Tutorial for *EnergyFlux*

Michael Allshouse
Center for Nonlinear Dynamics and Department of Physics, University of Texas at Austin

The Matlab code and GUI for the Green's function based approach for determining the velocity, pressure, and energy flux by Allshouse, Lee, Morrison, and Swinney are available at [JFM LINK HERE]. The latest version is available at the following URLs [MATHWORKS LINK HERE] and [CHAOS LINK HERE]. This guide contains information that is needed to use the GUI and a tutorial to analyze the attached data set. All of the necessary equations and derivations are from the JFM manuscript. To demonstrate the implementation of the code, a data set from our simulated results is included.

1 Input Data Format

The user must first supply the .mat file which contains the density perturbation field and the coordinate grid. There is also the option to include the buoyancy frequency, N, and the time step; however, these parameters can be input manually in the GUI. The names of the various arrays can be user-specified, but the defaults are as follows: Density perturbation - *rho*, horizontal coordinate: x, vertical coordinate, z, buoyancy frequencey - N, and time step - dt.

Density perturbation: The density perturbation is a single array that contains density perturbation data for multiple time steps. The first dimension is the z direction, the second dimension is the x direction, and the third is time. In order to calculate the time derivatives, there must be density perturbation for at least two time instances.

Coordinate arrays: The x and z coordinate arrays must be in the same shape as the density perturbation array minus the time dimension. These arrays are in the form of outputs for the Matlab function "meshgrid."

Optional parameters: The buoyancy frequency and the time step are optional parameters that can be included in the data file. Both values must be scalars.

2 Pre-execution steps

The following steps must be performed before the calculation.

- *Data set selection* - By clicking the "Browse" button for the "Matlab File with Data" you will have the ability to select the .mat file containing the necessary data.
- *Output folder selection* - By default the output data will be saved into the same folder that contains the input data. If you want to change this select the "Browse" button for the "Data Output Path."
- *Set Array Names* - Set the variable names for the density perturbation and coordinates. If you have save the buoyancy frequency and time step in the array file set these variable names as well. If you want to input this manually, simply replace the variable name with the value.
- *Units convention* - Select the SI or CGS units radio button based on the unit convention of your data set.

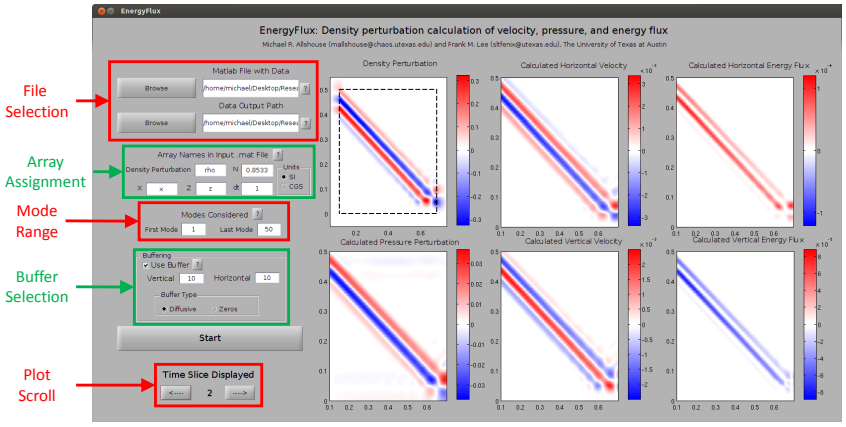

Figure 1: *EnergyFlux* GUI with labels.

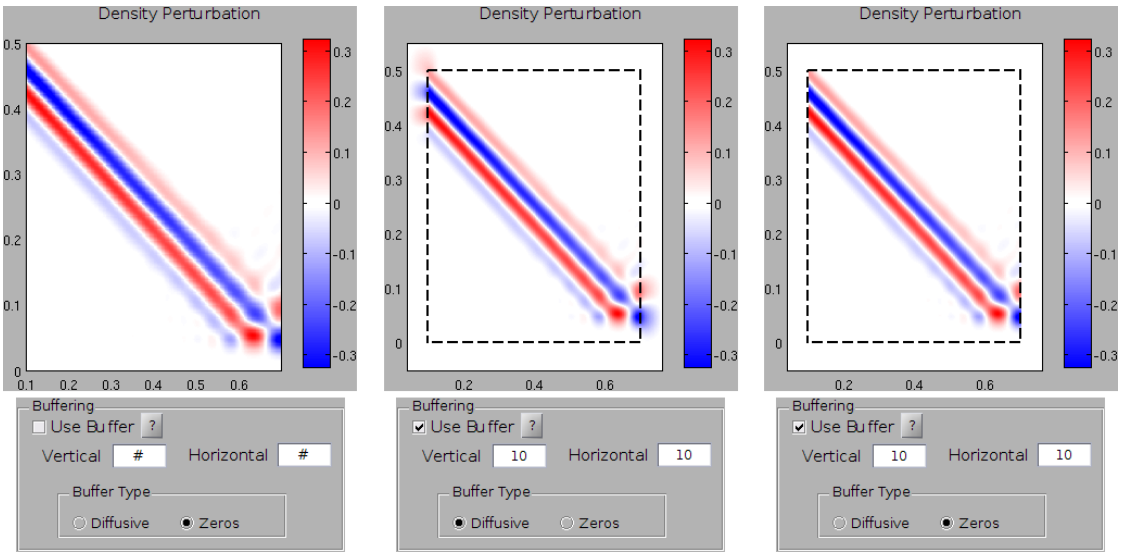

Figure 2: (left) No buffer used. (Center) "Diffusive" and (right) "Zero" buffering options.

3 Optional features

The user is given two optional features that can be used. The first sets the mode number range for the analysis. The second adds a data buffer around the original data set.

3.1 Mode number selection

The calculation performs a Fourier analysis on each horizontal slice of the data set and then for each mode within the calculation range performs the Green's function integration. There are potential situations where not all modes contribute significantly to the output result, and reducing the number of modes considered will reduce the run time accordingly. If you know what the mode range should be, you can replace the "#" with the number. If you are unsure what modes to consider, the code will automatically set them for you. The range automatically selected starts with the first mode and ends at the first mode where the spectral contribution is less than 1% of the maximum.

3.2 Data buffering

As explained in Appendix A in Allshouse et al., buffering the data can reduce erroneous signatures in the pressure field. To include buffering, check the box next to "Use Buffer." If a buffer is used, you will need to set its extent as a percentage of the domain in that direction. For example, a 10% vertical buffer will extend the domain up by 10% and down by 10% of the original height of the domain. There are two types of buffering. The "Diffusive" option creates a buffer with a gradual decay of the density perturbation from the edge of the original data set to zero at the new boundary. The "Zeros" option extends the domain with a density perturbation of zero. When buffering is used, the density perturbation plot will show the extended domain. The result of different "Buffering Selection" and the corresponding density plots are presented in Figure 2.

4 Execution and Results

To execute the calculation after the inputs, modes, and buffering have been set, click the "Start button." There will be a number of progress bars that pop up as the calculation is performed. As each calculation is completed the results will be plotted to the right. The output data will be saved into the designated folder as "greens_function_outputs.mat".

Note: You must be in the directory containing the GUI and the necessary functions or have added that directory to your path in order to run the GUI.

5 Tutorial

Along with the gui and functions, a data set has been provided to the user to give an example of the data format and to execute this tutorial.

5.1 Download and Start Up

Download the zip folder and extract the files. This should contain five .m files (*buffer_data, EnergyFlux, press_calculation, redblue, and vel_calculation*), *EnergyFlux.fig, example_data.mat*, and *EnergyFlux.pdf*. All the files must remain in the folder, and this must be the current directory of Matlab to run the GUI,

To start the GUI, you can run the command `EnergyFlux` in the Command Window. Double clicking *EnergyFlux.fig* will open the GUI but it will NOT allow you to use it.

5.2 Simple analysis

1. With the GUI open, click the top "Browse" button in the "File Selection" section. This will open a browser for you to select the file *example_data.mat*.
2. Select *example_data.mat* and click "Open".
3. To perform the most basic analysis click the "Start" button.
4. After the plots are produced, toggle between different time instances by selecting the right and left arrows in the "Plot Scroll" section.

5.3 Manually set buoyancy frequency and time step

1. Perform the first two steps of the "Simple analysis" if the GUI has been closed.
2. Set the buoyancy frequency by changing the "N" to 0.8533, which is the value from the paper.
3. Set the time step by changing the "dt" to 1.
4. To perform the analysis click the "Start" button.

5.4 Selecting the mode range

1. Perform the first two steps of the "Simple analysis" if the GUI has been closed.
2. Change the "Start Mode" to 2.
3. Change the "Last Mode" to 30.
4. To perform the analysis click the "Start" button.

5.5 Using buffering

1. Perform the first two steps of the "Simple analysis" if the GUI has been closed.
2. Check the box next to "Use Buffer."
3. Set the vertical and horizontal percentages to "10" representing an extension of the domain by 10% in all four directions.
4. Select the "Diffusive" option.
5. To perform the analysis click the "Start" button.