

EEGpal: Peak or Trace export module

Version 2.01, 20.11.2025

Video tutorial: <https://youtu.be/mZfsQAQNwGY>

The **Peak or Trace export** module has three possible functions. Firstly, the module enables the identification of the maximum (max) and minimum (min) positive and negative peaks within the specified time interval. The module generates an Excel or text file containing the amplitude and position.

The second functionality is to export the amplitude values for each time point, known as the timecourse. This enables statistics to be performed on the electrodes and times.

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

The module saves output files that can easily be loaded into your favourite statistical software, such as Excel, R or SPSS. By default, it saves .xlsx files, but you can also save .txt files with a tab as the delimiter.

This module is designed to work with epoched files (i.e. files processed by the **Epoching** module). Imported files must have an identical number of TFs, electrodes and sampling rate in the main EEGpal window.

Find peaks

The screenshot shows the 'EEGpal_PeaksTraceExport' dialog box. It has a title bar and a main area with various settings. The settings are organized into sections: 'Peak Detection or Trace Export' (with a help icon), 'OUTPUT OPTIONS', 'Time interval', 'Elements to process', and 'Warning'. The 'Time interval' section has fields for 'Relative to file start' (ms, tf) and 'Interval onset' (ms, tf). The 'Elements to process' section has a field for 'Electrodes' and a 'Peak' dropdown. The 'OUTPUT OPTIONS' section has a dropdown for 'Output format', a dropdown for 'Peak position need to be saved in ms or TF?', a 'Select uppermost folder where to save the files:' button, a 'Saving folder:' text box, an 'Optional suffix in output file name' text box, a 'Do you want to save also the Check file?' dropdown, and a 'Save another .xlsx informing if the findpeaks script is unsure of the result...' checkbox. The 'Warning' section contains a paragraph of text. At the bottom, there are 'Run', 'Save in memory', and 'Cancel' buttons. Numbered annotations 1 through 12 point to specific elements: 1 points to 'Sampling Rate (in Hz)', 2 points to 'What processing should I do?', 3 points to 'Interval onset', 4 points to 'Electrodes', 5 points to 'Peak', 6 points to 'Average selected electrode(s)', 7 points to 'Output format', 8 points to 'Peak position need to be saved in ms or TF?', 9 points to 'Select uppermost folder where to save the files:', 10 points to 'Optional suffix in output file name', 11 points to 'Do you want to save also the Check file?', and 12 points to the 'Run' button.

1. Sampling rate of the data. This should normally be transferred automatically from the main window. You can adjust this if necessary.
2. Choose the action 'Peaks detection'.
3. Specify the time interval. Warning: the value 0 marks the start of the file. This module does not take into account a possible .mrk file that could define a different origin. EoF=End of File.

4. Specify the electrode to be studied and the indices of the electrodes (1, 2, 64), rather than the names specified by the coordinate file (A1, A2, B32). A '*' means that all the electrodes will be study.
5. Indicate whether you want to detect positive peaks (i.e. looking for maxima) or negative peaks (i.e. looking for minima)
6. In addition, you can detect the peaks on the average of specified electrodes in **4**, on the global field power (GFP) or on the dissimilarity (Dis).
The result for GFP and Dis is independent of the parameter **5**. Indeed, these 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. Indicate whether the peak position should be specified in milliseconds (ms) or time frames (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 file. Read the section **How can I use the check file?** in the FAQ below.
12. There are three validation buttons:
 - a. The **Run** button will carry out the processing parameterized in the module.
 - b. The **Save in memory** button will store all the parameters in memory and close the module without performing the processing.
 - c. The button **Cancel** closes the module without processing and without keeping the entered parameters in memory. The same effect will be achieved by closing the module window.

FAQ

What the output looks like?

Amplitude							Position						
A	B	C	D	E	F	G	H	I	J	K	L	M	N
Files	Amp_e29	Amp_e31	Amp_e32	Amp_Average	Amp_GFP	Amp_GMD		Pos_ms_e29	Pos_ms_e31	Pos_ms_e32	Pos_ms_Average	Pos_ms_GFP	Pos_ms_GMD
P001_epoched_Trigger_5	2.21799964	-0.577037684	-0.124566528	0.165539247		0.117284528		203	233	239	209		223
P001_epoched_Trigger_6	1.438428262			0.125316745	0.742444627	0.101901773		204			212	238	224
P001_epoched_Trigger_7						0.043326292							248
P001_epoched_Trigger_8	0.744847564	0.361666973	0.60324159	0.560033193	0.56766138	0.040514724		221	210	223	220	219	242

The first columns show the amplitude and the last columns show the position in either milliseconds or 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 lower values. This detection is not affected by the boundaries of the interval (which are never identified as peaks). This function only detects local maxima. To detect local minima, the signal is inverted. This is why it is not possible to detect a positive peak (max) and a negative peak (min) at the same time. You must perform two separate runs, changing option **5**.

What happens if there are several local peaks?

The Find Peaks module will always return a single peak in the output file. For positive peaks, it will report the peak with the largest amplitude; for negative peaks, it will report the peak with the smallest amplitude. However, the check file informs you if more than one peak has been found within the specified time interval (point **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.

Files	Check_e29	Check_e31	Check_e32	Check_Average	Check_GFP	Check_GMD
P001_epoched_Trigger_5	0	0	0	0	1	2
P001_epoched_Trigger_6	0	1	1	0	0	0
P001_epoched_Trigger_7	1	1	1	1	1	2
P001_epoched_Trigger_8	0	0	0	0	0	2

Timecourse export

The screenshot shows the 'EEGPal_PeaksTraceExport' window. It has a title bar with standard window controls. The main area is divided into several sections:

- Peak Detection or Trace Export**: A header with a help icon. Below it, a text box explains the module's purpose: 'This module permit to generate different ERP exports which can be used to perform trace analysis in your statistic software (R, SPSS, ...). It allow to export the time course, the average or detect peak value for the specified time interval.'
- Sampling Rate (in Hz)**: A text input field with the value '2048' (labeled 1).
- What processing should I do ?**: A dropdown menu with 'Timecourse extraction' selected (labeled 2).
- Time interval**: A section with two columns: 'Relative to file start' and 'Time'. Under 'Relative to file start', there are 'Interval onset' (48.8) and 'Interval offset' (EoF) fields. Under 'Time', there are 'Interval onset' (100) and 'Interval offset' (1540) fields (labeled 3). A note below states: 'Time must be specified in the Cartool scale (first time fame is tf=0)'.
- Elements to process**: A section with 'Electrodes' (1 2 3) and a text box for 'Specify the channel number separate by space (e.g 1 2 64). Put * to select all channels' (labeled 4).
- Processing options**: Three checkboxes: 'Average selected electrode(s)' (checked), 'GFP' (checked), and 'Dis/GMD' (checked) (labeled 5).
- OUTPUT OPTIONS**: A section on the right with 'Output format' (xlsx) (labeled 6), 'Save in 2D text files (timeXelectrodes) or one 3D mat file ?' (2D files) (labeled 7), 'Select uppermost folder where to save the files:' (Select folder) (labeled 8), 'Saving folder:' (D:\AYBEK_DATA\projet_EEG\Intercepti on_Project_NS_2023\data\temp\dev\test) (labeled 8), and 'Optional suffix in output file name' (POI2) (labeled 9).
- APPLY**: A section at the bottom right with 'Run', 'Save in memory', and 'Cancel' buttons (labeled 10).

1. Sampling rate of the data. This should normally be transferred automatically from the main window. You can adjust this if necessary.
2. Choose the action 'Timecourse extraction'.
3. Specify the time interval. Warning: the value 0 marks the start of the file. This module does not take into account a possible .mrk file that could define a different origin. EoF=End of File.
4. Specify the electrode to be studied and the indices of the electrodes (1, 2, 64), rather than the names specified by the coordinate file (A1, A2, B32). A '*' means that all the electrodes will be study.
5. In addition, you can compute the average of specified electrodes in 4, on the global field power (GFP) or on the dissimilarity (Dis).
6. elect the output file format: .xlsx, .txt (with tabs as delimiters) or .ep (to open in Cartool without electrode names). This option is only available if you choose to save the 2D file in the parameters 7.
7. You have two choices here:
 - a. 2D files: Record the timecourse separately for each input file. This will produce a table with a header. Each line represents a temporal point (tf) and each column represents a channel (first the electrodes, then the average, the GFP and the GMP)
 - b. 3D files. All timecourses will be recorded in a single file. This will create a .mat file without headers. The file is organized as follows:
 - i. x=time (in tf)
 - ii. y=channels (first electrodes, then avg, GFP and GMP)
 - iii. 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. There are three validation buttons:
 - a. The **Run** button will carry out the processing parameterized in the module.

- b. The **Save in memory** button will store all the parameters in memory and close the module without performing the processing.
- c. The button **Cancel** closes the module without processing and without keeping the entered parameters in memory. The same effect will be achieved by closing the module window.


FAQ

What the output looks like?

In the case of '2D files':

	A	B	C	D	E	F	G
1	TF	e1	e2	e3	ElectodesAvg	GFP	GMD
2	100	1.234532833	1.049366713	0.017198961	0.767032836	0.664941613	0.047183277
3	101	1.275371075	1.068965673	-0.045417983	0.766306255	0.672078973	0.046032681
4	102	1.31501174	1.087960005	-0.106108233	0.76562117	0.679115012	0.044789148
5	103	1.35315311	1.106446385	-0.16424723	0.765117422	0.685975159	0.043464413
6	104	1.389520764	1.124536037	-0.219229162	0.764942547	0.692579629	0.042075817
7	105	1.423870206	1.142351747	-0.270471841	0.765250037	0.698845332	0.040646875
8	106	1.455986738	1.160024524	-0.317421049	0.766196738	0.704689646	0.039205279
9	107	1.485692024	1.17768991	-0.359557867	0.767941356	0.710033653	0.037783662
10	108	1.512841582	1.19548285	-0.396405786	0.770639549	0.714805705	0.036419
11	109	1.537326097	1.213536739	-0.427526057	0.774445593	0.718945267	0.035152482
12	110	1.559072852	1.231978536	-0.452533811	0.779505859	0.722406319	0.034029141
13	111	1.57804811	1.250921249	-0.471093923	0.785958479	0.725161344	0.033096255
14	112	1.594250798	1.270467401	-0.482928753	0.793929815	0.727203517	0.032401006
15	113	1.607719779	1.290703297	-0.487816185	0.80353563	0.72855073	0.031987852
16	114	1.618523836	1.311691999	-0.485595614	0.814873407	0.729246342	0.031892775
17	115	1.626769423	1.333476901	-0.476169825	0.8280255	0.729362117	0.032138993
18	116	1.632590771	1.356075048	-0.459504902	0.843053639	0.728997936	0.032732598

In case of a '3D file':



Variables - output

output 1441x6x27 double

```
val(:, :, 1) =
```

1.6168	2.7466	0.6411	1.6682	0.9762	0.0141
1.5920	2.6935	0.6484	1.6446	0.9699	0.0144
1.5681	2.6385	0.6548	1.6205	0.9635	0.0147
1.5451	2.5823	0.6606	1.5960	0.9569	0.0151
1.5233	2.5252	0.6657	1.5714	0.9504	0.0155
1.5027	2.4676	0.6702	1.5468	0.9439	0.0159

Should I use a '2D files' or '3D file' ?

depends on the programme you are going to use to open it. The 2D file is a standard matrix that is easier to work with using software such as Excel or SPSS. However, having multiple files could make performing statistics more difficult. This is why the option to have a 3D file containing all the data has been added to this module. This would be suitable for the R software, which can import .mat files using the R.matlab library:

```
R> library(R.matlab)
R> data <- readMat('file.mat')
```

Average time interval

Peak Detection or Trace Export ?

This module permit to generate different ERP exports which can be used to perform trace analysis in your statistic software (R, SPSS, ...). It allow to export the time course, the average or detect peak value for the specified time interval.

Sampling Rate (in Hz) 1

What processing should I do? 2

Time interval

Relative to file start [ms] [tf]

Interval onset: 3

Interval offset: 3

Time must be specified in the Cartool scale (first time frame is tf=0).

Elements to process

Electrodes 4

Specify the channel number separate by space (e.g 1 2 64). Put * to select all channels

☐ Average selected electrode(s)

☒ GFP 5

☒ Dis/GMD

OUTPUT OPTIONS

Output format 6

Select uppermost folder where to save the files: 7

Saving folder: 7

Optional suffix in output file name 8

APPLY 9

1. Sampling rate of the data. This should normally be transferred automatically from the main window. You can adjust this if necessary.
2. Choose the action 'Average time interval'.
3. Specify the time interval. Warning: the value 0 marks the start of the file. This module does not take into account a possible .mrk file that could define a different origin.
EoF=End of File.
4. Specify the electrode to be studied and the indices of the electrodes (1, 2, 64), rather than the names specified by the coordinate file (A1, A2, B32). A '*' means that all the electrodes will be study.
5. In addition, you can compute the average of specified electrodes in 4, on the global field power (GFP) or on the dissimilarity (Dis).
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. There are three validation buttons:
 - a. The **Run** button will carry out the processing parameterized in the module.
 - b. The **Save in memory** button will store all the parameters in memory and close the module without performing the processing.
 - c. The button **Cancel** closes the module without processing and without keeping the entered parameters in memory. The same effect will be achieved by closing the module window.

FAQ

What the output looks like?

A	B	C
Files	TimeAvg_GFP	TimeAvg_GMD
P001_epoched_Trigger_5	1.055172855	0.048798455
P001_epoched_Trigger_6	1.08188914	0.050110248
P001_epoched_Trigger_7	0.236965597	0.028066206
P001_epoched_Trigger_8	0.506421099	0.028998623