Using the CSDMS Modeling Tool

Notes for Hands-on

Run and Couple Surface Dynamics Models

Step 1 Activate your Secure Connection

You will need to have a VPN (Virtual Private Network) to have a secure encrypted tunnel from your computer to the University of Colorado's network whether you are on campus or off. Please have the VPN or Network Connect Software installed on your own system. Find this software at the University of Colorado site:
http://www.colorado.edu/cns/vpn/
One of the optional software packages needs to be installed on your computer. To then log in to the CU network, you will need to know your CU user name, and your IdentiKey.

Step 2 Download the latest CMT tool

Download the CMT tool from the CSDMS website.
http://csdms.colorado.edu

Step 3 Log In to Beach

You will be requested to log in to the CSDMS High Performance Computing Cluster (the HPCC is called Beach). You requested account for this and received log-in information from us. For this step you will need again your CU user name, and in addition your Beach password.

Step 4 Go to the CMT Workspace and Open HydroTrend Project

Now you are in the CSDMS Modeling Tool. It will automatically start up in the CMT Help, for the exercises we will use the CMT Workspace.

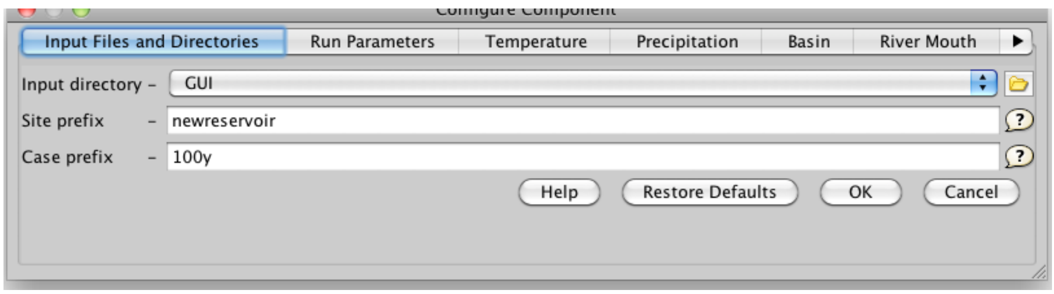
- Under the File Menu, choose 'open project'.
- Go to the Coastal Group. Open Project: Hydrotrend + Avulsion +CEM.
- Drag in HydroTrend Component from the Component Palette to the Driver Space.

Now you are ready to set up simulation!

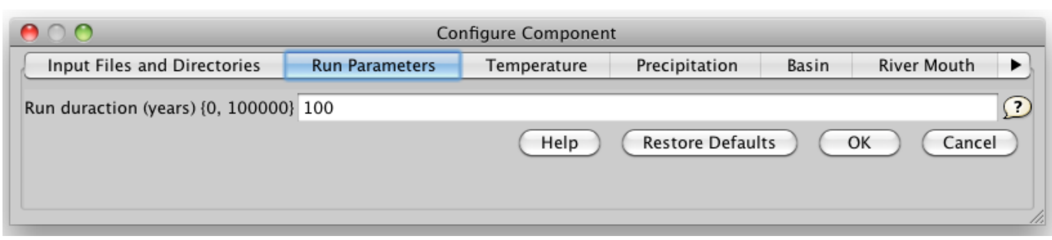
HYDROTREND SIMULATIONS

We are using a small theoretical river basin of ~1990 km^2, with 1200m of relief and a river length of ~100 km. The basin geometry parameters we will not be able to change in this hands-on. All parameters that are shown by default in the CMT for the HydroTrend Configuration are based on a present-day, temperate climate.

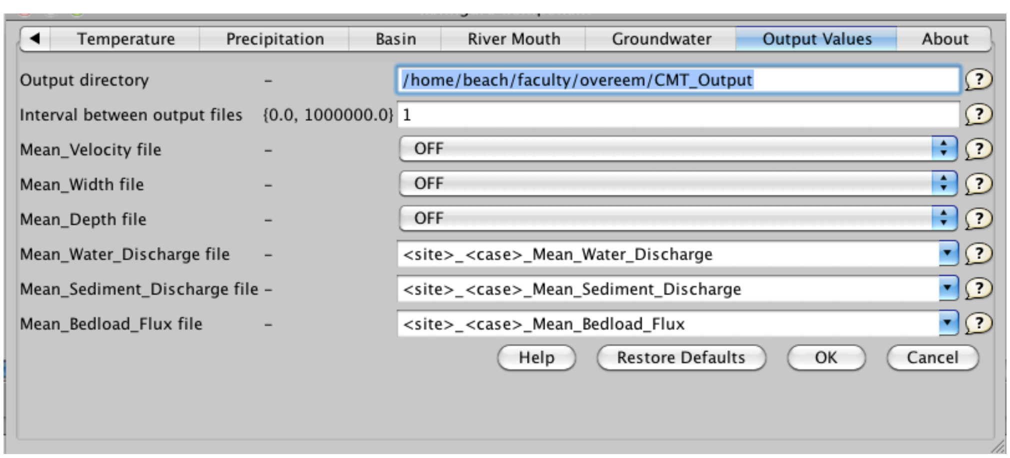
>>Make sure you have the first tab of the 'Configure Menu" set to GUI. This means that the input to the model is being received from the GUI. Here you can specify your simulation name (the combination of the site and case prefixes).



>> Set the run to 100 years duration in the second tab.

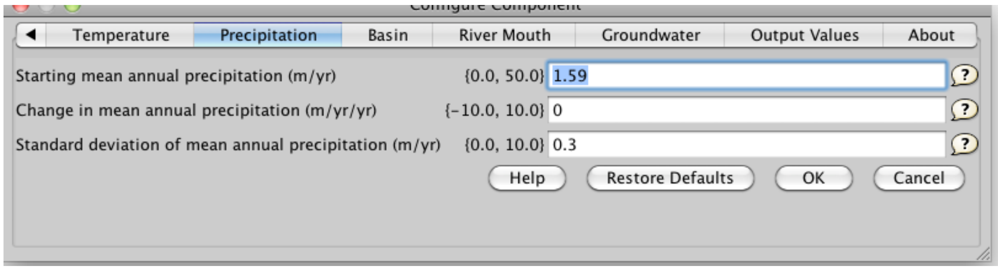


>> We will leave most process parameters as default values.
>> You can specify what output files will be generated, and at what time interval (1 day).



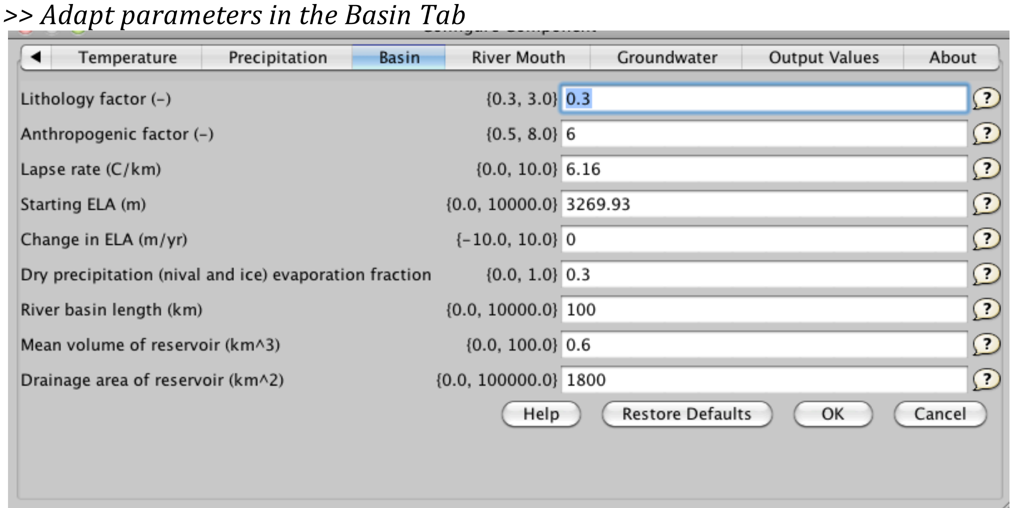
Experiment 1: Climate Change

>> Scenario: 100 % increase of precipitation over the next century.
>> Adapt parameters in the Precipitation Tab



Experiment 2: Model the Impact of a Reservoir

>> Scenario: a new reservoir is planned, 1km long, 10 m deep and 200m wide. It wil be in the coastal plain, and 1800km2 of upstream drainage area drains towards this reservoir.
>> Adapt parameters in the Basin Tab



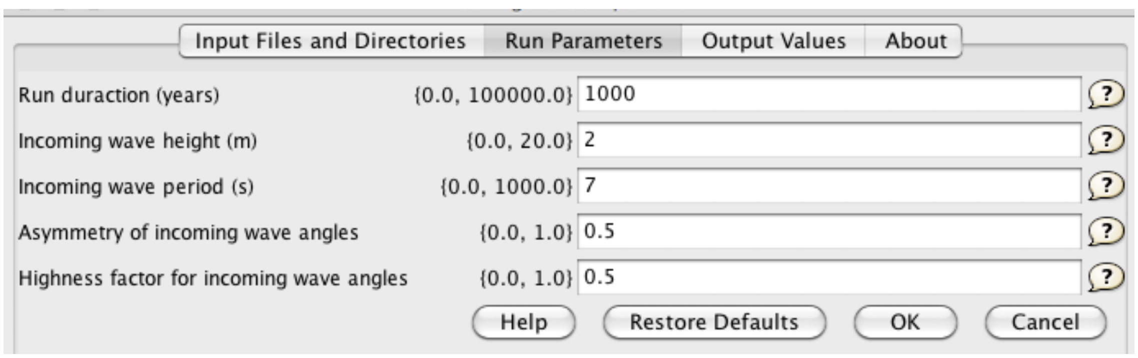
• *Run your simulations, look at your results in the CMT Console.*
Q-bar = mean annual water discharge, Qs-bar = mean annual sediment load.

COUPLED SIMULATIONS: Avulsion-CEM-WAVES

- Now drag in CEM to be the driver component.
- Drag the Avulsion Component into the Arena, you will see the connection between the River provider port and the River Use Port light up.
- Drag in the WAVES component
- Drag in the ConstantScalar component. This last component is a generic service component that can provide any scalar to another component, and it needs to be manually connected to the Discharge Port.

Experiment 1: Wave Angle Scenarios

- *Run experiments for 1000 timesteps, one fixed river, constant sediment discharge (250 kg/s).*
- *Make changes to the incoming wave field by adapting the Asymmetry and Highness factors in the configure Menu of WAVES component.*
- *Three possible combinations: U=0.75. A=0.5, U= 0.5, A=0.75, U = 0.75, A=0.75*



Experiment 2: Multiple Rivers

- *Run experiments for 1000 time steps, run with two fixed distributary channels, constant sediment discharge (250 kg/s). One scenario can be for widely switching channels.*
- *Make changes to the distributary channels by adapting the number of rivers in the configure Menu of Avulsion components.*