

# User guide to APEX output analyzer

Cody Zilverberg

29-Dec-2014

The APEX output analyzer is designed to allow APEX model users to quickly evaluate many different output variables and the relationships between variables after running a simulation. It can also be used to compare the results of two different simulations. This tool is still in the early stages of development and may contain errors. We anticipate continuing to improve its functionality and ease of use.

The tool contains several “tabs” listed across the top of the page. The use of each tab is described below.

## Generic output file

This tab is designed to be compatible with multiple APEX/EPIC/SWAT/ALMANAC output files, but to date it has only been extensively tested with a couple APEX output files (\*.SAD, \*.DGZ). Files that follow the format of the \*.SAD file are likely to work. That is to say, the file contains a number of lines at the top that can be ignored, followed by a row of column names, followed by data. Each row should contain data for one day, month, or year. A file such as the \*.OUT file will not work because it does not follow this format. You may notice several error messages displayed when you first load the webpage. These should disappear after you have successfully loaded your output file.

1) Use the “Browse...” button to locate the output file of your choice (e.g., simulation.SAD). You may choose to change the label of the simulation from the default (“Sim1”) but it is not necessary to do so.

2) Ensure all settings are correct for your file. The defaults are set for a \*.SAD file.

- You may need to adjust the number of lines that should be skipped when reading the output file. You can determine how many lines to skip by opening your file in a text editor and counting them.
- Output files typically have a number of initial columns that provide information (such as the date) that you do not want to plot. You can skip these by adjusting the second parameter
- Identify the names of the columns that include the name of the crop, the year, month, day, and subarea. These are CASE SENSITIVE. That means you must use upper- or lower-case letters exactly as they appear in the output file and the list of y-axis variables seen on the left of the screen.
- Once you have done the following correctly, you should see a preview of your data file in table format. You may optionally choose to load a second simulation output file of the same type (both should have the same file extension and same simulation duration).

- 3) After a file has been successfully loaded, the program will tell you the model years included in the simulation output file in a message box in the upper left corner of the screen. Below this box, you have the option of viewing a subset of the model years.
- 4) Select one or more (maximum of five) y-axis variables for plotting
- 5) Select one or more crops for plotting
- 6) Select the x-axis type. Choosing “Month” or “Year” will cause the program to aggregate data, if necessary.
- 7) You may choose to aggregate the data across years and/or crops using checkboxes. Choosing to aggregate years will create a single plot for all years. Unchecking this box will create a plot for each year. If you have many years in your subset, the graphs become so small they are not readable. You may also choose the aggregation function desired. In most cases, the desired function will be the “mean”, but other functions are also available.
- 8) If your simulation contains more than one subarea, you can choose which one to plot.
- 9) Once the data is plotted as you like, you can choose to download the dataset used for plotting, which will contain all of the aggregation options you selected.
- 10) As an option, you can create a custom calculated variable. You can select two of the y-axis variables and a single mathematical operator. Your calculated variable is plotted by choosing “Custom” from the list of y-axis variables.

### **X-Y plot**

This tab is designed to plot two output variables, one on the x-axis and one on the y-axis. It will use the same files and aggregation options selected on the Generic output file tab.

Follow the instructions for loading and aggregating a file using the Generic output file tab. Then, switch to the X-Y plot tab and select the two variables you wish to plot

### **Weather plot**

Use this tab to view an APEX daily weather input file (\*.DLY).

- 1) Use the “Browse” button to locate your \*.DLY file. After a file has been successfully loaded, the program will tell you the model years included in the weather file in a message box in the upper left corner of the screen.

2) Below the message box, you have the option of selecting a subset of the model years. Select the years you wish to view.

3) Select your y-axis variable. Note that selecting “Accumulated precipitation” requires extra computing time and may need several minutes to produce a graph.

4) You may choose to aggregate years, or to place multiple years on a single graph.

### **S-curve**

This tab shows the user what an APEX S-curve looks like, given two x-y pairs. The blue dots represent the input points. You can move the mouse around the screen, and the S-curve values at the location of your mouse will be displayed.

You must input two x-y pairs at the left of the screen. Both should be whole numbers. For example, if your APEX parameters for an S-curve were 15.50 and 80.95, your X1 would be 15, your Y1 would be 50, your X2 would be 80, and your Y2 would be 95.