NOTE: The Four models are all included in this single computer code. To run the individual models, set "Flag 1" and Flag 2" in the "Edit Parameters" window under the "Model Options" tab. Flag 1 sets the uptake options for the plant component of the model and Flag 2 sets the uptake options for the microbial component of the model. Values for the flags are as follows:

Uncoupled model:	0
Liebig model:	1
Concurrent limitation model:	2
Acclimating model:	3

Four parameter files have been provided with these flags preset for each of the four models.

INSTRUCTIONS FOR SETTING UP AND RUNNING THE MODEL:

Set up a directory for the model on your hard drive.

Place in that directory the 'model.exe', '*.par', and '*.drr' files. These contain the compiled code, sample parameter files, and a sample driver files, respectively.

To run the model:
Double click the 'model.exe' icon in your file manager. A blue form will appear on your screen. The first thing you must do to run the model is specify the files to be used for the parameters, drivers, and output. If these files exist, there are three ways these specifications can be made (recommended method is in yellow):

First, you can type the path and names of the files in the edit boxes next to the respective labels. To get into the edit boxes, either single click on the appropriate box or type 'Alt-p', 'Alt-d', or 'Alt-u'. This way of doing things is hopelessly old fashion and not recommended. In any case, to use this method you need to know the path and name of the files ahead of time.

Second, you can select files by going through the 'Files' option on the main menu, then select the appropriate type of file, and the 'Choose...' option. This option will bring up a browse window from which you can navigate through your directory structures to find the file you want.

Third, there is a shortcut for the second option described above. If you single click on the file label (i.e., the label in black

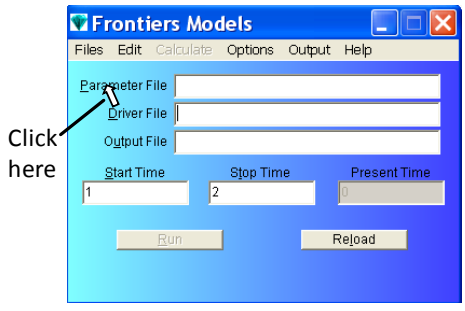


Fig. 1: Shortcut to specifying files.

letters to the left of the three edit boxes; see Fig. 1), you will be placed directly into the browse window.

Do not worry if the parameter, driver, and output files do not exist. You can create them from within the model. In fact this is the recommended way of creating new files; to create these files outside of the model you would have to figure out the formats for each of the files. The easiest way to make new files is to make changes to an existing file FROM WITHIN the model. Sample parameter files are provided. We'll deal with the parameter files later. Let's deal with the other two types of file to get rid of the easy ones first.

The output file is rewritten every time you run the model. The only thing you need to remember is to specify a path and name (the model will not let you run until this specification is made). By clicking on the label to the left of the driver edit box, you will be placed in a browser. Simply navigate to the proper folder and enter the name of the output file. If you do not want to overwrite an earlier run, make sure to give the output file a new name.

The driver file contains all the time-series data needed to run the model. To edit an existing or create a new driver file, select 'Edit' from the main menu and 'Drivers' from the pull-down menu or select 'drivers', 'Edit' from the pull-down menus under the main-menu 'Files'. An edit window will open (Fig. 2; the edit window will behave much like the note-pad editor that came with your windows software). If a driver file was previously specified, its contents will be displayed in the edit window. If you did not previously specify a driver file, two lines of text will be displayed in the window. In the first line will be the name of the variables (i.e., time and the drivers). In the second line will be the units for each of these variables. Below these lines, you will need to fill in data for time and all of the drivers, but you only need to specify these data for 1) the initial values of the drivers and 2) for time steps when the drivers change. For example, if you wanted to simulate an instantaneous change in time step ten, but otherwise maintain everything constant then you only need to enter two lines of data. In the first would be a 1, for time one, followed by the initial values for all the drivers. In the second line would be a 10, for time ten, followed by the new values for ALL of the drivers, even if their individual values have not changed. The model expects there to be a comma between the drivers and for the drivers to be in the same order as they are listed at the top of the file. Otherwise the values can be placed anywhere on the line. You can have the drivers change as many times as you want during a run. You could even specify the drivers for each and every timestep of the simulation.

You can also cause the drivers to ramp form one value to another over a specified time period. To create this ramp, specify the time and drivers for the beginning and end of the ramp as above, but put a minus sign (-) in front of the time at the

end of the ramp. The model will interpolate linearly between these two end points.

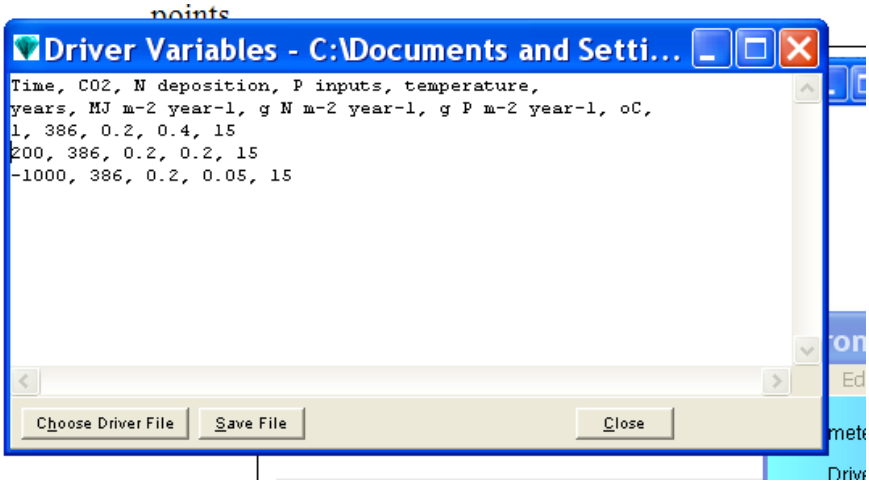
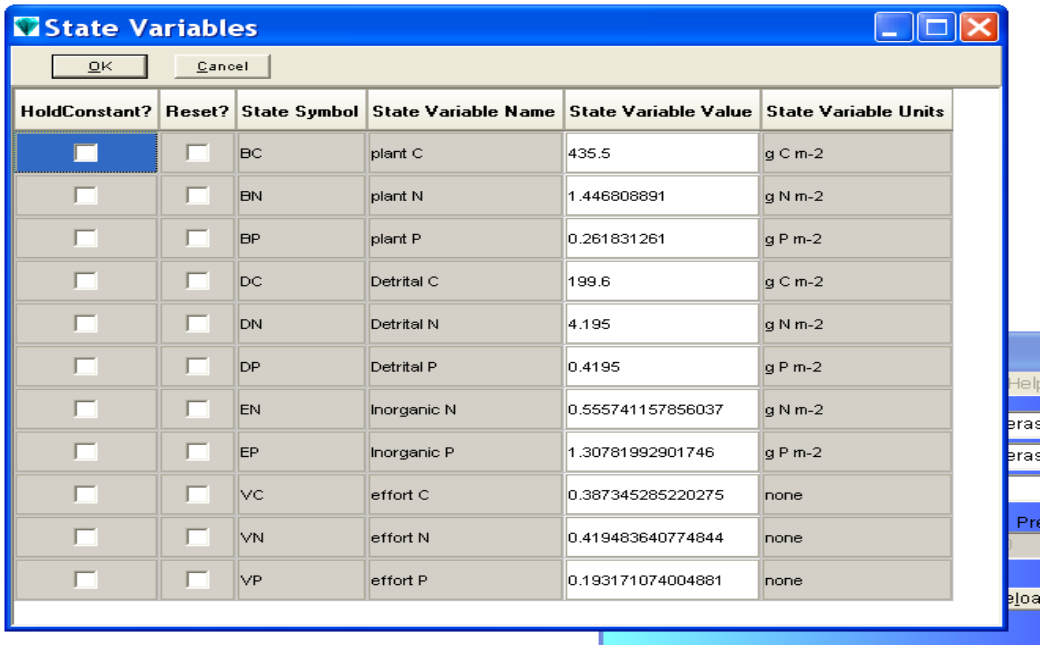


Fig. 2: Edit window for drivers. Values will remain constant between times 1 and 200, jump to new values at time 200, then ramp between the specified values between times 200 and 1000.

The parameter file is also modified through the 'Edit' on the main menu. The parameter file contains both the parameters and the initial values for all the state variables. These two types of data are treated independently. Thus, on the pull-down menu under the main-menu 'Edit' there are choices for both parameters and state



HoldConstant?	Reset?	State Symbol	State Variable Name	State Variable Value	State Variable Units
☐	☐	BC	plant C	435.5	g C m-2
☐	☐	BN	plant N	1.446808891	g N m-2
☐	☐	BP	plant P	0.261831261	g P m-2
☐	☐	DC	Detrital C	199.6	g C m-2
☐	☐	DN	Detrital N	4.195	g N m-2
☐	☐	DP	Detrital P	0.4195	g P m-2
☐	☐	EN	Inorganic N	0.555741157856037	g N m-2
☐	☐	EP	Inorganic P	1.30781992901746	g P m-2
☐	☐	VC	effort C	0.387345285220275	none
☐	☐	VN	effort N	0.419483640774844	none
☐	☐	VP	effort P	0.193171074004881	none

Fig. 3: State variables edit box.

variables. By selecting 'State variables', a form will appear with six columns. In the first two there are check boxes to either hold the values constant or reset them on a periodic basis (this is useful for debugging but will not be covered more here). Next are the symbols for the variables, followed by a column with the written-out name of the variables, then the initial values for these variables, and finally their units. If a parameter file was not previously specified, the initial values column will have all zeros in it. Simply edit that column to indicate the initial values you want to use.

When you are finished, click on the 'OK' button and you will return to the main form. AT THIS POINT YOU HAVE NOT SAVED THE CHANGES TO THE PARAMETER FILE. To save your changes use the 'Save' or 'Save as' options on the pull-down menu under the main-menu 'Files' (a cntr-S will also save the parameter file). Remember that 'Save' or 'cntr-S' will overwrite the specified file, so if you want to retain the old file, use the 'Save as' option (the model will then automatically shift over to the 'Save as' file as its default).

By selecting 'Parameters' from the pull-down menu under the main-menu 'Edit', a second form will appear with tabs for each of the processes for which parameters must be specified. See the accompanying Model Description for parameter definitions.

The parameters are listed under the processes to which they have been associated ('process' is used broadly here, e.g., Active Biomass is not truly a process).

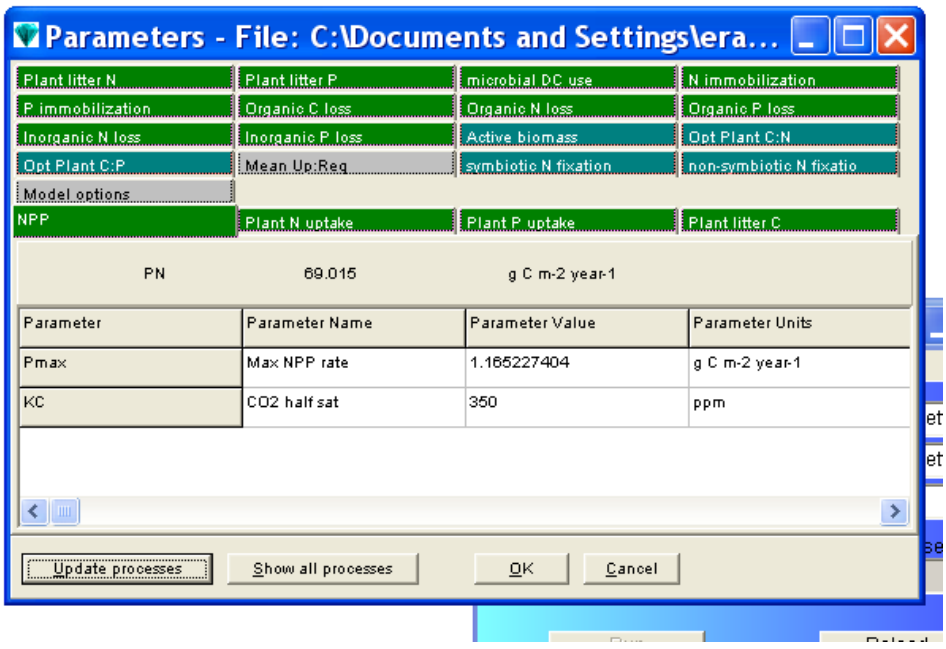


Figure 4: Parameter edit window.

There is a separate tab for each process. On the sheet under each tab is a table with the parameter names, values, and units on it. At the top of the sheet, the

current value of the process is given (usually the initial value). To update the process value after you change parameter values, click on the 'Update process' button at the bottom of the window. Some parameter changes will have a cascading effect on other processes in the model. Also, some processes do not have tabbed sheets associated with them because they have no parameters specifically linked to them. If you want to view all the processes to see what effect the parameter changes have had, click on the 'View all processes' button at the bottom of the window. You can leave the all processes window open while you edit the parameter values. When you click on 'update processes', the processes on the process window will also be updated. When you update the processes or open the view-processes window, an overwrite warning will be displayed; just click OK.

Once you have made all the changes you want to make, click on the 'OK' button. **IF YOU WANT TO MAKE SURE YOUR CHANGES ARE SAVED, SAVE THE PARAMETER FILE NOW.**

You now have all the files specified. The next thing you need to do is specify the duration of the simulation. Click the edit box under the 'Stop Time' label and enter the number of time steps you want the simulation to run. After entering the stop time, the run button becomes active. You can run the model at this point.

OK, now run the model.

You can view output from within the software using the main-menu 'Output'. There are four options in the pull-down menu.

The first option allows you to open a 'Chart' where the output data can be plotted. When the window opens, there will be a blank graph at the top and a list of all the output data at the bottom. Select the variables to plot by clicking on them (they will then be highlighted). As many as ten variables can be plotted at the same time. Once you have selected the variables you want to plot, simply click on the 'Update Chart' button. You can also change parameters and rerun the model from within the chart. Click on the word 'parameter' above the edit boxes to the right of the chart and a list of all the parameters will appear. Select the parameter you want to change, edit its value, then UNCHECK the 'Parameters' check box below the edit boxes and press the 'Run' button. Once the model has stopped running, click 'update chart'. If you want to reset the parameters back to their original values, check the 'Parameters'' check box and rerun the model; all the parameters will be reset to the values saved in the parameter file. If you want to continue a run where the simulation left off, uncheck the 'States' checkbox and run the model.

If you do not like the scale for the plot, you can change it by selecting the 'y-axis', 'scale' or 'x-axis', 'scale' choices on the pull-down menus under the 'Chart' option

at the top of the window. OR.. you can place your pointer at the upper left corner of the portion of the graph you want to expand, hold the left button on your mouse down, and drag the pointer to the lower right corner of the portion of the graph you want to expand. By holding down the right mouse button, you can drag the field of view in any direction.

By default, the independent variable on the plot is time. You can select any desired variable for your independent variable by selecting 'x-axis', 'define' from the pull-down menus under the 'Chart' option.

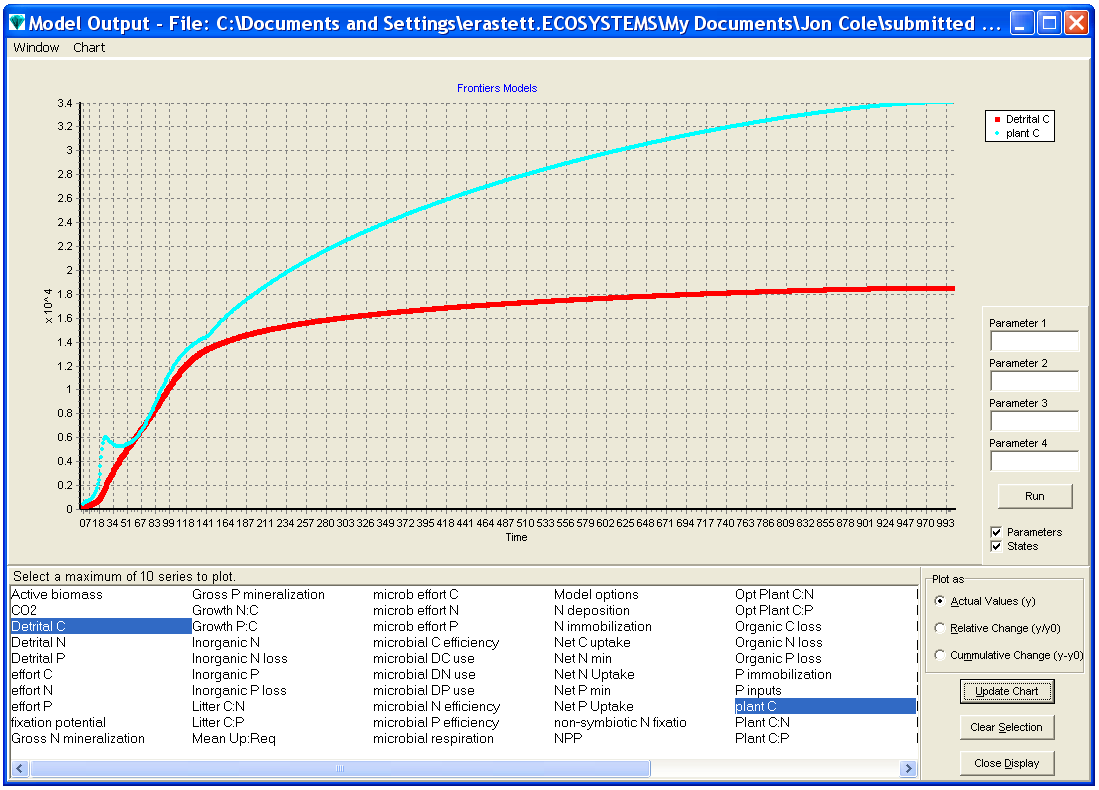


Fig. 5: Chart output window.

You can also print the graph by selecting 'print' form the pull-down menu under the 'Chart' option.

To clear selected variables, click the 'Clear selection' button

To get out of the graph, click the 'Close Display' button.

The second output option allows you to view the output data as a data 'Table'. Selecting this option will open a window with a table containing all the output data.

The third option allows you to upload an existing output file and view it with options one or two above.

The final option (Autochart) is a toggle that will cause the model to automatically go into 'chart' mode at the end of a simulation.

If you want to rerun the model, you can reset the state variables to the values stored on the parameter file by using the 'Reload' button at the lower right of the main window. With this button you can also reload the parameters saved on that file. Make sure to un-check the 'parameters' option if you do not want to overwrite the parameters currently being used by the model.

Have fun!