[Name Pending] User Manual

MOWS

**Introduction**

This software is meant to simulate the spread of invasive worms over a specified region

**Installation**

To run this software, you need NetLogo (version 5.3.1 or later). You can get the latest version of NetLogo from [here](https://ccl.northwestern.edu/netlogo/download.shtml). Once NetLogo is installed, you can move onto starting the simulation program.

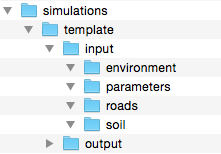
Tip: We recommend you look at NetLogo’s own [user manual](https://ccl.northwestern.edu/netlogo/docs/) to get familiar with the general interface. Click on ‘Interface Guide’ on the left panel in the online manual. We will provide additional information about program specific controls later in this document.

**Starting the Program**

Double click the “worm\_sim\_main.nlogo” file present in the main ‘worm\_simulation’ folder to start the program. This should fire up NetLogo and you will be greeted to the interface tab.

Alternatively, you can start NetLogo and then go to File>Open in the menu bar and open the “worm\_sim\_main.nlogo” file from there.

**Setting up Folders**

If you need to save/load data or import GIS, the software requires a specific folder hierarchy. All simulations will be present inside the “simulations” folder. Each simulation directory will have a main folder, which will have the name that you will use in the program. Sample simulation folders are provided with the program. One of them is shown below:

This will be the name you put in the “save\_name” box in NetLogo

All simulations must have these folders following the hierarchy shown

You can use the sample simulation folders for testing purposes and as templates for future simulations. Duplicate the template directory by right clicking on it and selecting “Duplicate”. Replace the top folder’s name from “template” to your desired simulation name. All the inner folders should have the same names as the template’. Input data for your simulation will go into these folders as described further down.

**Placing Input Data**

**Setting up a Simulation**

Once all the files are in the correct folders, we can start a simulation from NetLogo. The following steps will guide you through running a simulation. We will use “testSim” as a name for our simulation but you may use the name you chose.

1. Click Initialize
2. Click Load GIS

Note: This may take some time depending on your machine

1. Enter the command *initialize\_monitors* into the Command Center at the bottom

Note: This step will add preset monitors to the simulation. You may choose to skip this step and add your own monitors and save their names. For more information, look at the Adding Monitors section.

1. Click Save Environment

Note: This is to save the environment for faster loading in the future.

1. Under *Species Control*, click on the *species\_number* drop down menu and select “1”
2. Slide the *worm\_population* slider to 100
3. *ph\_tolerance, temperature\_tolerance,* and *species\_genetic\_diversity* should be set to zero and *Random\_Insertions?* should be set to false
4. Set the speed to 0.5

Note: This speed should ideally be calculated according to the dimensions of the region you are simulating.

1. Click Add
2. Click on any point on the map

Note: This will add the worms onto a single point on the map. Alternatively, you could select a region and add them randomly. For more information, look at the Adding Worms section.

1. Click *Save* under *Species Control*
2. Enter 10 into *number\_of years* text box
3. Click on Go

You should see the simulation start and run for 10 years. Once it finishes, it will store all the data collected into the output folder in your simulation directory.

Tip: Next time, if you want to load the same simulation environment and agents, you only need to click on *Setup Simulation*

**Running Multiple Simulations**

We will now look into a tool called *BehaviorSpace* for running multiple automatic simulations.

1. Delete everything from the output folder in your simulation directory
2. Click on Tools in the Menu Bar
3. Select BehaviorSpace
4. Click New
5. Type in test as the experiment name
6. Delete everything in the text box holding all the variables
7. Type in the following text instead (do not copy/paste): [“ph\_tolerance” 0 0.1]

Note: This will run two simulations, one with *ph\_tolerance* as 0 and the another with 0.1

1. Set *Repetitions* to 3
2. Add the following command to *Setup Commands:* setup\_bs
3. Click OK
4. Select *test* from the BehaviorSpace menu and click Run
5. Make sure *Spreadsheet output* and *Table output* are unchecked
6. Click OK

The simulations should start running now. You can increase the slider from *normal speed* to the right to make the simulations run faster. After the simulations are done, we can look into plotting the results.

**Plotting Results**

You will first need to open the main folder where the program resides.

1. Open simParams.txt
2. Delete everything if it is not blank
3. Add the following line to it: test 3 [“ph\_toelrance” 0 0,1]

Note: The format for plotting parameters will always be the same.

*<save\_name> <repetitons> <BehaviorSpace argument>*

1. Save the file and exit
2. Double click *plot.command* in the main folder

You should now see a bar chart showing densities across five regions for the two different pH tolerance levels

**Interface Guide**

**Program Controls**

This section of the guide will look at controls specific to the program