

---

# **ECODATA Apps**

**Justine Missik**

**Apr 30, 2024**



# CONTENTS

<b>1 Overview</b>	<b>1</b>
<b>2 Python package</b>	<b>3</b>
<b>3 Contents</b>	<b>5</b>
3.1 Installing the apps . . . . .	5
3.2 User guide . . . . .	10
3.3 Environmental data . . . . .	21
3.4 Support . . . . .	26
3.5 Developer guide . . . . .	26
<b>4 Indices and tables</b>	<b>29</b>



**OVERVIEW**

ECODATA-Prepare is a set of Python-based apps to access and process remote sensing and other environmental data products and prepare them for further use along with animal movement data. ECODATA-Prepare is designed to be used along with the [ECODATA-Animate](#) tool to create customized movies of animal movement data, and can also be used to prepare input to resource, habitat and step selection models. Development is supported by MathWorks® and the NASA Earth Science Division, Ecological Forecasting Program, as part of [Room to Roam: Y2Y Wildlife Movements](#) project.

Within ECODATA-Prepare,

- The *[Tracks Explorer App](#)* visualizes movement track location points and point densities with several static maps, allows selecting a spatial frame around the track points that is relevant for analysis, and prepares a .geojson file that can be used in the NASA EARTHDATA AppEEARS interface to extract many different remote sensing data products within this research area. This app also provides the longitude-latitude coordinates of the frame around the research area, which can be used to request data from ECMWF or many other environmental data sources, which can be converted to NetCDF format if needed for further steps described below.
- The *[Gridded Data Explorer App](#)* allows you to interpolate and subset the temporal and spatial resolutions of environmental data in the form of a temporal stack (or single static map) NetCDF file. It can also read in several other data formats (which could be resaved as NetCDF). You can also read GIS polygons as shapefiles and mask the environmental data outside or inside the polygons. It subsequently calculates data summaries by period and polygon.
- The *[Subsetter App](#)* allows you to clip relevant features out of large GIS files.
- The *[Movie Maker App](#)* produces an animation file out of a stack of static maps which were produced by the ECODATA-Animate program.

Read our [overview](#) of how to use Movebank, MoveApps, ECODATA-Animate and ECODATA-Prepare to discover and process tracking data, remote sensing data, shapefiles and other layers to create custom visualizations and input for ecological analysis. To try the software before working with your own data, see our [example](#) based on publicly-available data sources.

For help, [submit a GitHub issue](#) or contact [support@movebank.org](mailto:support@movebank.org).

1. Evaluate tracking data with **Movebank** and **MoveApps** and choose data to animate.



2. Make a .geojson file for this area with the **ECODATA-Prepare Tracks Explorer**.



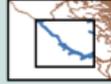
3. Use this .geojson file to request environmental data from **NASA AppEEARS**.



4. Process the environmental data with the **ECODATA-Prepare Gridded Data Explorer**.



5. Extract relevant features from large vector layers with the **ECODATA-Prepare Subsetter**.



6. Create a labels file for features or events to include.



7. Use **ECODATA-Animate** to combine layers, choose viewing options, and create images.



8. Create an animation from the images with the **ECODATA-Prepare Movie Maker App**.



9. Use and share these files and settings for more movies, exploration and analysis.



## PYTHON PACKAGE

The functions underlying the ECODATA-Prepare apps can also be used directly as a python package (i.e. without the GUI interface). The package documentation is [here](#).



## CONTENTS

### 3.1 Installing the apps

#### 3.1.1 Installers

Download installers for the latest release:

OS	Architecture	Download
Windows	x86_64	<a href="#">ecodata-0.2.7-Windows-x86_64</a>
macOS 14	arm64 (Apple Silicon)	<a href="#">ecodata-0.2.7-MacOSX-arm64.pkg</a>
(*) macOS 13	x86_64	<a href="#">ecodata-0.2.7-MacOSX-x86_64.pkg</a>
Linux	x86_64	<a href="#">ecodata-0.2.7-Linux-x86_64.sh</a>

(\*) Use this installer for all Macs that are **not** an Apple Silicon (M-series) Mac running macOS 14 (i.e., use for all x86 (Intel chip) Macs, or any Mac running macOS 13).

Release notes for the latest release can be found [here](#), along with all available installers. Command line (.sh) installers for Mac are also available. You can see all releases (including previous versions and pre-releases) [here](#).

1. Download the correct installer for your operating system.
2. Run the installer and follow the prompts. You can change the default installation location if you wish. This installation location is where the program files will be installed (i.e., files not intended to be accessed by the user after installation). If you have a previous version already installed in the default location, you will need to remove the previous version before installing the new version, or install the new version in a different location. If opening the .pkg in Mac, you might get a message that it “cannot be opened because it is from an unidentified developer”. If this happens, you can right-click or press **Control** click on the application, and then select **Open**. You can then select **Open** to run the installer.
3. A file used to launch the apps (ecodata.command for Mac and ecodata.bat for Windows) will be copied to your Downloads folder. You can move this file to a convenient location after the installer is finished running. **Note:** If you already have an ecodata.command or ecodata.bat file in your Downloads folder from a previous installation, remove it before installing the new version.

After the installation is complete, you can *get started*.

### 3.1.2 Alternative installation methods

The apps can also be installed from the command line. If you have issues with the first installation method (e.g. your computer's security settings are preventing the installer from running), you can use this method as a backup.

#### Install using conda (recommended)

ECODATA is available on [conda-forge](#) and can be installed directly using conda. This method requires installing conda.

#### Install ECODATA using conda

1. Download and install Miniconda. See [instructions for installing Miniconda and download installers here](#). If you already have conda installed, you can skip this step.
2. Open a terminal.

**Mac/Linux:** Run the commands from the built-in “Terminal” application, or any other terminal app.

**Windows:** Run the commands from the Anaconda Powershell Prompt (this was installed by Miniconda). You can find this by searching for “Anaconda Powershell Prompt” in the Start menu.

You can check that your conda installation was successful by running `conda list`. If conda has been installed correctly, a list of installed packages appears.

3. Install ECODATA in a new conda environment:

```
conda create -n eco ecodata -c conda-forge
```

This will install ECODATA in a new conda environment named “eco”. This includes downloading all of the packages for the application, so it may take some time to finish, depending on your internet speed. Once the environment is successfully created, the terminal output should look something like:

```
#  
# To activate this environment, use  
#  
#   $ conda activate eco  
#  
# To deactivate an active environment, use  
#  
#   $ conda deactivate
```

If you get the message “An unexpected error has occurred”, try running the command again. Your internet connection might have dropped briefly and caused the downloads to fail.

---

**Important: For Windows:** Don't click anywhere in the command prompt while commands are running! The Windows command prompt has a “feature” called QuickEdit Mode that is enabled by default and causes programs to pause when clicking inside the command window. It will not give any indication that a program is paused, so you might think the program is stuck when it has been paused accidentally.

---

## Launching ECODATA from the command line

Launch the ECODATA-Prepare apps using the command below. For **Mac/Linux**, you will run the command below from the built-in Mac “Terminal” application, or any other terminal app. For **Windows**, you will run the command using the Anaconda Powershell Prompt. If this is not already open, find it by searching for “Anaconda Powershell” in the start menu.

First, you need to activate the conda environment where ECODATA is installed:

```
conda activate eco
```

Next, launch the apps:

```
python -m ecodata.app
```

A window will open on your default web browser, showing the main app gallery page (the apps are running locally at `localhost:5006`). There may be a short wait the first time you launch the apps, or the first time you launch after an update.

You may receive a message “Do you want the application “python3.11” to accept incoming network connections?” You can click Allow.

Keep the terminal application you used to launch the program open while running the app. To shut down the apps, close the terminal that you used for launching.

After the installation is complete, you can *get started*.

### 3.1.3 Updating ECODATA-Prepare from the terminal

You can check which version of ecodata you have installed by running:

```
conda activate eco
conda list ecodata
```

You can see new releases on the [releases page](#)

If you want to install a new release, you can update your version of the apps by running the command below:

```
conda activate eco
conda update ecodata
```

After the installation is complete, you can *get started*.

### Install using Anaconda (legacy method, not recommended)

See below for steps to install and run the ECODATA-Prepare apps. For **Mac/Linux**, you will run the commands below from the built-in Mac “Terminal” application, or any other terminal app. For **Windows**, you will install Anaconda and then run the commands below using the Anaconda Powershell Prompt. You can launch this by searching for “Anaconda Powershell” in the start menu, or by launching Anaconda Navigator and looking for “Powershell Prompt” in the home screen.

---

**Note:** The apps are still under development, and installation will be simplified soon!

---

### Installing Anaconda

1. Anaconda Distribution is an free, open-source distribution platform for managing and deploying Python and R packages. Install Anaconda if you do not have it on your computer. [Download the installer here](#) and then run the file to install the program.
2. **For Windows only:** To address compatibility issues, downgrade conda to the previous version (v4.14). In the Anaconda Powershell prompt, run:

```
conda install conda=4.14
```

### Preparing the Python environment and installing ECODATA-Prepare

3. *Download the conda environment file here.* This file specifies the Python package requirements used by the apps, and can be deleted following installation.
4. Open a terminal (for Windows, use Anaconda Powershell).
5. Navigate to the directory where you saved the environment file. Run `cd <path_to_directory>` (note that you need to replace the path with the actual path to the directory where you saved the file), for example:

```
cd Downloads
```

6. You can confirm you navigated to the correct directory by running “ls” to list the files in that directory:

```
ls
```

You should see the environment file in the output, for example:

```
Directory: C:\Users\jmissik\Downloads

Mode                LastWriteTime         Length Name
----                -
-a----             11/30/2022  12:38 PM        23470 environment-clean-build.yml
```

If your directory has many files, replace “ls” with “ls -lt” to sort files by the time they were last modified.

7. Install the environment (note that if your environment file is not called “environment-clean-build.yml”, you need to replace this name with the actual name of your file):

```
conda env create --file environment-clean-build.yml --name eco
```

This will download all of the packages for the application, so it may take some time to finish, depending on your internet speed. Once the environment is successfully created, the terminal output should look something like:

```
done
#
# To activate this environment, use
#
#   $ conda activate eco
#
# To deactivate an active environment, use
#
#   $ conda deactivate
```

If there were issues with the installation, you can try using the locked environment file for your operating system instead: *Windows | Intel Mac | M1 Mac*

- After the Python environment is completed, you need to install the ECODATA-Prepare application itself. From the terminal, run:

```
conda activate eco
pip install git+https://github.com/jemissik/ecodata.git@main
```

After the installation is complete, you can [get started](#).

### Launch using Anaconda

If you have installed [Anadonda](#) (Anaconda PowerShell for Windows, Anaconda Navigator for Mac), you can also launch ECODATA-Prepare through this program.

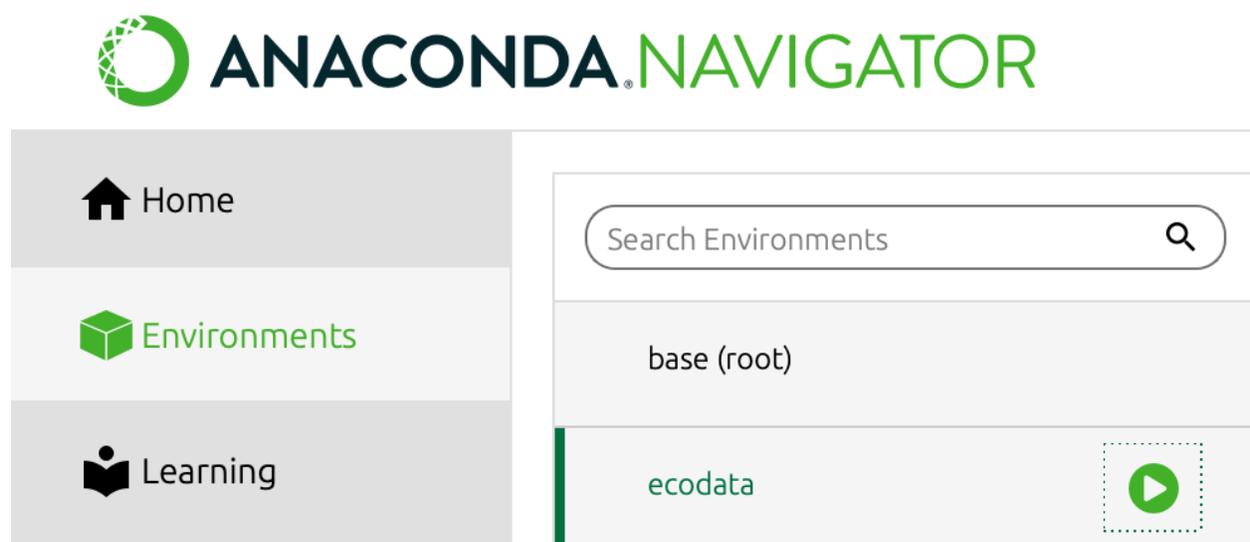
To launch in Windows,

- Open Anaconda PowerShell.
- Copy-paste the below code into the prompt window, and press Enter:

```
conda activate eco
python -m ecodata.app
```

To launch on Mac,

- Open Anaconda Navigator.
- Select Environments from the upper left and navigate to “ecodata”.



- Hit the play button and select “Open Terminal”.
- A Terminal window will open. Enter the following and hit Return:

```
python -m ecodata.app
```

## 3.2 User guide

---

**Tip:** Want a downloadable version of the docs? This documentation is also [available for download](#) as a PDF, Epub, or zipped HTML.

---

### 3.2.1 Contents

#### Getting started

#### Overview

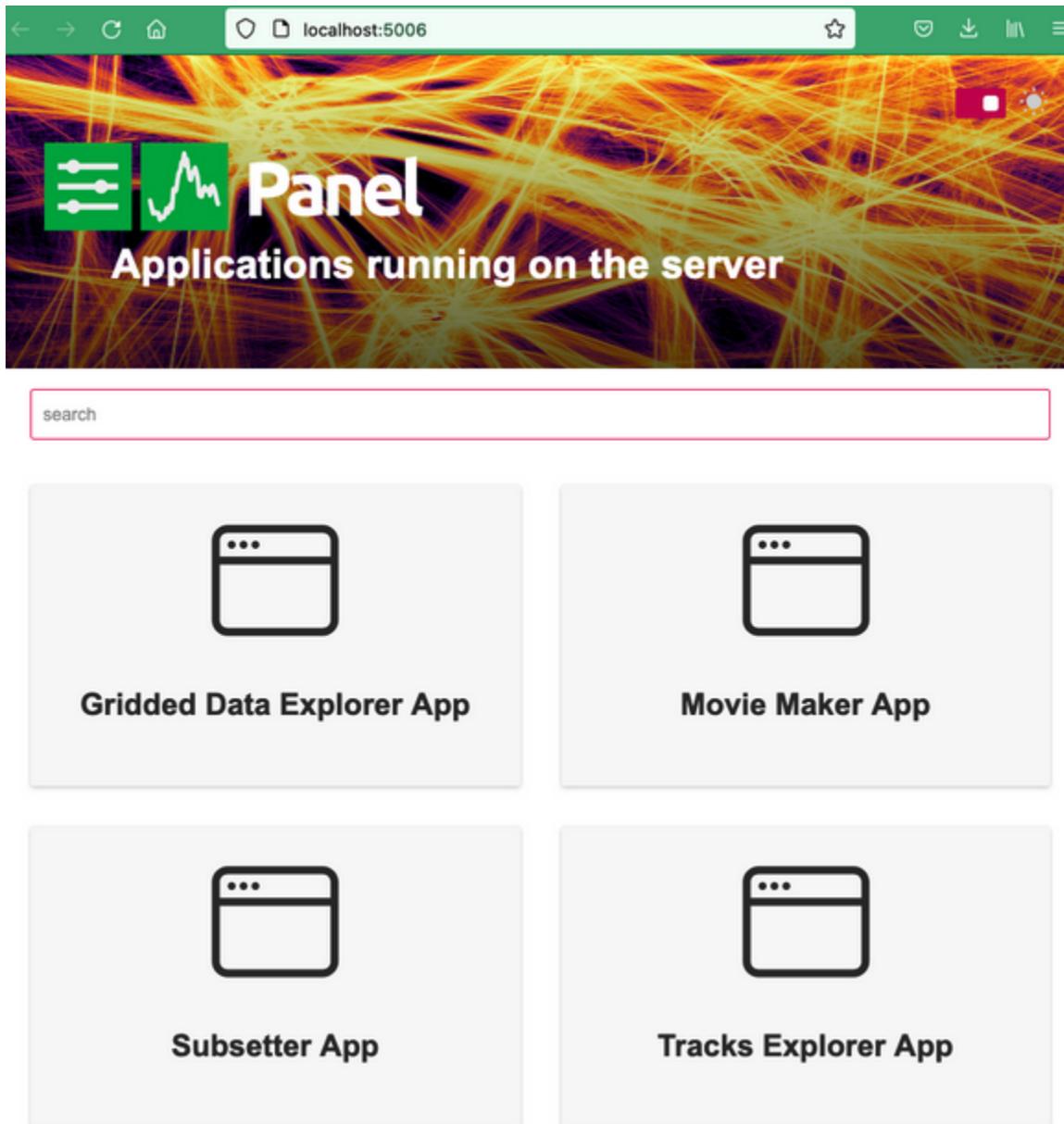
ECODATA-Prepare is a set of four Python-based apps to read, process, and create animations from animal tracking data and gridded environmental data. [Read here](#) for an overview and to learn how to install, contribute and get support. Before using this program, gather files you want to use with the app. This could include:

- a .csv file of tracking data, in Movebank's .csv format
- NetCDF files of environmental data (you can [order these](#) from NASA or ECMWF)
- shapefiles
- a folder of sequentially-numbered .png files that are frames for an animation (you can create these with [ECODATA-Animate](#))

If needed, see the [installation instructions](#) to install or update the program.

See [below](#) for instructions to open the program. When it opens, it will display the main panel, showing the four apps. Click on an app to launch it. From there, you can navigate between apps within the interface, or switch between them by pasting these URLs in your browser window:

- Main app gallery: <http://localhost:5006>
- Tracks Explorer App: [http://localhost:5006/tracks\\_explorer\\_app](http://localhost:5006/tracks_explorer_app)
- Gridded Data Explorer App: [http://localhost:5006/gridded\\_data\\_explorer\\_app](http://localhost:5006/gridded_data_explorer_app)
- Subsetter App: [http://localhost:5006/subsetter\\_app](http://localhost:5006/subsetter_app)
- Movie Maker App: [http://localhost:5006/movie\\_maker\\_app](http://localhost:5006/movie_maker_app)



### Opening ECODATA-Prepare

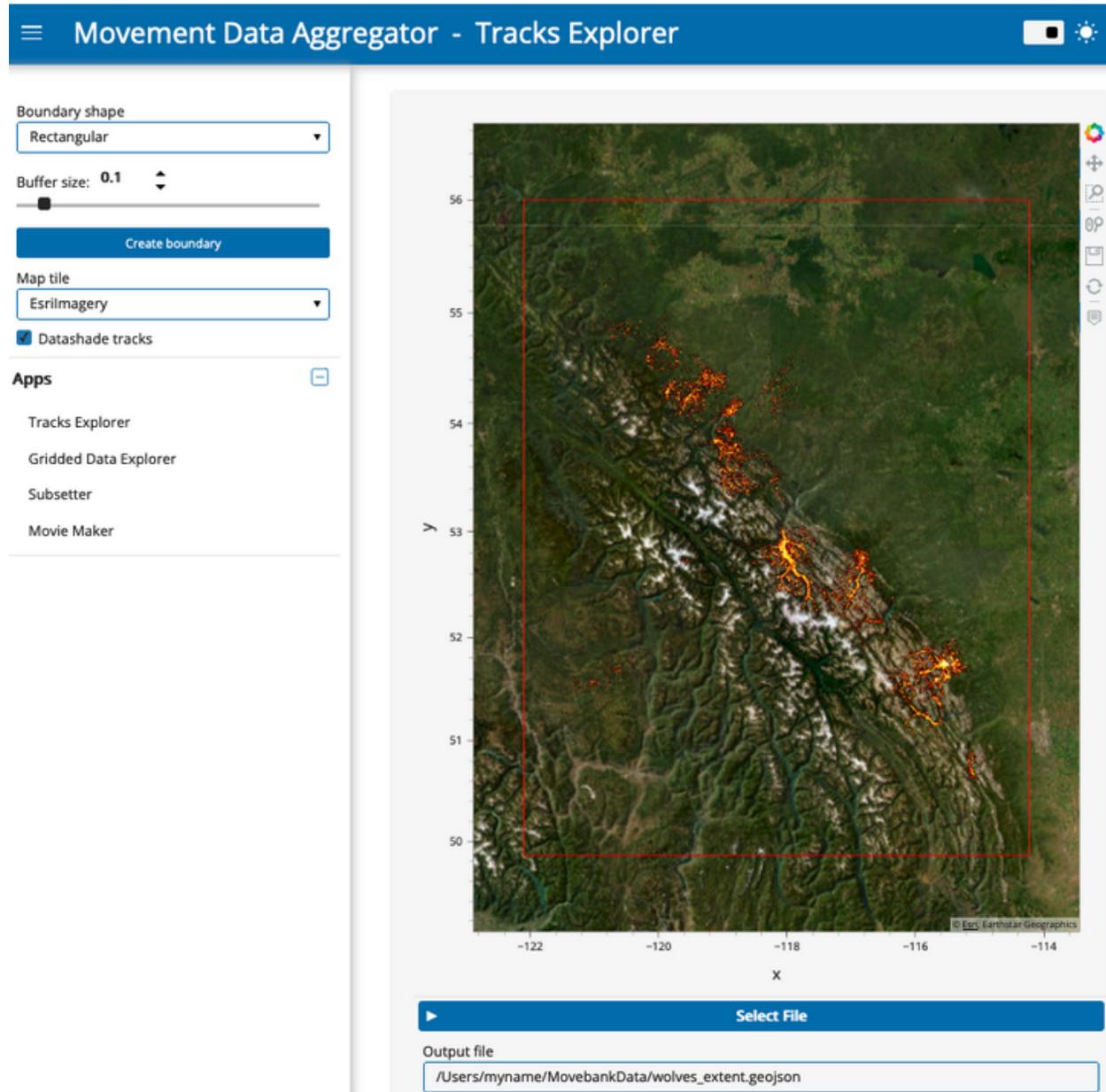
**Note:** These instructions assume that you installed ECODATA using the *Installers*. If you installed ECODATA using one of the *alternative methods*, you will need to launch the apps using the instructions there.

Double-click on the launch file (`ecodata.command` for Mac, `ecodata.bat` for Windows). A terminal window will open, indicating that the apps are launching. When the apps finish launching, a window will open on your default web browser, showing the app gallery. The apps are running locally at `localhost:5006`. *Note: There may be a short wait (10+ seconds) before the browser window opens, particularly if you are launching the apps for the first time.*

You may receive a message “Do you want the application “python3.11” to accept incoming network connections?” You can click Allow.

Keep the terminal window open while running the app. To shut down the apps, close the terminal that opened when the apps launched.

### Tracks Explorer

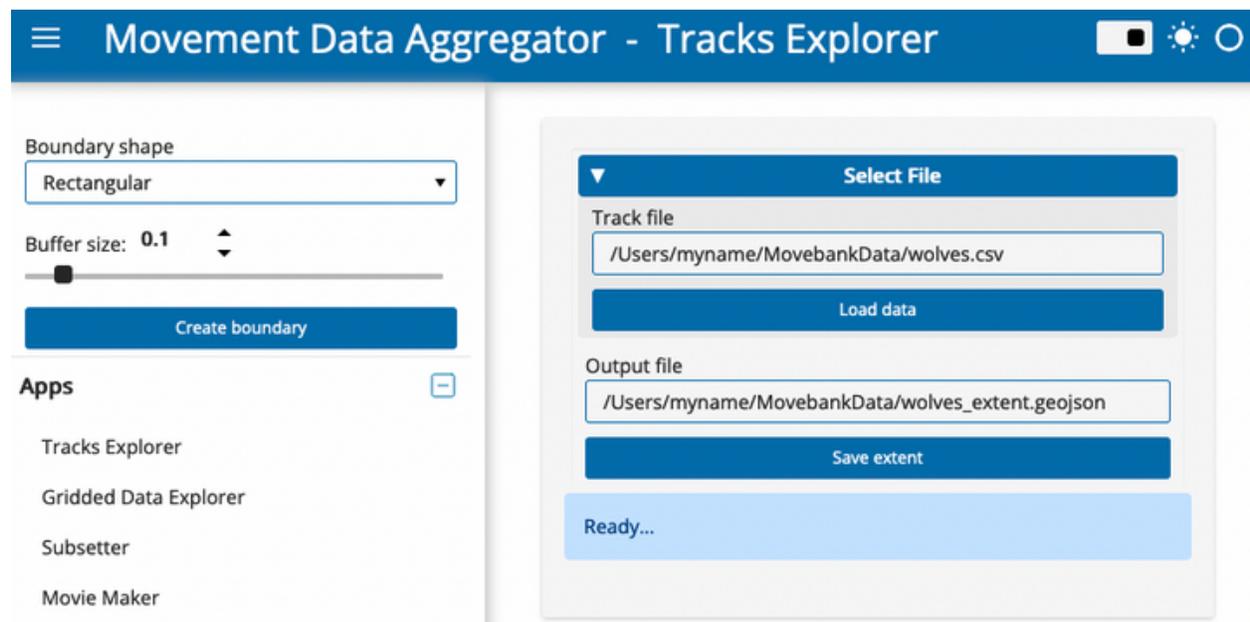


## App features

With the Tracks Explorer App, you can

- Visualize point locations from your Movebank data as individual points or aggregated point density. You can select from a variety of map tiles to use as a background.
- Automatically generate a spatial frame around the track points and save this as a .geojson file that can be used in the NASA AppEEARS interface to extract remote sensing products for the research area.

## Using the app

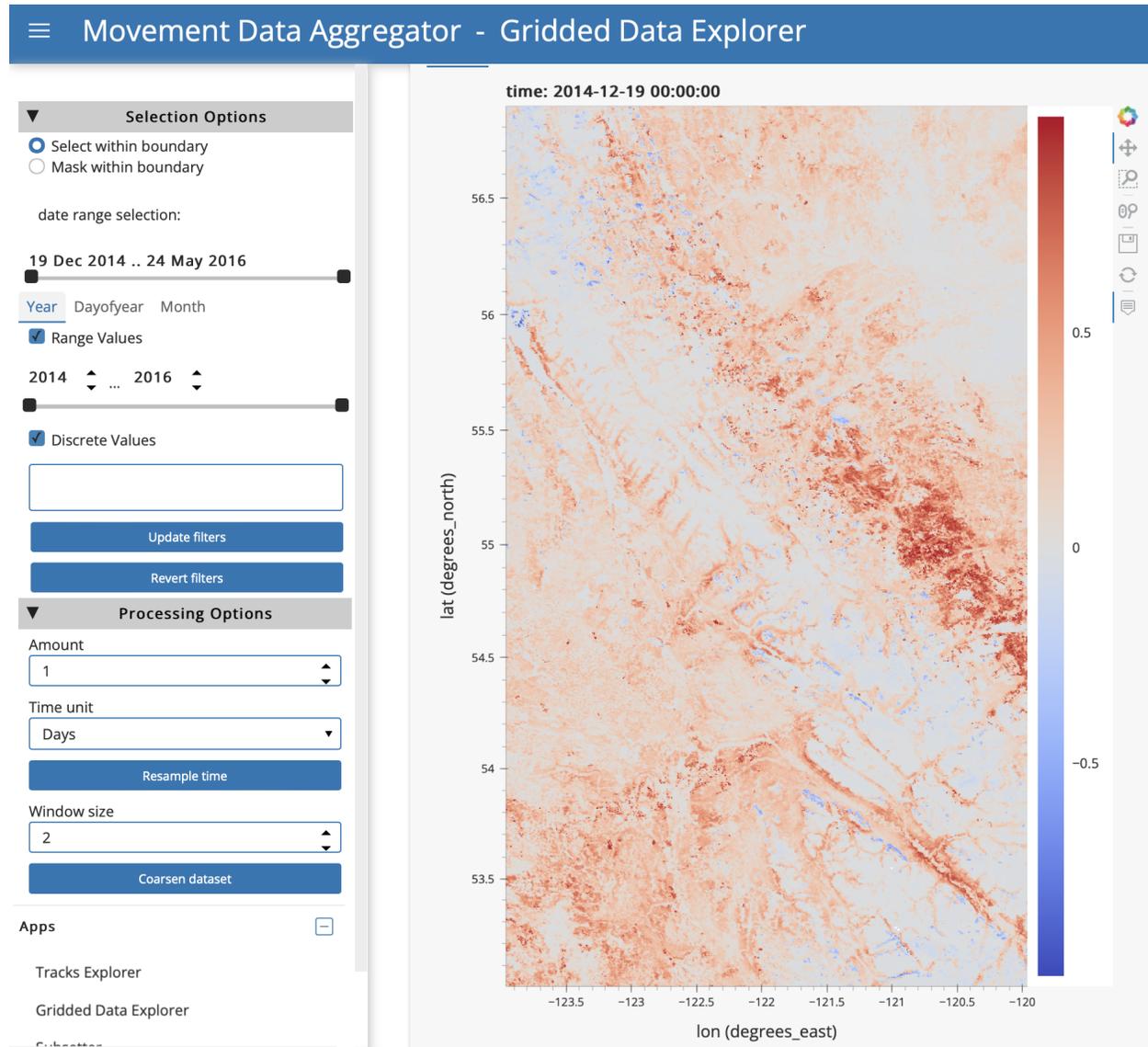


*Data from Hebblewhite and Merrill, study Hebblewhite Alberta-BC Wolves*

1. If you haven't already, download a local copy of the tracking data you want to use, in Movebank's .csv format. It is possible to subset or combine data from different studies.
2. *Launch the Tracks Explorer App.*
3. Under "Select File", paste the full filepath to a .csv file of tracking data, including the file name. In the current version, this .csv file needs to use Movebank's format. An example filepath is "/Users/myname/MovebankData/wolves.csv".
4. Click "Load data". An interactive map will appear of your dataset, where you can pan/zoom to explore the dataset. If a map does not load, view the status message. The message "Error.. Check options and try again" likely indicates that your filepath is incorrect.
5. You can change the background map tile using the drop-down menu in the sidebar.
6. Toggle the "Datashade tracks" checkbox in the sidebar to turn on/off the point density aggregation feature. When unchecked, individual semi-transparent points will be displayed. Note that for very large datasets, plotting will be slow if the "Datashade tracks" option is turned off.
7. A boundary for the tracks dataset is shown on the map in red. The default option uses a rectangular boundary with a buffer size of 0.1 (scale relative to the extent of the tracks). You can adjust the buffer size or change the boundary shape to a convex hull using the widgets in the sidebar.

- To save the boundary to a .geojson file, scroll down and if needed, edit the file name and location under “Output file”. Then simply click the “Save extent” button.

### Gridded Data Explorer



### App features

With the Gridded Data Explorer App, you can

- Read, interpolate and subset the temporal and spatial resolutions of environmental data in the form of a temporal stack (or single static map) NetCDF file.
- Read GIS polygons and mask the environmental data outside or inside of the polygons.
- Calculate data summaries by time period and spatial area.
- Save processed environmental data and summary statistics for further use.

## Using the app

**Tip:** Throughout the App, scroll down to view status messages indicating application progress and next steps.

1. Under “Input environmental dataset”, paste the full filepath to a NetCDF (.nc) file received from NASA AppEEARS (*see instructions*) or any other source. The filepath should include the file name. An example filepath is “/Users/myname/RS\_Data/NDVI.nc”.
2. Click “Load data”. If it is able to read the file, the app will load available variables from the file into drop-down lists for Latitude, Longitude, Time and Variable of interest, and will attempt to select the correct variables automatically. If nothing happens, you might need to correct the filepath.
3. If the app does not assign the correct variable names from the file, you can specify them manually by clicking on the field and selecting from the dropdown list.
4. In the “Variable of interest” dropdown, select the variable you would like to explore.

▼ **Input environmental dataset file**

File path

/Users/myname/RS\_Data/NDVI.nc

Latitude

lat ▼

Longitude

lon ▼

Time

time ▼

Variable of interest

\_500\_m\_16\_days\_NDVI ▼

Load data

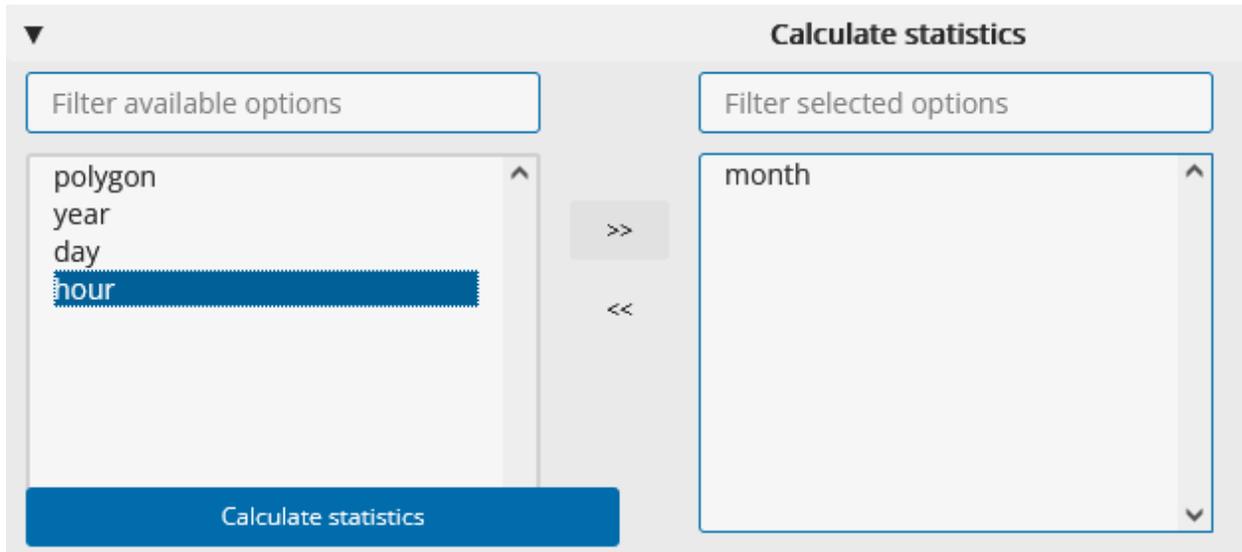
Update variable names

5. Once the input parameters are set, click “Update variable names”. Plots of the data will appear.
6. At the top of the area with the plots, there are two tabs (“Charts” and “Data”). If you click the “Data” tab, you can see more information about the file contents, including the dimensions of the dataset and the variables in the file. Click on the “Show/Hide attributes” icon to the right of a variable to view variable metadata including long names, units, and legend classes.

The screenshot shows the 'Data' tab of the application. At the top, it displays 'xarray.Dataset'. Below this, there are sections for 'Dimensions', 'Coordinates', and 'Data variables'. The 'Dimensions' section shows '(time: 34, lat: 685, lon: 1589)'. The 'Coordinates' section lists 'time' (datetime64[ns], 2022-03-30 ... 2022-12-19), 'lat' (float64, 69.79 69.79 69.79 ... 66.95 66.94), and 'lon' (float64, -141.7 -141.6 ... -135.0 -135.0). The 'Data variables' section lists 'crs' (int8, -127), '\_500\_m\_16\_day...' (float32, nan nan nan nan ... nan nan nan nan), and several other attributes like 'grid\_mapping', 'valid\_min', 'valid\_max', 'long\_name', 'units', 'scale\_factor\_err', 'add\_offset\_err', 'calibrated\_nt', 'origname', and 'fullnamepath'. A 'Show/Hide attributes' button is visible on the right side of the 'Data variables' section.

7. In the “Charts” tab, there are a variety of options to explore the selected variable in space and time.
8. **Filter by area:** Under “Input polygon file”, you can input a polygon file to select or mask data inside polygon boundaries. Selecting will keep data *inside* polygons, and masking will keep data *outside* polygons. Paste the full filepath to a polygon file (e.g., .shp), including the file name, and click “Load file”. When the file loads, it will be plotted on the Chart view above. Select an option and then click “Update filters”. To remove the filter, click “Revert filters”.
9. **Filter by time:** To filter the data for specific times, use the options under “date range selection” under Selection Options on the sidebar. There are time filtering options in the sidebar if you want to filter the data for a specific time range or time conditions (e.g., daytime only, specific seasons or months). You can use the time slider bars to filter by a range of values or enter discrete values into the box. For example, to filter by month you would click the “Month” tab above the “Range Values” option and either use the sliding bar to restrict your data to months of interest or enter numeric months into the Discrete Values box (note that when you click in this box, a dropdown appears with the month values from your Appearances dataset). Available ranges and categories for discrete filtering will depend on the content of your dataset. Select your options and then click “Update filters”. If you want to remove the filters you have applied, click “Revert filters”.
10. **Resample by time:** It is possible to resample the data by time. The app will estimate attribute values from the original values at the requested time schedule by interpolating when upscaling, or using means for downsampling. Upscaling could be used to show smooth transitions in an animation with daily timestamps of a product with a 16-day resolution; downsampling could be used to reduce storage and processing needs for a large hourly dataset if you need only daily data. Under “Processing Options” in the sidebar, select the desired time frequency and click “Resample time”. Once processing is complete, the status at the bottom of the page will show “Dataset Resampled”.
11. **Coarsen spatial resolution:** It is possible to coarsen (downsample) the spatial resolution of the data using block aggregation, for example to reduce file sizes or processing time for subsequent uses. Under “Processing options” in the sidebar, select a window size for coarsening and click “Coarsen dataset”. The window size will indicate the number of pixels in the x and y directions over which a mean should be calculated. Once processing is complete, the status at the bottom of the page will show “Aggregation completed”.

12. After applying filters or processing to the data, you can save the results as a new .nc file. Under “Output file”, choose a name for the file. You can specify a filepath, or provide only a file name to have it saved in a default folder on your computer. Select “Save dataset”. Below, a message “File saved to: [filepath]” will indicate where the file is stored.
13. **Calculate summary statistics:** First select grouping options in the “Calculate statistics” panel, then click “Calculate statistics”.



Once processing is complete, the status at the bottom of the page will show “Calculations completed”, and a table of the summary statistics will appear in the “Statistics” panel. You can save this output as a new .csv file. Under “Output file for statistics”, choose a name for the file. You can specify a filepath, or provide only a file name to have it saved in a default folder on your computer. Select “Save statistics”. The status message will update to indicate where the file is stored.

**Statistics**

Output file for statistics:  Save statistics

month	hour	count	mean	std	min	25%	50%	75%	max
6	0	41340	4.999670	0.012425	4.000000	5.0	5.0	5.0	5.0
	6	41340	4.999678	0.012139	4.023438	5.0	5.0	5.0	5.0
	12	41340	4.999686	0.011863	4.046875	5.0	5.0	5.0	5.0
	18	41340	4.999693	0.011598	4.070312	5.0	5.0	5.0	5.0
7	0	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
	6	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
	12	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
8	0	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
	6	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
	12	42718	5.000000	0.000000	5.000000	5.0	5.0	5.0	5.0
9	0	39962	4.999299	0.014767	4.500000	5.0	5.0	5.0	5.0
	6	38584	4.999274	0.014962	4.515625	5.0	5.0	5.0	5.0
	12	38584	4.999274	0.014940	4.531250	5.0	5.0	5.0	5.0
	18	38584	4.999274	0.014962	4.515625	5.0	5.0	5.0	5.0

Calculations completed

### Subsetter

☰
⊞
☀
⊝
⊚

**Apps** +

- Tracks Explorer
- Gridded Data Explorer
- Subsetter
- Movie Maker

### Create a subset!

GIS file

Bounding Box
Track Points
Bounding Geometry

Lat min

Lon min

Lat max

Lon max

Clip features at boundary edge

Output file

Show plot of subset

Create subset

Ready...

## App features

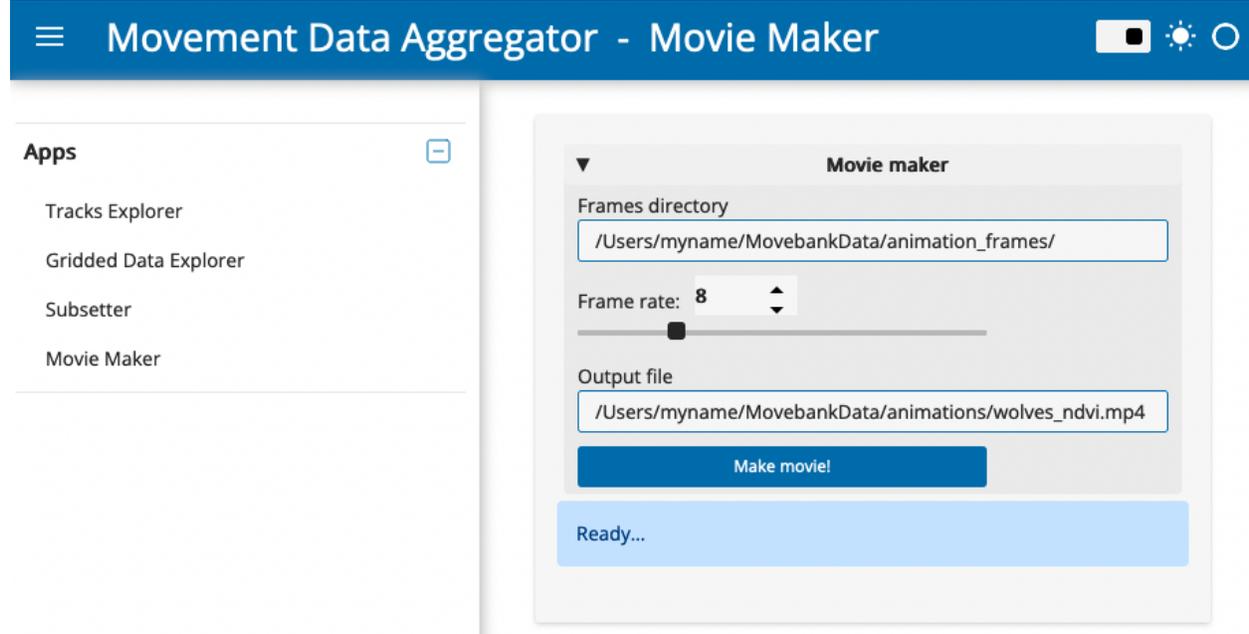
With the Subsetter App, you can

- Clip relevant features out of large shapefiles based on a bounding box, the extent of a tracking dataset, or a custom polygon.
- Save the extracted subset for further use.

## Using the app

1. In the “GIS file” field, paste the full filepath to a GIS dataset (e.g., a shapefile or file geodatabase) from which you want to extract a subset. The filepath should include the file name, or if the dataset consists of multiple files (e.g., for a shapefile) the folder in which the files are saved. An example filepath is “/Users/myname/global\_roads\_dataset.gdb”.
2. Next, define the spatial area for which you want to extract data from the GIS dataset. You can do this using a bounding box, track points, or a bounding geometry. There are specific settings for each option:
  - **Bounding Box:** Specify the latitude and longitude range for a rectangular bounding box. Provide coordinates in decimal degrees, WGS84 coordinate reference system.
  - **Track Points:** Provide the full filepath, including the filename, for a .csv file of tracking data containing [location-long](#) and [location-lat](#) in decimal degrees, WGS84 coordinate reference system. The app will draw a boundary around the track data: choose whether to do this using a rectangular or convex hull method, and optionally define a buffer around which to expand the boundary. This buffer is relative to the extent of the tracking data (the selected buffer size times the greater of the north-south or east-west extent).
  - **Bounding Geometry:** Provide the full filepath, including the filename, for a file (.geojson, .shp or .gdb) defining a polygon. This could include output of the *Tracks Explorer App*.
3. For any of the options above, decide whether to clip features that overlap with the edge of the spatial area defined in the previous step. Uncheck “Clip features at boundary edge” to retain complete lines or polygons that overlap with the boundary edge. By default, the app will plot the data subset once it is created, along with the geometry that was used to create the subsetting boundary, and the actual geometry that was used to define the subset. If you don’t want to plot the dataset, uncheck “Show plot of subset”.
4. Change the name of the output file and specify a filepath if you wish. Then click “Create subset”. A shapefile will be saved for the subsetted dataset, and a plot will appear unless you turned the plotting option off.

## Movie Maker



## App features

With the Movie Maker App, you can

- Create an animation file out of a stack of static maps that become frames in the animation.
- Animations can include tracking data along with other layers, including *environmental data from NASA Appears*, elevation contours, and other features. Frames using these layers can be created with [ECODATA-Animate](#).

## Using the app

1. Paste the full path to a folder of images to use as frames of the animation. You can create these frames using [ECODATA-Animate](#). Make sure there are no other files in the folder.
2. Select the frame rate for the video, in frames per second.
3. Optionally, change the filename and filepath defining where to save the output.
4. Click “Make movie”. The movie will be saved as a .mp4 file in the selected folder. Be sure not to change the folder location or name while the app is processing.

---

**Tip:** If you have issues with the video not playing properly, check whether it will play properly using [VLC Media Player](#). It may be a format compatibility issue.

---

## 3.3 Environmental data

### 3.3.1 Requesting environmental data from NASA

You can use a .geojson file from the *Tracks Explorer app* to submit a request for environmental data from NASA's Application for Extracting and Exploring Analysis Ready Samples (AEEARS) service.

1. Go to NASA AppEEARS: <https://appears.earthdatacloud.nasa.gov/>
2. These data are free, but you will need an EarthData account to request data. Register at <https://urs.earthdata.nasa.gov/users/new>, and then sign in with your EarthData account.
3. In the top menu bar, click "Extract > Area".
4. Click "Start new request". The interface looks like this:

#### Extract Area Sample

**Enter a name to identify your sample**

**Selected file**

To clear a polygon, draw a new polygon or upload a vector polygon file.

**Upload a file or draw a polygon using the or icon**

Drop a vector polygon file containing the area feature(s) to extract or [click here](#) to select the file.

Supported file formats:

- Shapefile (.zip including .shp, .dbf, .prj, and .shx files)
- GeoJSON (.json or .geojson)

**Start Date**

Is Date Recurring?

**End Date** ⓘ

**Select the layers to include in the sample** ⓘ

**Terra MODIS Vegetation Indices (NDVI & EVI)** ×

*MOD13A1.006, 500m, 16 day, (2000-02-18 to Present)*

ⓘ _500m_16_days_EVI <span style="float: right;">+</span>
ⓘ _500m_16_days_MIR_reflectance <span style="float: right;">+</span>
ⓘ _500m_16_days_NDVI <span style="float: right;">+</span>

**Selected layers**

Select the layers to include in the sample

5. In the Extract Area Sample page, first upload the .geojson file you created in the Tracks Explorer. Alternatively, you can draw a polygon on the page or upload another .geojson or .shp file. It will display a preview of the area

for which data will be provided.

6. Select a date range. You can choose calendar dates for the start and end of the period, or select “Is Date Recurring?” to select a specific range of dates across years.
7. Select the data layers you want. Type a product name (e.g., “MYD10A2”) or description (e.g., “snow”) and it will display a list summarizing relevant products. To review all options, available products are listed at <https://appears.earthdatacloud.nasa.gov/products>. Also see the searchable catalog of many of these products at [https://lpdaac.usgs.gov/product\\_search/?status=Operational](https://lpdaac.usgs.gov/product_search/?status=Operational). Once you have selected a product(s) from the list, click the “+” button on each desired layer. The layers will appear in the “Selected Layers” box to the right.
8. In “Output options” at the bottom of the page, select “NetCDF-4” as the file format and “Geographic” as the projection, like this:

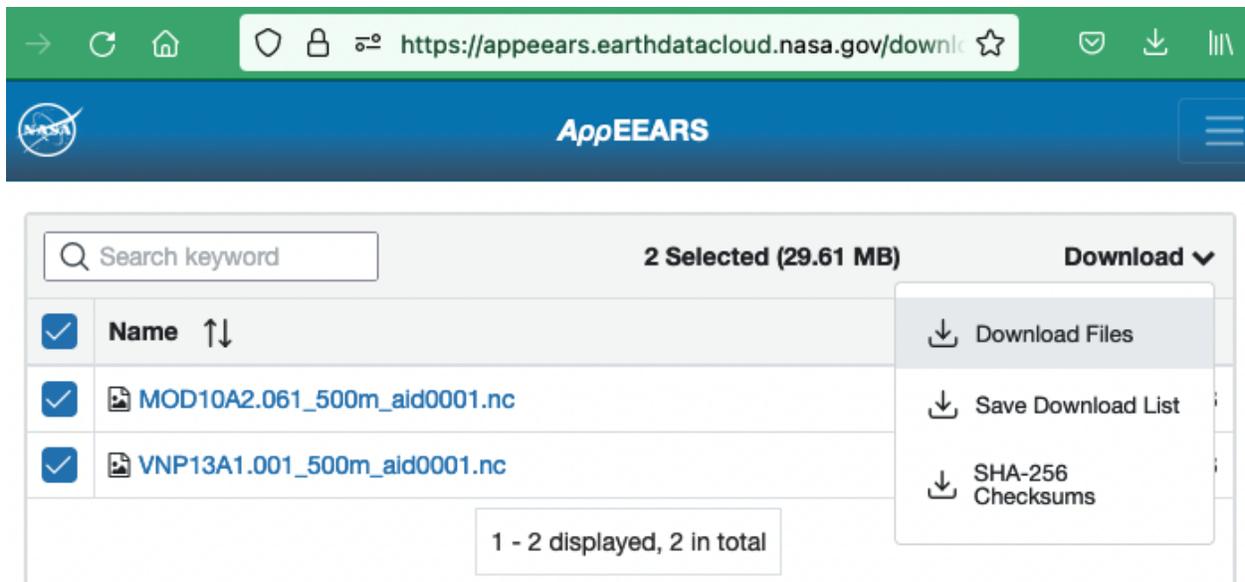
**Output Options**

**File Format:** NetCDF-4

**Projection:**

**Geographic** ✕  
Datum: WGS84  
EPSG: 4326  
PROJ.4: +proj=longlat +datum=WGS84 +no\_defs

9. Select “Submit”. Your submission will be added to a queue and you will receive emails from [appears-noreply@nasa.gov](mailto:appears-noreply@nasa.gov) confirming the submission and notifying you when the data are ready for download. Keep in mind that notifications could end up in your spam box. The time needed to complete the request likely depends on the volume of data requested and the number of other requests their service has recently received. In our tests so far they have come through within a few hours.
10. When your data are ready for download, you will receive an email from [appears-noreply@nasa.gov](mailto:appears-noreply@nasa.gov). Click the “Download” link in the email.
11. The AppEEARS website will open in a new tab (log into your account if prompted) and you will see a list of files available to download. Supporting files are listed first, however you will need to download the resulting NetCDF (.nc) file for the next step in the ECODATA-Prepare app (Gridded Data Explorer).
12. Check the box to the left of the file name and select Download > Download Files at the right. The downloaded file will appear in your Downloads folder, which you can then move to a local folder. We also recommend downloading at least the README.md file under Supporting files. This will document details of your request and results that you may need to return to later, including data quality information and how to cite your results.



The screenshot shows the AppEEARS web interface. At the top, there is a NASA logo and the text "AppEEARS". Below this is a search bar with the placeholder "Search keyword". To the right of the search bar, it says "2 Selected (29.61 MB)" and "Download" with a dropdown arrow. The table below has two columns: a checkbox column and a "Name" column with an up/down arrow. Two files are listed and selected: "MOD10A2.061\_500m\_aid0001.nc" and "VNP13A1.001\_500m\_aid0001.nc". A dropdown menu is open to the right of the table, showing three options: "Download Files", "Save Download List", and "SHA-256 Checksums". At the bottom of the table, it says "1 - 2 displayed, 2 in total".

<input checked="" type="checkbox"/>	Name ↑↓	
<input checked="" type="checkbox"/>	 MOD10A2.061_500m_aid0001.nc	<input type="checkbox"/>
<input checked="" type="checkbox"/>	 VNP13A1.001_500m_aid0001.nc	<input type="checkbox"/>

### 3.3.2 Requesting environmental data from ECMWF

ECMWF ERA5 hourly data can be requested [here](#). Note that you will need to make an account to request data.

To request data:

1. Go to the [data request page](#).
2. In the “Variable” section, select the variables you wish to download, for example as shown here:

## ERA5 hourly data on single levels from 1940 to present

Overview

Download data

Quality assessment

Documentation

Clear all

### Product type

Reanalysis  
 Ensemble spread

Ensemble members

Ensemble mean

Select all Clear all

### Variable ?

#### Popular

10m u-component of wind  
 2m temperature  
 Mean wave period

10m v-component of wind  
 Mean sea level pressure  
 Sea surface temperature

2m dewpoint temperature  
 Mean wave direction  
 Significant height of combined wind waves and swell

Surface pressure

Total precipitation

Select all Clear all

3. Select the times of interest.
4. Under “Geographical area”, select “Sub-region extraction” and enter the bounding box coordinates for the area of interest (max and min longitude and latitude).
5. Under “Format”, select “NetCDF (experimental)”.
6. If you have entered all required information and are logged in, you will have the option to “Submit Form” and send the request. An example of selections and options for steps 4-6 is shown here:

**Geographical area** ?

**Whole available region**

With this option selected the entire available area will be provided

**Sub-region extraction** ?

North  
55

West  
-120.8

East  
-117.8

South  
53

**Format**

GRIB  NetCDF (experimental)

[Clear all](#)

**Terms of use**

Licence to use Copernicus Products [View terms](#)

[Show API request](#) [Show Toolbox request](#) [Submit Form](#)

## 3.4 Support

ECODATA is in the early stages of development, and any feedback is welcome! If you have any suggestions or feature requests, encounter any bugs, or come across places where the documentation is unclear, please [submit a Github issue](#).

## 3.5 Developer guide

### 3.5.1 Contributing

Check out this [simple guide for using git](#).

1. Create a new branch for your contributions.
2. Commit your changes in this branch.
3. [Open a pull request](#) to merge changes from your branch into the repository's `develop` branch.

### 3.5.2 Documentation

Documentation for this project is created using Sphinx and is hosted at Read the Docs (<https://ecodata-apps.readthedocs.io/>). The source files for these pages are located in the `docs/apps` folder of the repository. To edit the documentation, edit the markdown files in this folder (or sub-folders). Note that the `docs/index.md` file specifies the contents for the docs site. If a sub-folder has a `index.md` file, that file specifies the contents for that section of the docs site (e.g. `docs/user_guide/index.md`). If files are added or removed, the corresponding index files will also need to be updated.

### Building the docs

After editing the pages, you can look at a build of the pages to see how things will actually look in the docs website. There are two options for this:

- Option 1: [Open a pull request](#), and Read the Docs will build a preview of the docs pages. A link to the build can be found near the bottom of the page of the PR, in the merge checks section (once the build is finished, click on “Details” for the `docs/readthedocs.org:ecodata-animate` item. You may have to click “Show details” next to where it says “All checks have passed”). You can push additional commits to the open PR if you want to change anything after seeing the preview build.
- Option 2: Build the docs locally. You will need to have python and the docs requirements installed.
  - To install the doc requirements: *[Developing Installation instructions](#)*
  - Then Run `PROJECT=apps sphinx-build -b html docs docs/_build/apps` from the repo's root directory
  - To view the build, open the `index.html` in the `docs/_build/apps` directory that was created.

## Versions of the docs

- Read the Docs builds multiple versions of the documentation (for different branches of the repository). In the bottom corner of the docs pages, there is a box indicating which version you are viewing. You can click on that box to pick a different version.

## Developer installation

1. Clone the repository, and navigate to the root directory of the cloned repo
2. Create the environment for the package:

```
mamba env create --file ecodata-env.yml --name eco-dev
```

3. Activate the eco-dev environment (this will install the package in editable mode):

```
mamba activate eco-dev
```

4. With the eco-dev environment activated, install the additional dev requirements (needed for docs, testing, code style, etc):

```
mamba env update --name eco-dev --file ecodata-dev-env.yml
```



## INDICES AND TABLES

- genindex
- modindex
- search