CNES Large Scale SWOT Simulator

User's Tutorial for Terrestrial Surface Water Applications

Nicholas J. Elmer
NASA Postdoctoral Program, NASA Marshall Space Flight Center
NASA Short-term Prediction Research and Transition (SPoRT) Center
Huntsville, Alabama, USA
April 27, 2020

Contents

1. Introduction

The CNES Large Scale SWOT Hydrology Simulator creates proxy SWOT water surface elevations (WSE). A large portion of the effort required to run the CNES simulator is upfront in creating the simulator input. This tutorial describes the steps to create the simulator input and then run the CNES simulator. Since many applications will utilize a time series of proxy SWOT WSE, a short description of that process is included in the Appendix for advanced users.

2. Download and Set-up the CNES Simulator

2.1 Linux / Mac

The CNES simulator can be downloaded from Github at https://github.com/CNES/swot-hydrology-toolbox. The README.md file, replicated below, describes the cloning process using the "git clone" command, setting environment variables, and creating the conda (https://www.anaconda.com/products/individual) environment.

Get the repository

1. Clone **swot_hydrology_toolbox** repo

```
% git clone https://github.com/cnes/swot-hydrology-toolbox.git
```

The repository **swot_hydrology_toolbox** should be assignated to the **SWOT_HYDROLOGY_TOOLBOX** variable.

```
% export SWOT_HYDROLOGY_TOOLBOX=your_installation_path/swot-hydrology-toolbox
```

2. Clone **RiverObs** repo

```
% git clone https://github.com/SWOTAlgorithms/RiverObs.git
```

The repository **RiverObs** should be assignated to the **RIVEROBS** variable.

```
% export RIVEROBS=your_installation_path/RiverObs
```

Python environment installation

Setting up a conda environment

To create a conda environment, execute

```
cd $SWOT_HYDROLOGY_TOOLBOX
conda env create -f environment.yml
```

To activate this environment, if the first option was used, type

```
conda activate swot-env
```

To deactivate this environment, type

```
conda deactivate
```

Execute the toolbox

After activating your Python environment, you have to set your PYTHONPATH variables:

```
export PYTHONPATH=$SWOT_HYDROLOGY_TOOLBOX/processing/src/:$RIVEROBS/src:$PYTHONPATH
```

2.2 Windows / PC

Although Windows/PC users can use Git Desktop (https://desktop.github.com/) to download the simulator, the steps below demonstrate an approach that may be used by those users unfamiliar with git.

1. Download the zipped folder (.zip) of CNES swot-hydrology-toolbox in the directory of your choice from https://github.com/CNES/swot-hydrology-toolbox. Right-click on the file in Windows Explorer to unzip/extract.

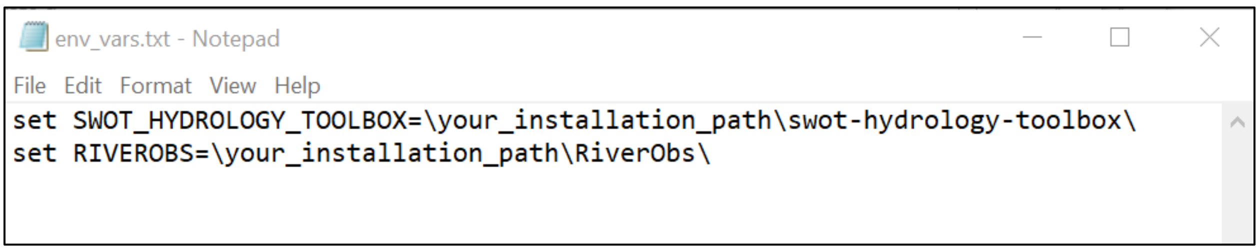
2. With anaconda installed (https://www.anaconda.com/products/individual), open Anaconda Prompt (Go to Windows Search bar (lower left of desktop, type "Anaconda Prompt").

   Note: Anaconda Prompt basic commands are `cd` (change directory) and `dir` (list contents).

3. Set environment variables for the simulator. Additional background information and documentation can be found at: https://docs.conda.io/projects/conda/en/latest/user-guide/tasks/manage-environments.html#saving-environment-variables

    a. Define the environment variables. In Windows Explorer, go to C:\path\to\conda\anaconda3\envs\swot-env\etc\conda. There should be an activate.d and deactivate.d directory. If unsure of your \path\to\conda, type "%CONDA_PREFIX%" in Anaconda Prompt to list \path\to\conda.

    b. Create env_vars.bat in the activate.d folder to add "set SWOT_HYDROLOGY_TOOLBOX=\your_installation_path\swot-hydrology-toolbox\" and "set RIVEROBS=\your_installation_path\RiverObs\" as separate lines. The easiest way to create this file is in Windows Explorer. Open a simple text file, add the set commands, and then save. Then copy and change extension from .txt to .bat.



    c. Now, create env_vars.bat in the deactivate.d folder to add "set SWOT_HYDROLOGY_TOOLBOX='''' and "set RIVEROBS='''' as separate lines. Follow the same steps as in (c) above.

4. Create swot-env conda environment.

    a. Create the environment from the provided .yml file.

    ```
    >>cd %SWOT_HYDROLOGY_TOOLBOX%
    >>conda env create –f environment.yml
    ```

    b. Activate the swot-env environment. To activate, type:

    ```
    >>conda activate swot-env
    ```

    Note: To deactivate later after running the simulator, if needed, you simply need to type: >>conda deactivate

5. Finally, we need to make sure python can find all of the modules needed for the simulator. With the swot-env environment activated, on the command line type the following commands:

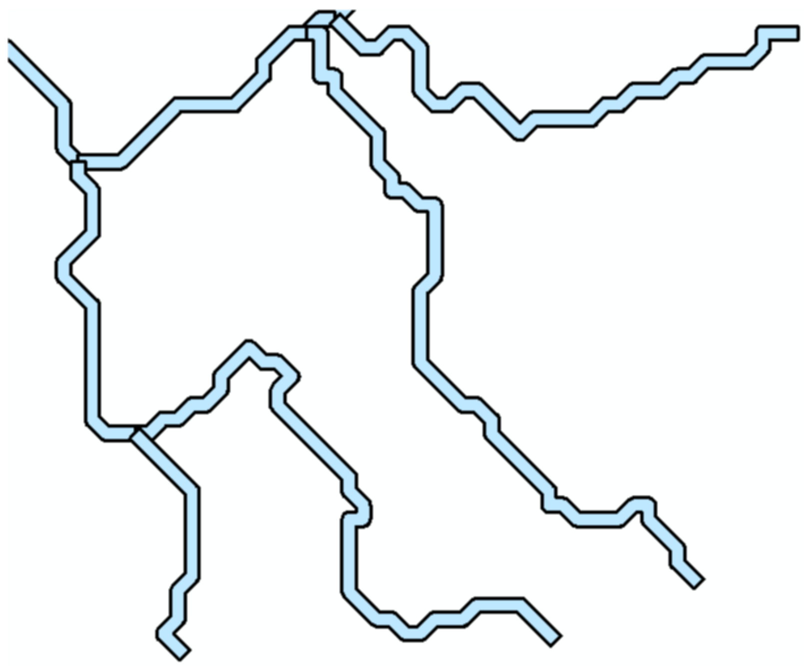
```
>>conda develop %SWOT_HYDROLOGY_TOOLBOX%\processing\src
>>conda develop %RIVEROBS%\src
```

3. Create the input polygon shapefile(s)

Once the CNES Simulator is set up, you need to create the required polygon shapefile input, one shapefile per observation time. The CNES simulator uses the polygon extent to calculate river width, whereas a height (HEIGHT) attribute specifies the initial WSE and a river flag (RIV_FLAG) designates polygons as river segments. There are many ways to create a polygon shapefile, and the ideal process will vary depending on your application. However, four methods are demonstrated below based on different starting points. Although the steps are based on ArcGIS, QGIS or even Python (for experienced users) can be used instead.

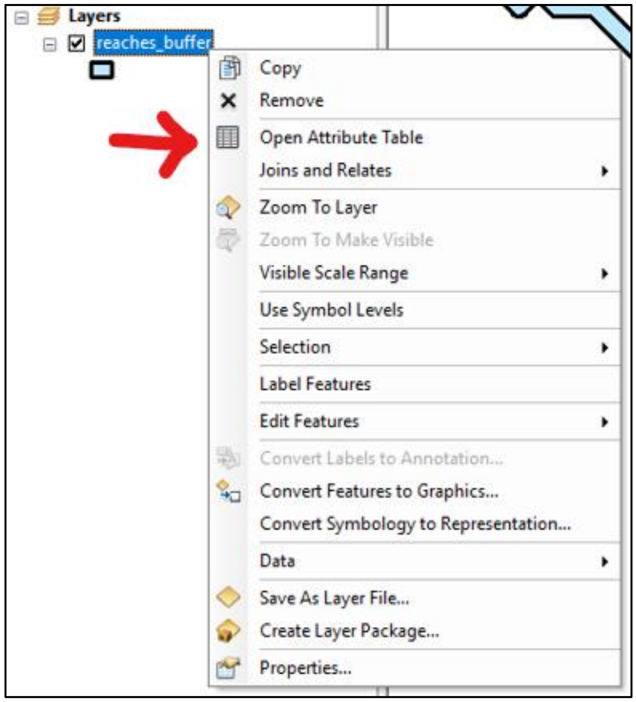
A. You already have a polygon shapefile

If you already have a polygon shapefile of your river network similar to what is shown below, you simply need to add surface height (HEIGHT) and river flag (RIV_FLAG) attributes for each river segment. Depending on which version of the simulator you are using, you may need to add an identification field (id) as well.

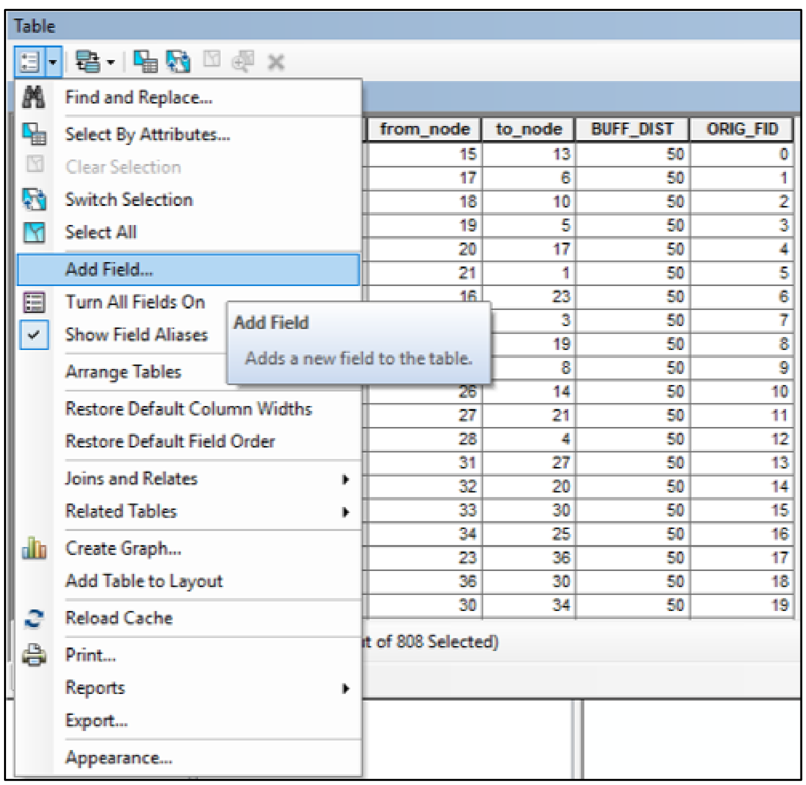


In ArcGIS:

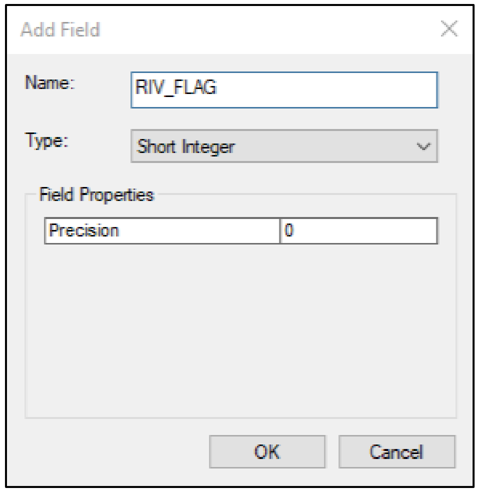
1. Right-click on your reach polygon, then click “Open Attribute Table”.



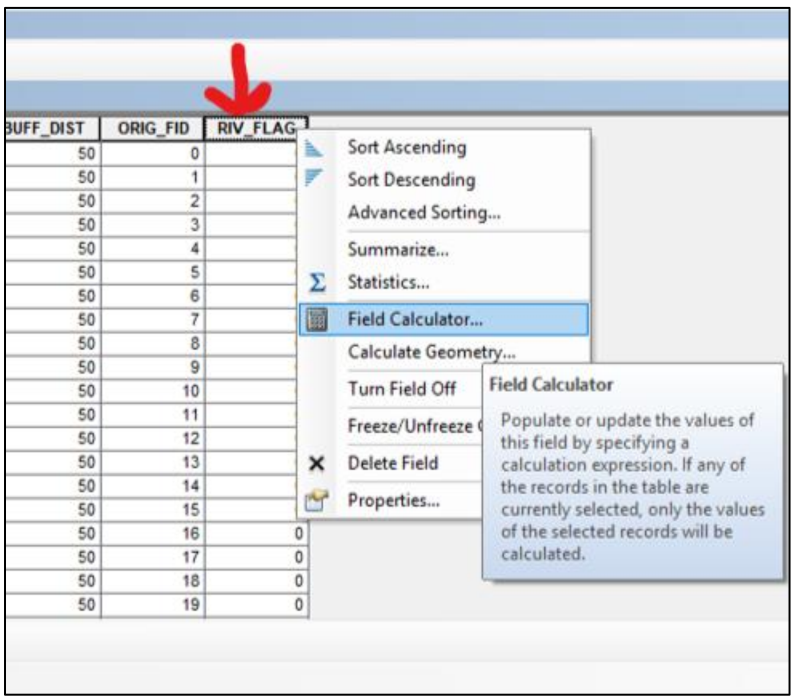
2. Click the “Table Options” dropdown and select “Add Field”.



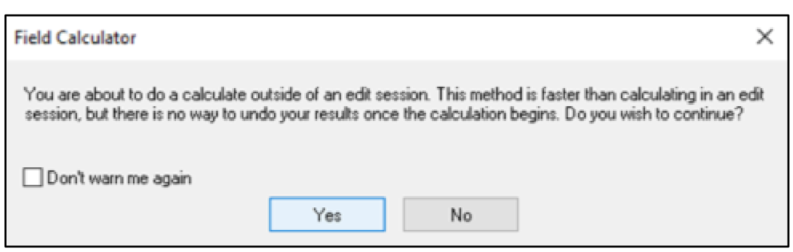
3. Name the field “RIV_FLAG” and specify the type as a short integer. Click “OK”.



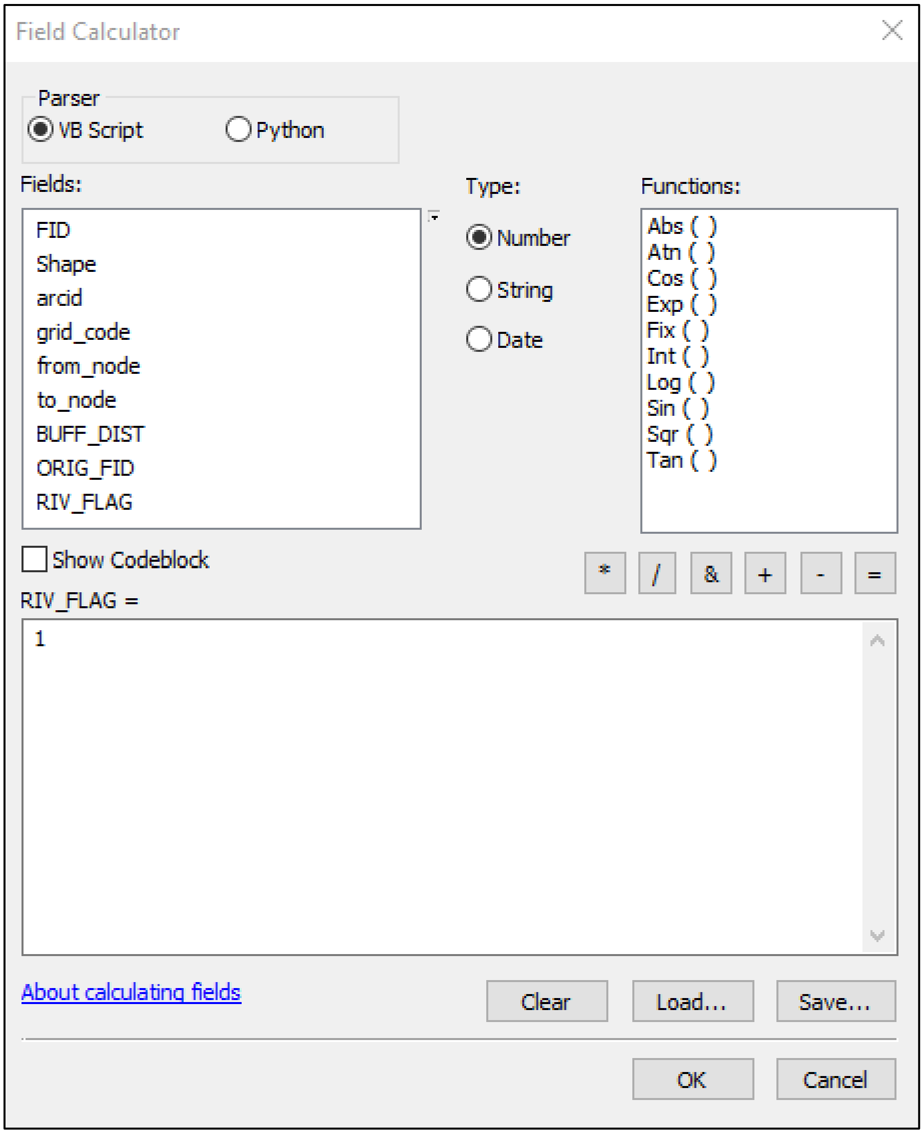
4. Right-click on the RIV_FLAG header and select “Field Calculator”.



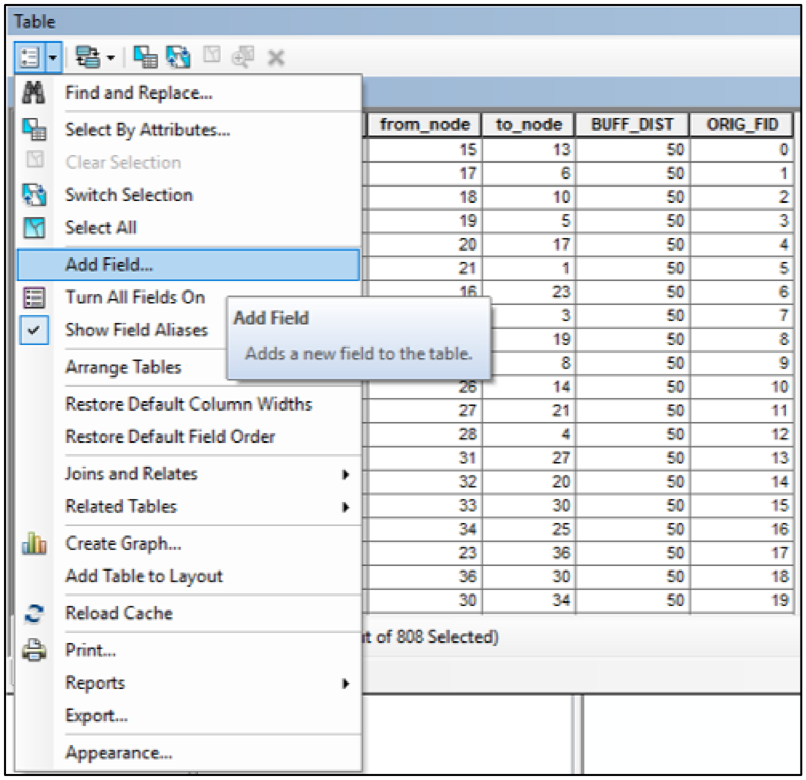
5. If you receive this message, click “Yes”.



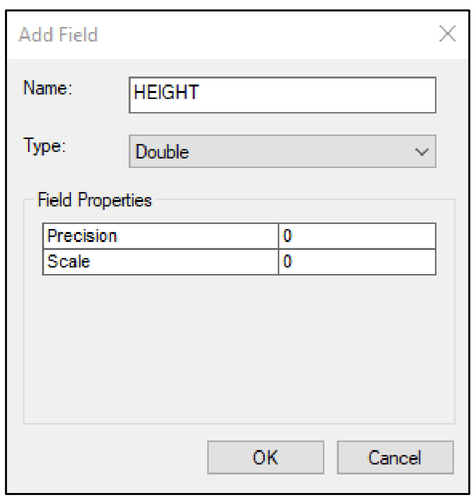
6. Set RIV_FLAG to “1” for all river segments. RIV_FLAG should be set to 1 for river segments, but 0 for an lakes or reservoirs along the network. Then click “OK”.



7. Now add the HEIGHT attribute. Click the “Table Options” dropdown and select “Add Field”.



8. Name the field “HEIGHT” and specify the type as “Double”. Click “OK”.



9. Populate the HEIGHT field. Height values can be manually assigned from observations (in situ or satellite) or a best estimate using the Field Calculator. For complex height assignments, such as those defined by model output, you can use a simple Python script using the Pyshp module (https://pypi.org/project/pyshp/) to define the HEIGHT field programmatically. If you are an advanced user and wish to process a time series using the CNES simulator, refer to the Appendix. The final attribute table for the input shapefile is shown below (Note that only the HEIGHT and RIV_FLAG fields are required).

201209010200_SusitnaRiver_height_reach

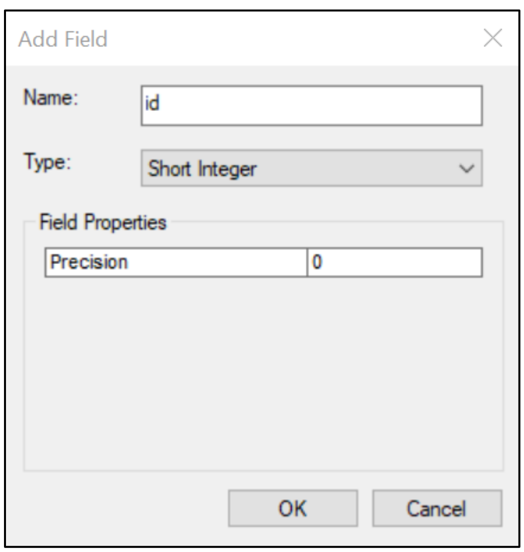
FID	Shape	arcid	grid_code	from_node	to_node	halfwidth	midlat	midlon	BUFF_DIST	ORIG_FID	HEIGHT	WIDTH	link	RIV_FLAG
0	Polygon	2349	5	2320	2496	62.5	63.309616	-147.348604	62.5	0	773.1807	125	2350	1
1	Polygon	2462	5	2496	2601	62.5	63.289706	-147.406328	62.5	1	768.3092	125	2463	1
2	Polygon	2505	5	2601	2642	62.5	63.273131	-147.42358	62.5	2	765.8078	125	2506	1
3	Polygon	2569	5	2642	2703	62.5	63.267518	-147.433082	62.5	3	764.2208	125	2570	1
4	Polygon	2735	5	2703	2884	62.5	63.250622	-147.448351	62.5	4	761.9642	125	2736	1
5	Polygon	2794	5	2884	2937	62.5	63.226147	-147.467224	62.5	5	759.4504	125	2795	1
6	Polygon	2838	5	2937	2986	62.5	63.215489	-147.479786	62.5	6	758.5098	125	2839	1
7	Polygon	3026	5	2986	3180	62.5	63.195512	-147.495404	62.5	7	756.9344	125	3027	1
8	Polygon	3033	5	3180	3190	62.5	63.174808	-147.505726	62.5	8	754.9109	125	3034	1
9	Polygon	3216	5	3190	3375	62.5	63.159357	-147.51017	62.5	9	751.8478	125	3217	1
10	Polygon	3362	5	3375	3528	62.5	63.127082	-147.511037	62.5	10	746.8743	125	3363	1
11	Polygon	3369	5	3528	3534	62.5	63.115571	-147.52043	62.5	11	745.04	125	3370	1
12	Polygon	3466	5	3534	3631	62.5	63.105082	-147.519335	62.5	12	744.1782	125	3467	1
13	Polygon	3669	5	3631	3845	62.5	63.078689	-147.52362	62.5	13	742.43	125	3670	1
14	Polygon	3693	5	3743	3870	62.5	63.035575	-149.615256	62.5	14	440.4758	125	3694	1
15	Polygon	3778	5	3845	3961	62.5	63.048005	-147.528084	62.5	15	738.5846	125	3779	1
16	Polygon	3783	5	3870	3966	62.5	63.018162	-149.633335	62.5	16	410.6104	125	3784	1
17	Polygon	3905	5	3966	4091	62.5	63.004926	-149.640056	62.5	17	394.3589	125	3906	1
18	Polygon	3927	5	4091	4109	62.5	62.992155	-149.658476	62.5	18	393.9034	125	3928	1
19	Polygon	3951	5	3961	4134	62.5	63.027077	-147.437126	62.5	19	733.2291	125	3952	1

1 (0 out of 368 Selected)

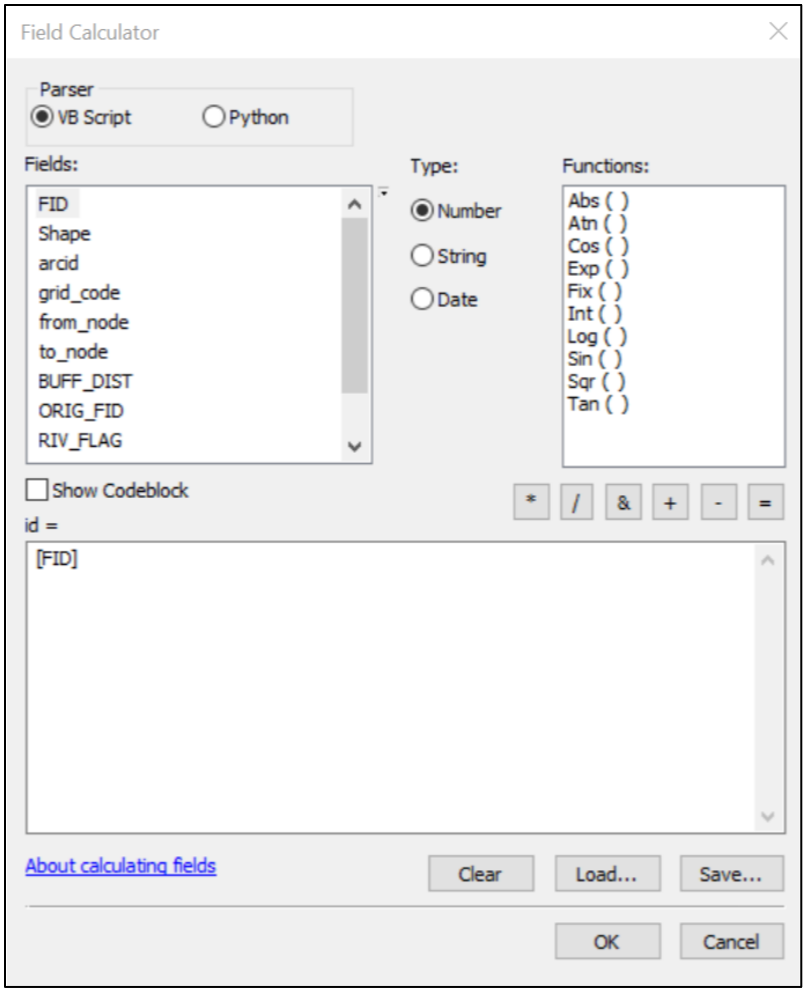
201209010200_SusitnaRiver_height_reach

In some cases, the entire river network of interest may be described by a single polygon. Since only one height value can be assigned per polygon, you will need to split the polygon into smaller pieces using the ArcGIS or QGIS editing and cutting tools (https://desktop.arcgis.com/en/arcmap/latest/extensions/3d-analyst/using-the-cut-polygons-tool.htm) if you wish to assign different height values to different areas of your river network. Once you have created the polygon shapefile with required attributes, move to Section 4 to run the CNES simulator.

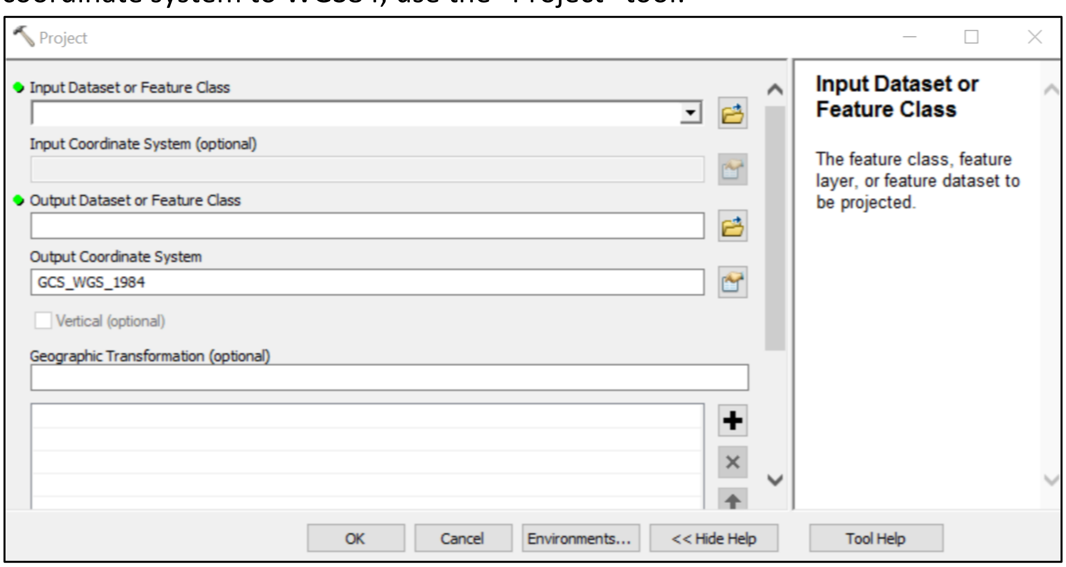
10. Now add the “id” attribute. Click the “Table Options” dropdown and select “Add Field”. Name the field “id” and specify the type as “Short Integer”. Click “OK”.



In the attribute table, right-click on the “id” header and select “Field Calculator.” Set id = [FID], and click “Ok”.

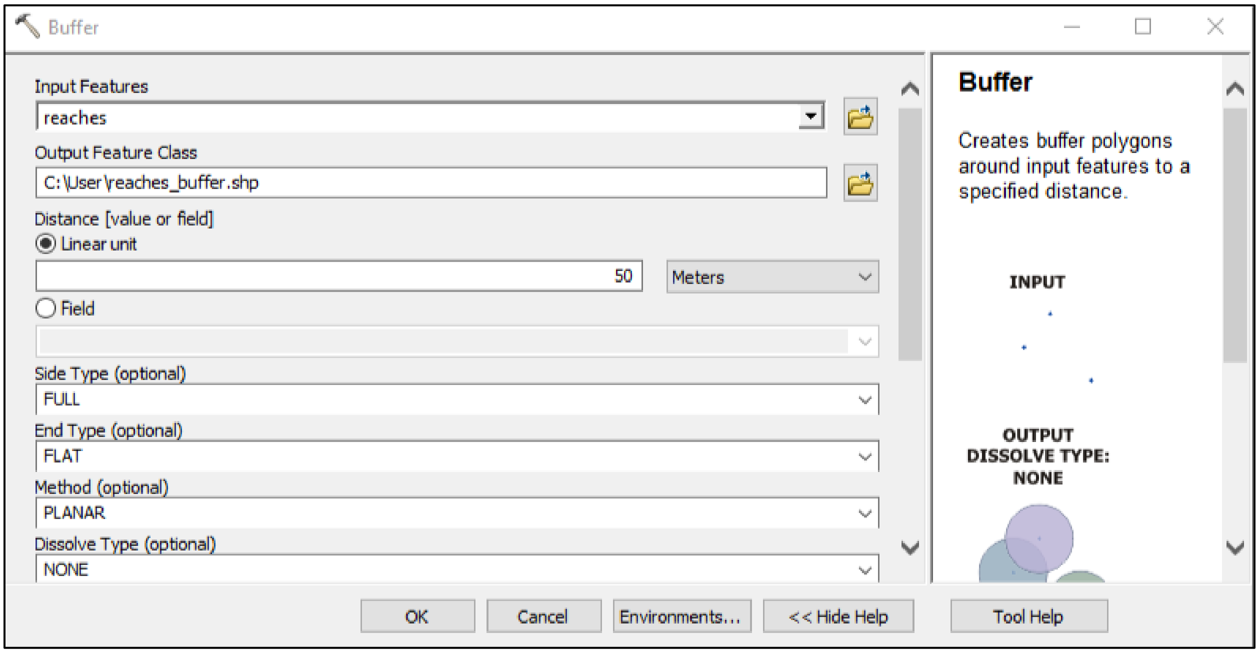


11. The CNES simulator requires that the input shapefiles have a WGS84 coordinate system. You can verify the coordinate system by looking at the shapefile properties. To set the coordinate system to WGS84, use the “Project” tool.

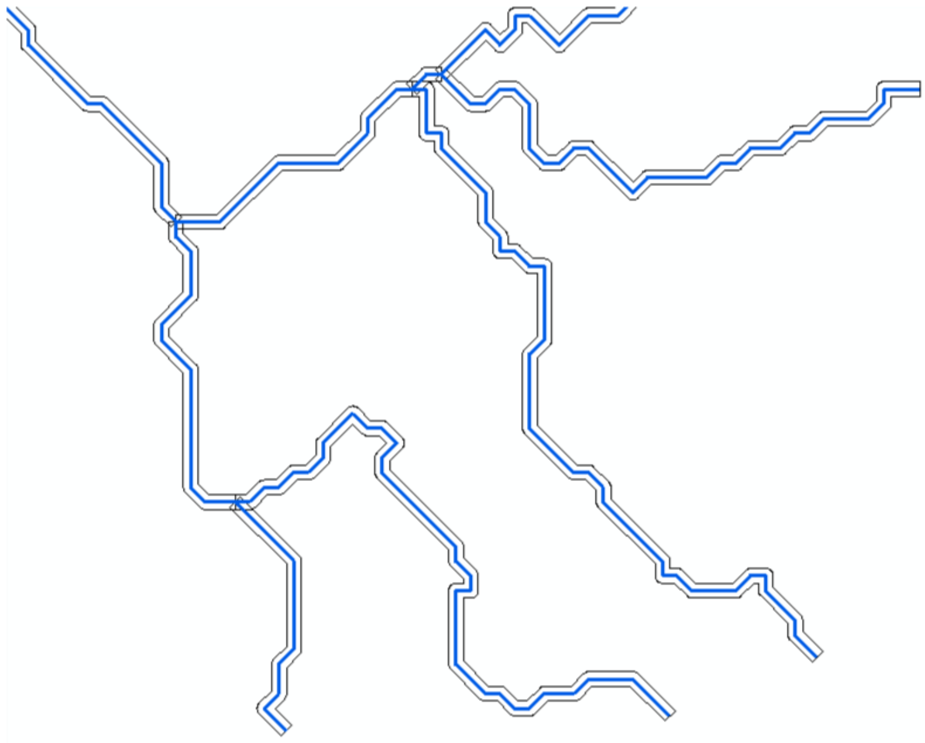


B. You have a polyline shapefile of river reaches

If you have a polyline shapefile of river reaches, you can easily convert to a polygon shapefile using the ArcGIS Buffer tool.

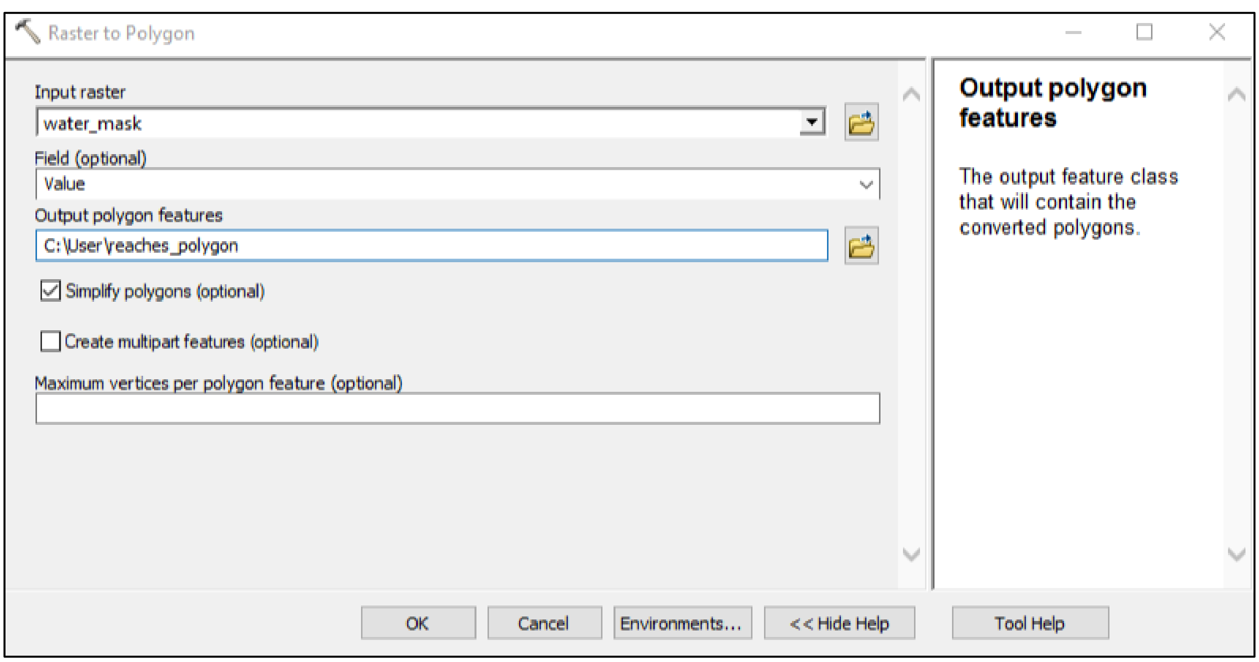


Distance can be specified using either a width field (if your polyline includes this attribute) or simply a constant linear value (e.g., 50 meters). Note that the distance is one-sided, meaning that to set a value of 50 meters results in a polygon width of 100 meters. A comparison between the polygon river reaches (black outline) and polyline river reaches (blue lines) is shown below. Once you have created the polygon shapefile, continue with the steps in Section 3A.

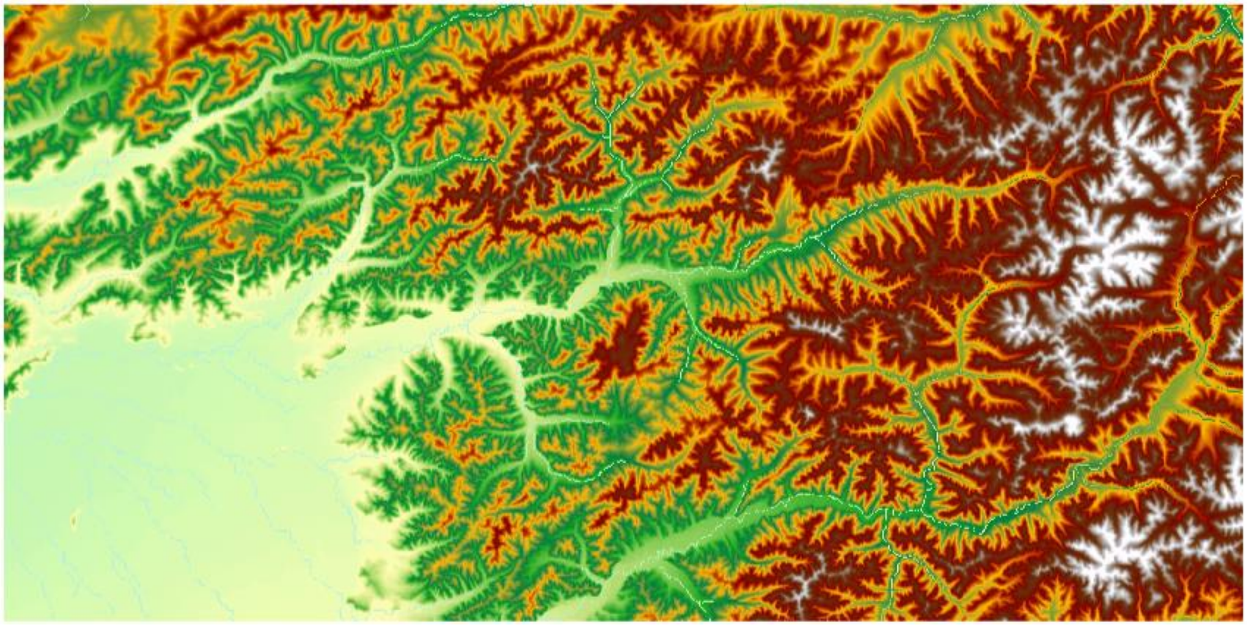


C. You have a water mask raster

If you have a raster with a water mask, you can create a polygon shapefile using the ArcGIS Raster to Polygon tool. Once you create the polygon shapefile, continue with the steps in Section 3A.



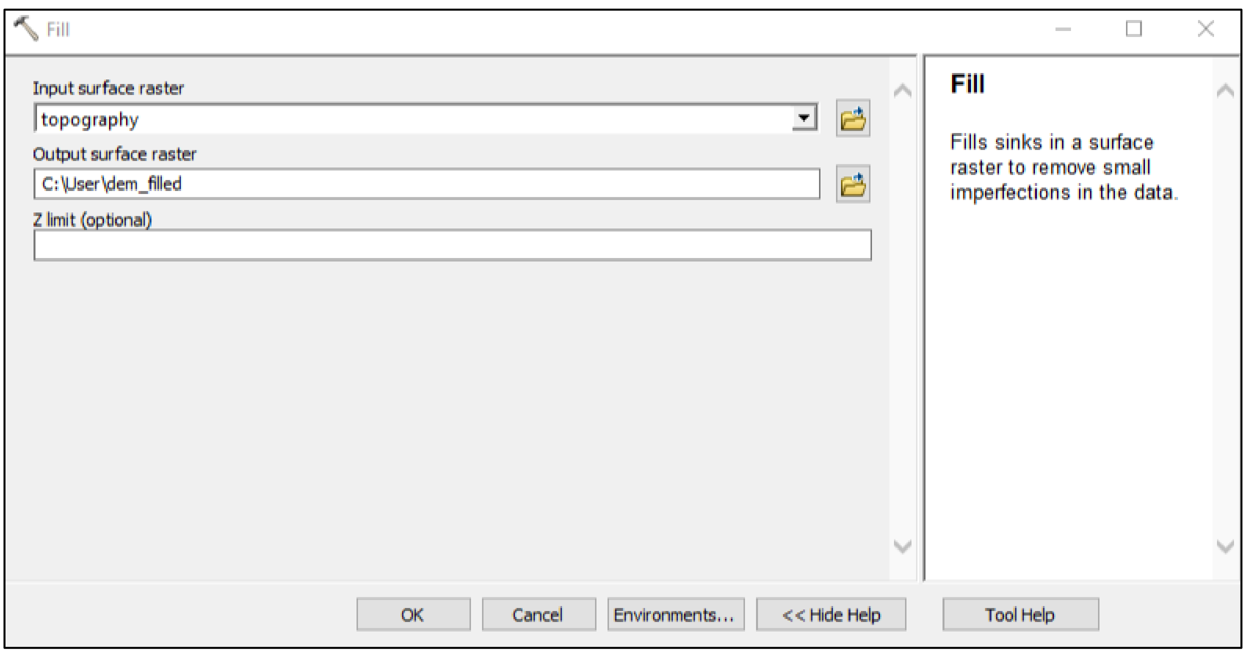
D. You have a Digital Elevation Map



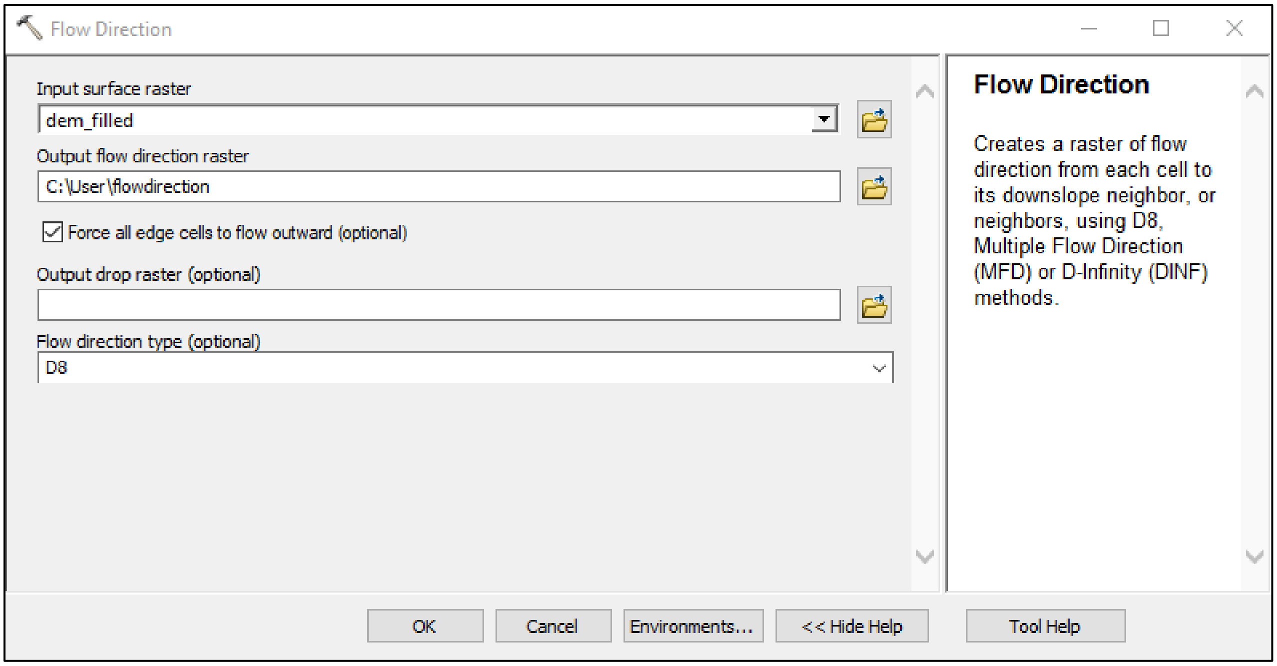
Example DEM from which a polygon shapefile of the river network can be derived.

If you have a Digital Elevation Map (DEM) similar to the figure above, you can derive the polygon shapefile using the ArcGIS Spatial Analysis Hydrology Toolbox. Using your DEM raster (named "topography" in this example), run the following tools in ArcGIS (similar tools are available in QGIS):

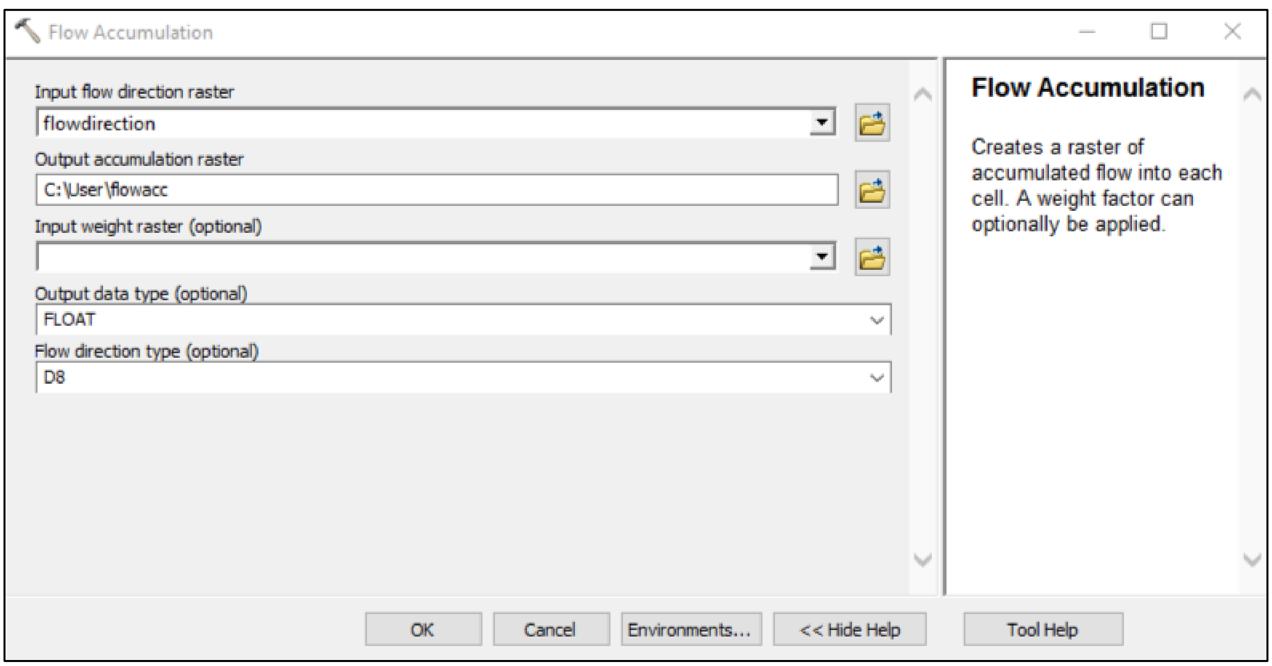
1. Fill. Hydrologically conditions your DEM to remove sinks.



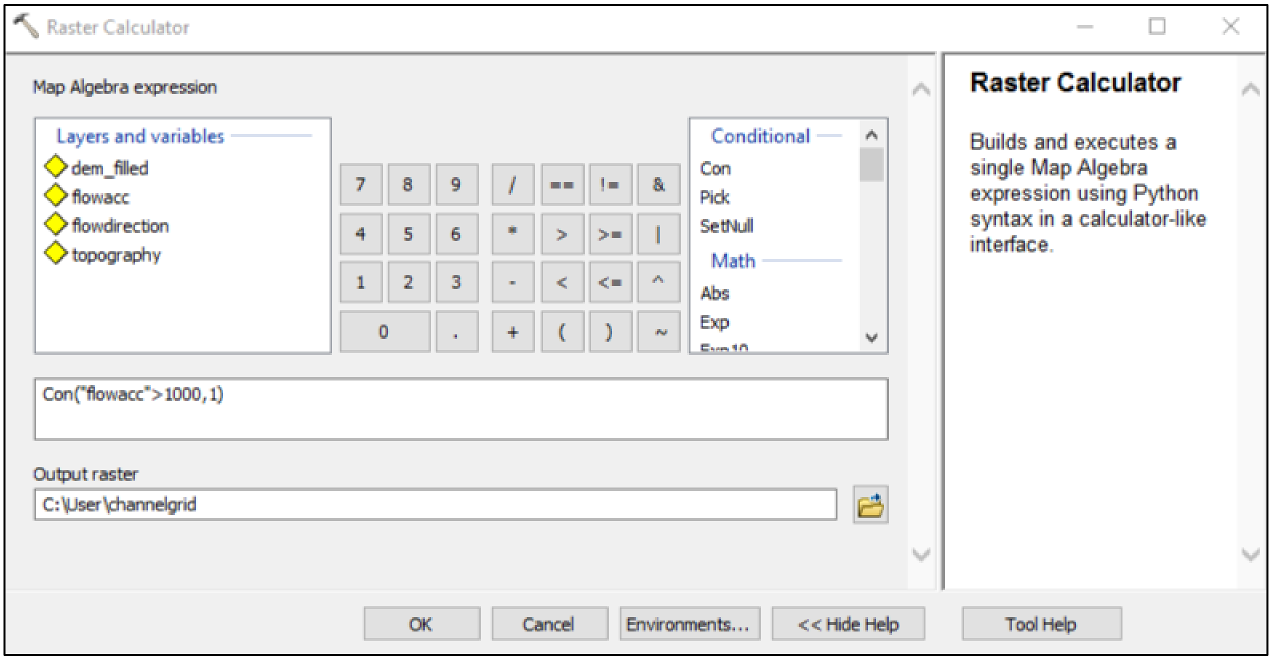
2. Flow Direction. Calculates the direction of flow based on elevation.



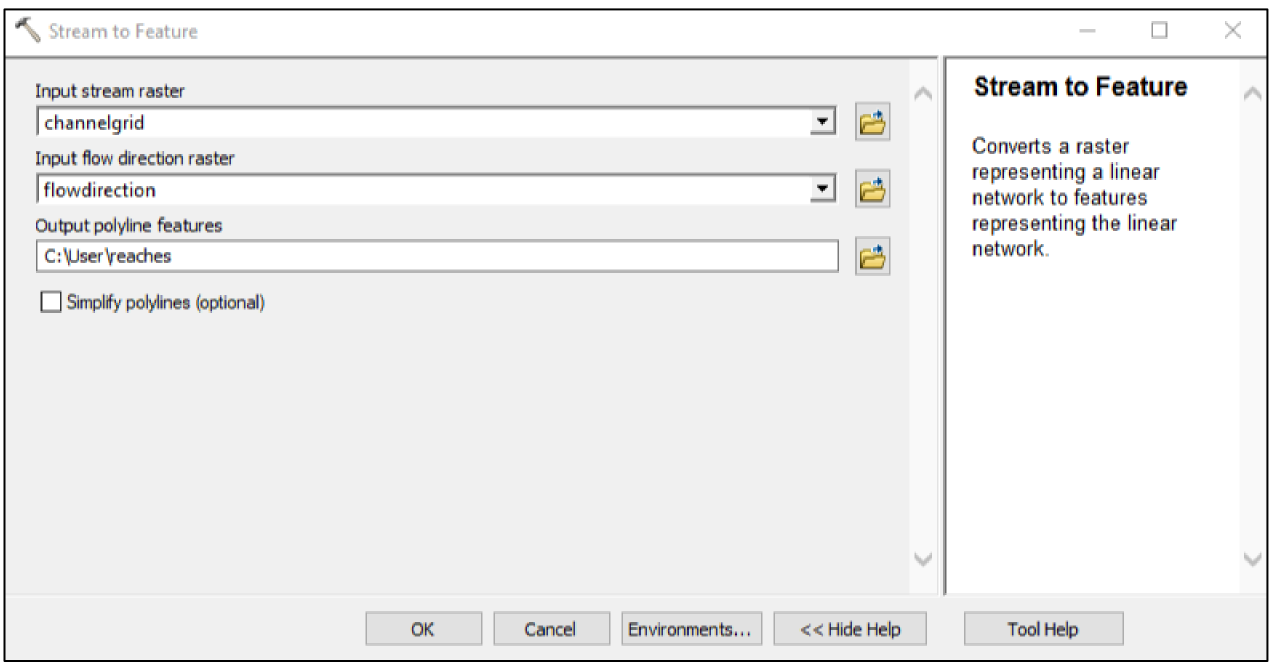
3. Flow Accumulation.



4. Raster Calculator. Extract the channel network using the command "*Con(flowacc >= Threshold, 1)*" where *flowacc* is your flow accumulation raster and *Threshold* is an integer (e.g., 1000). A smaller *Threshold* value yields a more complex channel network. The result is a raster where channel points are indicated by 1, with all other areas NoData.



5. Stream to feature. Yields a polyline shapefile.



6. Once these steps are completed, continue with the steps in Section 3B.

4. Run the CNES simulator

NOTE: If running in Windows/PC instead of Linux/Mac, replace any following instances of $SWOT_HYDROLOGY_TOOLBOX with %SWOT_HYDROLOGY_TOOLBOX%. Additionally, paths will need to use a back-slash (\) instead of a forward-slash (/).

A. Test the simulator installation

To test that everything is set up correctly, we will run the river_and_lake test case first.

```
>>cd $SWOT_HYDROLOGY_TOOLBOX/test/river_and_lake
>>python $SWOT_HYDROLOGY_TOOLBOX/select_orbit_cnes/select_orbit_cnes.py
    rdf/parameter_orbit.rdf output/orbit
>>python $SWOT_HYDROLOGY_TOOLBOX/sisimp/proc_sisimp.py
    rdf/parameter_sisimp.rdf
```

If these commands runs successfully and output is shown under the output/orbit and output/simu directories, the simulator has been set up correctly. See section 4.B to run a case with your own input.

B. Set up the working directory for a case using your own input data

With "SWOT_HYDROLOGY_TOOLBOX" and "RIVEROBS" set in Section 2 as environment variables, on the command line issue the following commands (Do not include ">>". This merely indicates a new command line):

```
>>cd $SWOT_HYDROLOGY_TOOLBOX/test/
>>mkdir testcase
>>cd testcase
>>mkdir data output rdf
>>cd output
>>mkdir orbit simu
>>cd ../rdf
>>cp ../../river_and_lake/rdf/*.rdf ./
>>cd $SWOT_HYDROLOGY_TOOLBOX/test/testcase
```

Move/save your input polygon shapefile(s) created in Section 3 to *$SWOT_HYDROLOGY_TOOLBOX/test/testcase/data/*. *$SWOT_HYDROLOGY_TOOLBOX/test/testcase* is your working directory, where *testcase* can be interchanged with a name describing your study area (e.g., Amazon, Mississippi, Scandinavia, etc.)

B. Set up your .rdf files

Navigate to your rdf directory:

```
>>cd $SWOT_HYDROLOGY_TOOLBOX/test/testcase/rdf/
```

1. Open parameter_orbit.rdf in your favorite Linux text editor (e.g., vi, gedit, nano, etc.). This file sets parameters describing the SWOT orbit which will be used to determine the

observation times for your study area. Specify your mission start time (used to determine SWOT overpass times), your study area bounding box (determined by your river network extent), and simulation start and end times. You may modify the other parameters as needed, although the defaults will likely work for most applications. Save and exit.

```
!== Mission specific parameters ==
Mission name = SWOT
Mission start time = 2015-02-01


!== Orbit parameters ==

!Directory of theoretical orbit files
Orbit repository = $SWOT_HYDROLOGY_TOOLBOX/select_orbit_cnes/swot_science_orbit_june_2015

!Studied area bounding box
DEM south latitude (deg) = 62.8
DEM north latitude (deg) = 65.5
DEM west longitude (deg) = -149.8
DEM east longitude (deg) = -142.0

!PixC mapping parameters
Azimuth spacing (m) = 21.875000
Swath width (m) = 120000.000000
NR cross track (m) = 10000.000


!== Pass plan parameters ==
passplan = yes
simulation_start_time = 2015-06-01
simulation_stop_time = 2015-07-06


!== Output parameters ==
GDEM Orbit prefix = test.gdem_orbit
```

2. Open parameter_sisimp.rdf in your favorite text editor. Specify the polygon shapefile path as *$SWOT_HYDROLOGY_TOOLBOX/test/testcase/data/* and the output directory as *$SWOT_HYDROLOGY_TOOLBOX/test/testcase/output/simu/*.
3. Specify the orbit parameters. The easiest option is shown below, with “Multiple orbit = no”. The Orbit” and “Cycle number” correspond to an entry in the orbit passplan file (*$SWOT_HYDROLOGY_TOOLBOX/test/testcase/output/orbit/passplan.txt) created in Section 4C.* If you want to process multiple shapefiles at once, see the Appendix for instructions.

```
!### Orbit parameters
!3 options =
!Multiple orbit = yes (default) => all orbit files in Orbit directory will be processed
!Multiple orbit = no => set the Orbit to a correct number found in the Orbit directory; only this orbit file will be processed
!Multiple orbit = passplan => orbit files will be processed according to passplan.txt file (generated if passplan = yes in select_orbit.rdf)
Multiple orbit = no
Orbit = 14
Cycle number = 1
```

4. Specify the height parameters. This tutorial describes the steps for using height model option 2. Ensure the “Height model” and “Height shp attribute name” parameters are not commented out and that the parameters for options 1 and 3 are commented out with a “!”. Make sure the height attribute field in your polygon shapefile matches the “Height shp attribute name” parameter.

```
!== Height ==

!### Option 1 - Constant height model, uniform over each water body; height varies sinusoidally with time
!Specific option: [Height model A = 0.] & [!Height model] => no height applied, height in output onlyt contains errors
!### Constant height model parameters (same height for each water body)
!Constant height model A = 0.   !=0 to disable
!Constant height model t0 = 47076
!Constant height model period (days) = 365.25
!### Complex 2D height model parameters (2D variations added for lakes > [Height model min area])
!Height model = polynomial    !=polynomial or gaussian; if disabled, only constant height model
!Height 2d model min area = 100.     !(ha) min area of water bodies on which to add complex 2D height model (default=100.)
!Height 2d model stdv = 1.        !stdv for gaussian model (ie Height model = gaussian)

!### Option 2 - Height is given from a specific attribute in the shapefile of water bodies
Height model = reference_height
Height shp attribute name = HEIGHT     !Name of the attribute (default=HEIGHT)
```

5. You may modify the other parameters in the parameter_sisimp.rdf as needed (e.g., error parameters), although the defaults will likely work for most applications. Save and exit.

C. Create the orbit file

Create the orbit file from your working directory with the command:

>>cd $SWOT_HYDROLOGY_TOOLBOX/test/testcase
>>python $SWOT_HYDROLOGY_TOOLBOX/select_orbit_cnes/select_orbit_cnes.py rdf/parameter_orbit.rdf output/orbit

A passplan.txt file (along with several other files) will be created in your *$SWOT_HYDROLOGY_TOOLBOX/test/testcase/output/orbit/* directory. Update your parameter_sisimp.rdf orbit parameters to match the passplan.txt orbit and cycle of interest.

D. Run the simulator

Now you are ready to run the simulator to create proxy SWOT WSE. Start the simulator from your working directory using the command:

```
>> cd $SWOT_HYDROLOGY_TOOLBOX/test/testcase
>>python $SWOT_HYDROLOGY_TOOLBOX/sisimp/proc_sisimp.py
rdf/parameter_sisimp.rdf
```

Output, including pixel cloud shapefiles, will be created in the *$SWOT_HYDROLOGY_TOOLBOX/test/testcase/output/simu/* directory. Further post-processing can be done using the parameter_river.rdf and the RiverObs simulator needed.

Appendix – Creating a time series (Advanced User)

A time series of input polygon shapefiles can be created from model output, but observations and best estimates may also be used. First, create the orbit file (Section 4C) to obtain observations times from the passplan.txt file (*$SWOT_HYDROLOGY_TOOLBOX/test/testcase/output/orbit/passplan.txt*). For each observation time in the passplan.txt file, create a separate polygon shapefile with the appropriate HEIGHT values. This can be done manually or using the Field Calculator in ArcGIS, or if you would like to do this programmatically using Python see the Pyshp manual (https://pypi.org/project/pyshp/) for more information.

Once a time series of input polygon shapefiles is created (one shapefile per time), a simple python wrapper (https://github.com/njelmer/swot-hydrology-toolbox/blob/master/scripts/process_multiple_shp_CNES.py) is available to process the full time series through the CNES simulator quickly and efficiently.