# EEGpal: **Peark or Trace export module**

Version 2.0, 29.12.2024

The module 'Peak or Trace export' has three possible functions. Firstly, it allows to find the highest positive (max) or negative (min) peaks in the specified time interval. The module generates an Excel or txt file with amplitude and position. The user has the possibility to record a check file informing if no peaks (=1) or several peaks (=2) have been detected in the time interval. In the case of multiple peaks, the main output file will by default only record the maximum of the minimum.

The second functionality is to export the amplitude values for each time point (called the timecourse). This allows to perform statistic over the electrodes and times.

The third functionality of this module is to calculate the average of the signal in the specified time interval. This would be useful for extracting values to make plots.

The module saved as output files which can be easily load in your favorite statistical software (like Excel, R, SPSS, …). By default it saved .xlsx files but you can also save .txt files with tab as delimiter.

This module is design to work with epoched files (after completion of the processing with the **Epoching** module). It required to have import files with identical length (number of TF), number of electrodes and sampling rate in the main windows of EEGpal.

### Find peaks

A screenshot of a computer

Description automatically generated

1. Sampling rate of the data. Normally should be automatically transfer from the main windows. You can adjust if it is not the case.
2. Choose the action **Peaks detection.**
3. Specify the time interval. WARNING: The value 0 is the beginning of the file. This module doesn’t take account about possible .mrk file which could define another origin.   
   EoF=End of File.
4. Specify the electrode to study, Specify the indices of the electrodes (1 2 64) and not the name specified by the coordinate file (A1 A2 B32).
5. Specify if you want to detect positive peaks (looking for maxima) or negative peak (looking for minima).
6. In addition, you can detect the peaks for the average of specified electrodes in **4**, on the global field power (GFP) or on the global map dissimilarity (GMD).   
   The result for GFP and GMD is independent of the parameter **5**. Indeed, the values are always positive so the module will always return the maxima.
7. Select if the output file format: .xlsx or .txt (with tab as delimiter).
8. Specify if the peak position should be specified in millisecond (ms) or in time frame (tf) in the output file.
9. Select the output folder where the output will be recorded.
10. You can specify manually a suffix in the output file name. For example: *PeakDetection\_Positive\_98-122ms\_P1.xlsx*
11. Record an additional check. Read the section **How can I use the check file?** in the FAQ bellow**.**
12. Run the module to generate the output. After completion, you can press on Done or Cancel to close the module.

FAQ

**What the output looks like?**

A screenshot of a computer

Description automatically generated

The first columns are the amplitude, and the last columns are the position, either in milliseconds or in a time frame according to parameter **8**.

**How is the peak detection performed?**

The module uses the Matlab function *findpeaks*. It specifically looks for a high value surrounded by smaller values. This detection is not perturbate by the boundary of the interval (which are never detected as peak). This function only detects local maxima. To detect local minima, the signal is inverted. It is why you cannot detect a positive peak (max) and a negative peak (min) at the same time. You must perform two separate runs by changing the option **5**.

**What happens if there are several local peaks?**

The Find Peaks module will always return only one peak in the output file. It will report the peak with the largest amplitude (for positive peaks) or the smallest amplitude (for negative peaks). However, the check file allows you to know if more than one peak has been found in the specified time interval (**3**).

**How can I use the check file?**

The optional check file allows you to know if only one peak is detected in the interval (**value=0**), no peak is detected (**value=1**) or several peaks are detected (**value=2**). You can use this information to look at the signal yourself in Cartool to possibly correct the position according to your own expertise.

A white sheet with black text and numbers

Description automatically generated

Timecourse export

A screenshot of a computer

Description automatically generated

1. Sampling rate of the data. Normally should be automatically transfer from the main windows. You can adjust if it is not the case.
2. Choose the action **Timecourse extraction.**
3. Specify the time interval. WARNING: The value 0 is the beginning of the file. This module don’t take account about possible .mrk file which could define another origin.   
   EoF=End of File.
4. Specify the electrode to study, Specify the indices of the electrodes (1 2 3) and not the name specified by the coordinate file (A1 A2 A3).
5. In addition, you can average of specified electrodes in **4**, the global field power (GFP) or the global map dissimilarity (GMD).
6. Select if the output file format: .xlsx or .txt (with tab as delimiter). This option is available only if you decide to save 2D file in the parameter **7**.
7. You have two choices here:
   1. 2D files: Record the timecourse separately for each input files. It will record a table with a header. Each line is a temporal point (TF) and each column is a channels (first electrodes, then avg, GFP and GMP).
   2. 3D files. Record all the timecourses in a single file. It will record a .mat file without headers. The organization of the file is:
      1. x=time (in TF)
      2. y=channels (first electrodes, then avg, GFP and GMP)
      3. z=input files
8. Select the output folder where the output will be recorded.
9. You can specify manually a suffix in the output file name. For example: *P032\_epoched\_Trigger\_1\_Timecourse\_100-1540tf\_POI2.xlsx for only one file* if you select ‘2D files’ in **7**, or *3D\_Timecourse\_100-1540tf\_POI2.mat* if you selected ‘3D file’ in **7**.
10. Run the module and generate the output. After completion, you can press on Done or Cancel to close the module.

FAQ

**What the output looks like?**

In the case of ‘2D files’:

A table of numbers and letters

Description automatically generated

In case of a ‘3D file’:

A screenshot of a computer

Description automatically generated

**Should I use a ‘2D files’ or ‘3D file’ ?**

It depends on which program you are going to use to open it. The 2D file is a standard matrix which is more easy to work on a software as Excel or SPSS. However, the multiplication of the files could make the life harder to perform statistics. It is why, the option to have a 3D file which contains all the data as been added to this module. It would be suitable for the software R which can import .mat file using the library R.matlab :

A black and white text

Description automatically generated

Average time interval

A screenshot of a computer

Description automatically generated

1. Sampling rate of the data. Normally should be automatically transfer from the main windows. You can adjust if it is not the case.
2. Choose the action **Average time interval.**
3. Specify the time interval. WARNING: The value 0 is the beginning of the file. This module don’t take account about possible .mrk file which could define another origin.   
   EoF=End of File.
4. Specify the electrode to study, Specify the indices of the electrodes (1 2 64) and not the name specified by the coordinate file (A1 A2 B32).
5. In addition, you can average of specified electrodes in **4**, the global field power (GFP) or the global map dissimilarity (GMD).
6. Select if the output file format: .xlsx or .txt (with tab as delimiter).
7. Select the output folder where the output will be recorded.
8. You can specify manually a suffix in the output file name. For example: *AverageTimeInterval\_200-250tf\_POI1.xlsx*
9. Run the module and generate the output. After completion, you can press on Done or Cancel to close the module.

FAQ

**What the output looks like?**

A screenshot of a computer

Description automatically generated