Last Updated: 16 February 2023

Prepared by: Kevin McGarigal

Tutorial 3: Batch processing multiple grids

In this tutorial, you will batch process multiple input grids. Note, this tutorial assumes that you now have a basic working understanding of FRAGSTATS from completing the first two tutorials and/or reading the detailed user guidelines that come with the FRAGSTATS software.

1. Open FRAGSTATS

First, open FRAGSTATS from the start menu or by double clicking on the FRAGSTATS icon on the desktop.

2. Open a FRAGSTATS model

Next, open a saved FRAGSTATS model. If you saved the model from tutorial #2, you can open that model (if is not already open). If you didn't save the model, open the provided model by clicking on the **Open** button on the toolbar or select **Open** from the File drop-down menu and navigate to the tutorial #3 directory and select the file named **fragmodel.fca**.

3. Create and input a batch file

Next, so that we can start fresh, remove any layers that are already loaded into the model. Specifically, click on **Remove all layers** in the Batch management section of the user interface on the Input layers tab. This will delete all grids previously loaded into the model.

Before you can import a batch file, you need to first create one. A batch file is a comma-delimited ascii file that lists the input grids to be analyzed according to the same model parameterization. Specifically, the batch file contains the full path and name of each input file and a description of the grid attributes associated with each input grid, including the cell size, background value, number of rows and columns, band number, and data type.

GeoTIFFs.--Open the provided **geotiffbatch.fbt** file in a text editor. Note, FRAGSTATS recognizes *.fbt as the extension for batch files, but this is not a required extension.

```
C:\Work\Fragstats\Tutorial\Tutorial_3\reg21b.tif, x, 999, x, x, 1, x, IDF_GeoTIFF
C:\Work\Fragstats\Tutorial\Tutorial_3\reg66b.tif, x, 999, x, x, 1, x, IDF_GeoTIFF
C:\Work\Fragstats\Tutorial\Tutorial_3\reg78b.tif, x, 999, x, x, 1, x, IDF_GeoTIFF
```

This file contains three rows, one for each input grid to analyze, and the following eight fields:

1. *Input grid*: The first field contains the full path and name of the input grid. Note, if we were using ArcGrids we would put the name of the grid folder in place of the grid name since the grid is actually a folder containing several files.
2. *Cell size*: The second field contains an integer value corresponding to the cell size (in meters).
3. *Background value*: The third field contains an integer value corresponding to the designated background value. Note, any class designated as background in the Class descriptors file (see tutorial #2) will be reclassified to this class value and treated as background.
4. *Number of rows*: The fourth field contains an integer value corresponding to the number of rows in the input image.
5. *Number of columns*: The fifth field contains an integer value corresponding to the number of columns in the input image.
6. *Band number*: The sixth field contains an integer value corresponding to the band number to interpret in the input image, which by default is #1 but can vary for some of the input data formats.
7. *Nodata value*: The seventh field contains an integer value corresponding to the nodata value.
8. *Input data format*: The last field contains a character string identifying the input data format, with the following options corresponding to the various input data format types: (e.g., IDF_GeoTIFF, IDF_ASCII, IDF_8BIT, etc.). Note, in this example, since the input grids are all GeoTIFFs, the grid attributes associated with cell size, number of rows and columns, and nodata value are not needed; an "x" is used in place of the argument.

To use the provided batch file, click on the **Import batch** button in the Batch management section of the user interface on the Input layers tab and navigate to the tutorial directory and select the **geotiffbatch.fbt** file.

Ascii grids (and other formats).--If you are working with ascii grids, open the provided **asciibatch.fbt** file in a text editor. Note, a batch file and the corresponding input grids are not provided for other input data formats, but it would be similar to the one shown here except the last argument that identifies the input data format would reflect the input data format (e.g., IDF_8BIT, IDF_GeoTIFF, etc.).

```
C:\Work\Fragstats\Tutorial\Tutorial_3\reg21b.asc, 50, 999, 102, 102, 1, 9999, IDF_ASCII
C:\Work\Fragstats\Tutorial\Tutorial_3\reg66b.asc, 50, 999, 102, 102, 1, 9999, IDF_ASCII
C:\Work\Fragstats\Tutorial\Tutorial_3\reg78b.asc, 50, 999, 102, 102, 1, 9999, IDF_ASCII
```

Note that with ascii files, you must specify all of the grid attributes associated with cell size, number of rows and columns, and nodata value.

To use the provided ascii batch file, click on the **Import batch** button in the Batch management section of the user interface on the Input layers tab and navigate to the tutorial directory and select the **asciibatch.fbt** file.

4. Complete the model parameterization

Next, complete the model parameterization. Running a batch file does not eliminate the necessity of completing the parameterization of FRAGSTATS; it only provides a mechanism for running FRAGSTATS on more than one landscape without having to parameterize and run each landscape separately. Specifically, you still must set analysis parameters and select and parameterize the individual metrics, as described in tutorial #2. If you haven't already completed the model parameterization to your satisfaction, do so now (see tutorial #2 for help if you need it) or simply go with the model as parameterized in the provided model.

5. Run the model and browse the results

Lastly, you are ready to **Run** the model and **Browse** the results, as before (see tutorial #2). The only notable difference between this run and the previous run on a single landscape (tutorial #2) is that the **Run list** in the top-right pane of the user interface now contains is list of outputs associated with this run. Specifically, the Run list contains a separate set of results for each of the input grids in the batch file. Click on each of the list items to view the corresponding patch, class and landscape results. To save the results, click on Save run as... and navigate to the desired directory and enter a basename for the output files. Note, the basename.patch file will contain all of the results for all of the input grids associated with this run. In this case, the file will contain the patch metrics for all of the patches in each of the three input grids. Similarly, the class and landscape files each will contain all of the results appended into a single file.