading Output Files for Graphing](#)  
[Generating Graphs of Model Output](#)

---

**Locating Output Files for Graphing**

Output files generated during execution of *ecosys* run scripts contain model results for **hourly** soil and plant [Carbon](#), [Nitrogen](#), [Phosphorus](#), [Water](#) or [Heat](#), and for **daily** soil and plant [Carbon](#), [Nitrogen](#), [Phosphorus](#), [Water](#) or [Heat](#) as entered for the current scene in the GUI. These output files can be found

- After click "Add" button from the above window, you will see a window below:



- Click the "Plot" button from the above window, you will see a window with plot. On the window below, you can click "Data Page" or "Plot n" tab which is highlighted with red circle to switch between data page and plotting window. You can right click the plot to modify, zoom, and save it etc.

