CHILD Users Guide

Version 2.0

Gregory E. Tucker[1]

Department of Civil and Environmental Engineering
Massachusetts Institute of Technology
Cambridge, MA 02139

Part IV-A of final technical report submitted to U.S. Army Corps of
Engineers Construction Engineering Research Laboratory (USACERL)
by Gregory E. Tucker, Nicole M. Gasparini, Rafael L. Bras, and Stephen T. Lancaster
in fulfillment of contract number DACA88-95-C-0017

April, 1999

1. To whom correspondence should be addressed: Dept. of Civil & Environmental Engineering, MIT Room 48-429, Cambridge, MA 02139, ph. (617) 252-1607, fax (617) 253-7475, email gtucker@mit.edu

CHILD Users Guide

Version 2.0

The CHILD model is a C++ program that simulates erosion, sedimentation, and landscape evolution, over either geologic or "human" time scales. CHILD was designed and written in 1996 and 1997 by Greg Tucker, Stephen Lancaster, Nicole Gasparini, and Rafael Bras, in the Department of Civil and Environmental Engineering at the Massachusetts Institute of Technology. The present version, released in April 1999, represents a significant upgrade to the original (1997) version. This document describes how to install, compile, set up, and run CHILD Version 2.0. This guide assumes that you are familiar with what CHILD is and what it can do; for basic information about CHILD and its capabilities, consult the document "Overview of the CHILD Model" (available at http://platte.mit.edu/~child). Other articles and documents related to CHILD are listed under References, below.

Installing and Compiling CHILD

CHILD was developed in C++ on a UNIX platform. The code is designed to be as platform-independent as possible (e.g., no direct window system interface is included); however, the instructions here pertain specificially to UNIX installation. As of this writing, the CHILD code is provided in a single tar-format file called `child_src_2.0.tar`. To unpack it, place this file in the desired master directory and type

```
% tar xvf child_src_2.0.tar
```

(where % represents the UNIX command-line prompt).

This will create a series of subdirectories, each containing a source file and its accompanying header file. The easiest way to compile the program is to use the UNIX `make` utitility. At this writing, a make file is provided only for Digital UNIX systems running the `cxx` compiler. The make file can be modified for other platforms and other compilers, however. To compile the code with `cxx` under Digital UNIX, use

```
% make -f makefile_cxx
```

The make utility will create an executable file called `child`.

Creating an Input File and Starting a Run

To run a CHILD simulation, you will need to provide an input file (hereafter called the "main input file") that contains various required and optional parameters. The main input file can have any name; by convention, main input files have the extension `.in`. Unless a simulation mesh is to be created from scratch, several additional files containing information about the initial topography and/or mesh configuration will be needed. These additional files are described below in reference to the various mesh input and/or generation options.

The command-line syntax for running the model is

```
% child myinputfile.in [-s]
```

where `myinputfile.in` represents the name of the main input file, the format of which is described below, and `-s` is an optional argument that, if given, suppreses printing of the current simulation time at each iteration.

The main input file contains a list of process coefficients, option switches, auxiliary file names, and other parameters. Some of these parameters are always required. Others are optional, and only need to be included when a particular option is selected. Any optional parameters included in the file but not needed in the run are simply ignored. The format for each parameter consists of a line of descriptive text followed by the value of the parameter itself on a second line. Here is an example showing the format of the main input file:

```
OPTREADINPUT: 0=new mesh, 1=existing mesh, 2=points, 3=DEM
1
INPUTDATAFILE:
mytin
INPUTTIME: which time to read from inputdatafile
0
RUNTIME: Duration of run (years)
100
OPINTRVL: Output interval (years)
5
OPTVAR: Option for random rainfall variation
1
PMEAN: Mean rainfall intensity (m/yr)
12
STDUR: Mean storm duration (yr)
0.07763
#0.003
ISTDUR: Mean time interval between storms (yr)
0.15753
#0.997
OPTDETACHLIM: 1 = use purely detachment-limited erosion
0
...etc.
```

At the start of each header line is an “item code” in capital letters. This correct code for each parameter must be included in order for the model to find and read the parameter (Table 1 lists required input parameters and their corresponding item codes). A hash mark (#) represents a comment line (here, the values 0.003 and 0.997 are commented out). Parameters can be in any order in the file. In the example above, the first parameter OPTREADINPUT is a required parameter that tells the model whether to create a new (synthetic) mesh, read in an existing mesh, create a mesh from an input set of (x,y,z) points, or create a mesh from a DEM. In this example, the option for reading in an existing mesh has been selected, and two additional optional parameters have been included to support this choice: INPUTDATAFILE, representing the base name for mesh input files, and INPUTTIME, representing the time-code for which input is read (this will normally be nonzero only when restarting a previous run). (Note that for the sake of brevity the example above does not include all required parameters). The sections below describe how to set up a CHILD run with different mesh input/generation options and with different process parameters and options.

Setting Up the Initial Mesh

CHILD provides several options for generating and/or reading in an initial mesh. You can have the model set up a mesh in any of the following ways

- Generate a new "synthetic" rectangular mesh with a hexagonal, perturbed-hexagonal, or random arrangement of nodes
- Read in an existing TIN (including a TIN output by another CHILD run)
- Generate a mesh from a given set of (x,y,z) points
- Generate a mesh from a DEM in Arc/Info ascii format

The following sections describe how to use these various options.

Creating a Mesh from Scratch

If desired, CHILD can create a starting mesh with a rectangular domain. Three examples of CHILD-generated starting meshes are shown in Figure 1. Points within the rectangular domain can be regularly spaced in a hexagonal lattice (Figure 1a), regularly spaced with random offsets in their (x,y) positions (Figure 1b), or randomly placed within the rectangular domain (Figure 1c). In the case of a random-offset mesh, the maximum offset amount is equal to + or - 25% of the nominal point spacing. In all of these cases, there are five possible boundary options (Figure 2): (1) a single outlet (open boundary point) in one corner; (2) an open boundary along one side (corresponding to y=0); (3) open boundaries along the y=0 and y=ymax sides; (4) open boundaries along all four sides; and (5) a single outlet point at a specified location. (Recall that an open boundary point is a boundary node at which water and sediment are allowed to exit). In each case except #3, the initial elevation is equal to a specified mean elevation plus or minus a random variable (uniformly distributed), with the minimum and maximum possible values equal to 50% and 150% of the mean elevation, respectively. In case #3 (open boundaries on two opposite sides), the elevation of the upper (y=ymax) boundary is specified (UPPER_BOUND_Z), and the elevations of the points are set so as to create a uniform gradient between the two boundaries, plus random vertical offsets.

To start a run with a new rectangular mesh, set OPTREADINPUT to 0. The parameter OPT_PT_PLACE controls the type of point placement (0, 1, and 2 for regular, offset, and random, respectively), and the parameter TYP_BOUND controls the open boundary configuration (0, 1, 2, 3, and 4 for corner, one side, two sides, four sides, and user-specified, respectively). Additional parameters required with these different options are discussed below.

Setting Up the Configuration of a Synthetic Mesh

The dimensions of the rectangular mesh are set using the parameters X_GRID_SIZE and Y_GRID_SIZE (in meters). Set the option for point placement, OPT_PT_PLACE, to 0, 1 or 2 for regular, perturbed, or random points, respectively. In the case of regular or perturbed-regular point placement, the mean point spacing (GRID_SPACING) must be specified. In the case of random point spacing, the total number of interior points, NUM_PTS, must be given. In any event, the mean elevation (MEAN_ELEV) and a seed for the random number generator (SEED) must always be provided when a new rectangular mesh is generated.

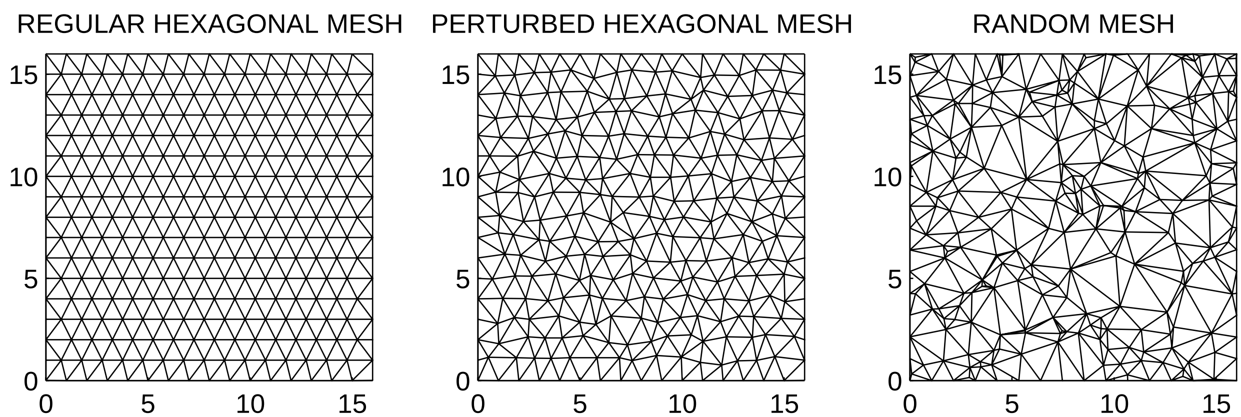

FIGURE 1. Meshes generated by CHILD, illustrating the different point-placement options.

Boundary Configuration Options

Set the option TYP_BOUND to specify the boundary type. For a two-sided boundary, you must also specify the elevation of the upper boundary (which can be different from zero) using the parameter UPPER_BOUND_Z. For a user-specified outlet location, give the outlet coordinates in OUTLET_X_COORD and OUTLET_Y_COORD (note that an outlet cannot be placed in the interior of the mesh).

Reading in an Existing Mesh

You can start a CHILD run using an existing triangulation. The triangulation might represent either a mesh from a previous CHILD run or a TIN output from a GIS system (for example). The input mesh data format is the same as CHILD's output format (see "Output Files" on

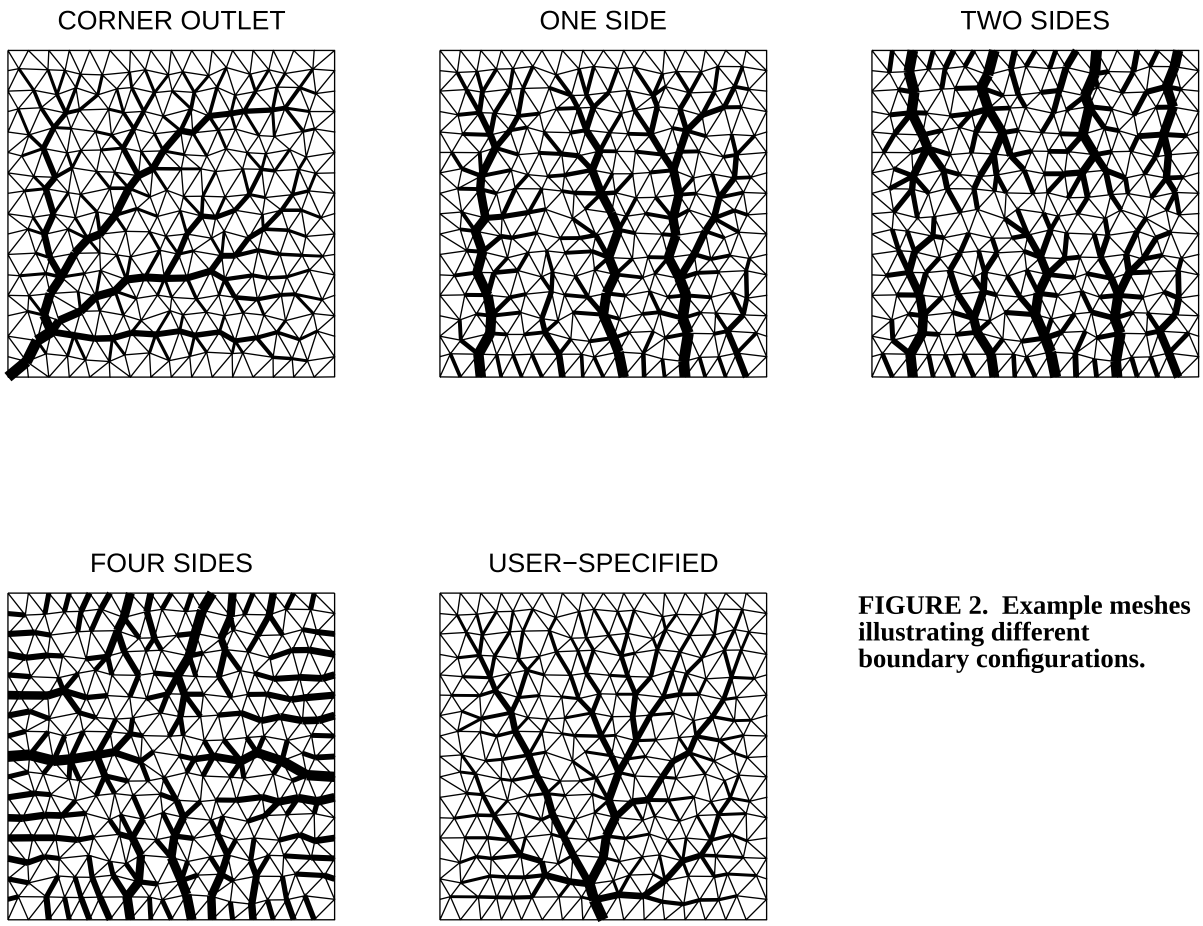

FIGURE 2. Example meshes illustrating different boundary configurations.

page 23), and consists of four ascii files that contain data on nodes, edges, triangles, and node elevations, respectively. The files must bear the same base name and have the extensions `.nodes`, `.edges`, `.tri`, and `.z` (as in `foo.nodes, foo.edges, foo.tri, foo.z`). Each file may contain triangulations for multiple time intervals (obviously in the case of GIS data this does not apply; only one "time slice" would be included in the file).

The format of the node file consists of the time represented by the triangulation on the first line, the number of nodes (including boundaries) on the second, and on the subsequent lines the following information for each node: x-coordinate (meters), y-coordinate (meters), ID of one of its connected edges, and a boundary code. (If inputting a mesh that isn't the output of a previous run, use 0 for the time code). For example:

```
 0
289
0.50627 1.134 1424 0
1.02798 1.80433 2 0
0.96638 2.79677 4 0
...etc
```

The nodes are assumed to be in order by ID number, starting with 0 and ending in N-1 (where N is the number of nodes). The boundary codes are 0 for an interior point, 1 for a closed boundary, and 2 for an open boundary. Note that points flagged as interior points cannot be on the perimeter of the mesh; an error will occur if any such points exist.

The format of the other three files is similar in that each time-slice begins with the time value and the number of elements (either edges, nodes, or triangles). In the edge file, each row must contain the ID of the origin node, the ID of the destination node, and the ID of the edge that lies counter-clockwise (relative to the origin) (see Tucker et al., 1999, for discussion of these data structures). Complementary edge pairs MUST BE TOGETHER on the list. An example of an edge file is shown below:

```
 0
1600
225 0 1417
0 225 34
1 0 30
0 1 1424
...etc.
```

Note that directed edges that share the same endpoints (complementary edges) are always grouped together. As with nodes, the edges should be listed in order by ID number.

Each row of the triangle file contains the ID numbers of the three vertex nodes in counter-clockwise order, the ID number of the three triangles opposite these vertex nodes (or -1 if no triangle lies opposite a given face), and the ID numbers of the three clockwise-oriented directed edges. These edges are listed in the same order as their origin vertices, i.e., edges 0->2, 1->0, and 2->1. Triangles are also listed in order by ID.

The node elevations file has the same header format as the others (time followed by number of nodes). The header for each time step is followed by a series of lines that contain the elevation in meters of each node, in order by ID number.

Note: CHILD includes a consistency checking routine to make sure the mesh format is valid, but this is not guaranteed to be foolproof. In particular, the model also does not test whether the input triangulation is Delaunay.

Creating a Mesh from a Set of Points

CHILD can construct a mesh from an input set of (x,y,z) points. Such points might be obtained from a field survey, a GIS point coverage, a sampled DEM, etc. To create a mesh from a set of points, set the OPTREADINPUT option to 2. You will then need to specify the name of an ascii file (parameter POINTFILENAME) containing the point data. The format for the point data file consists of a header line that gives the number of points, followed by a series of rows that contain the x and y coordinates, the elevation, and a boundary code for each point, as in the following example:

```
7
```

```
1 3 30 1
3 3 30 1
0 1.5 10 1
2 1.5 10 0
4 1.5 10 1
1 0 0 2
3 0 0 2
```

The point set in this example is used to construct the seven-node mesh shown in Figure 3. The possible boundary codes are 1 (closed boundary), 2 (open boundary), and 0 (mesh interior). Nodes flagged with boundary code zero must always be within the mesh interior; errors will result if the model encounters an active (0 boundary code) node on the mesh perimeter. In the above example, only one node lies in the mesh interior.

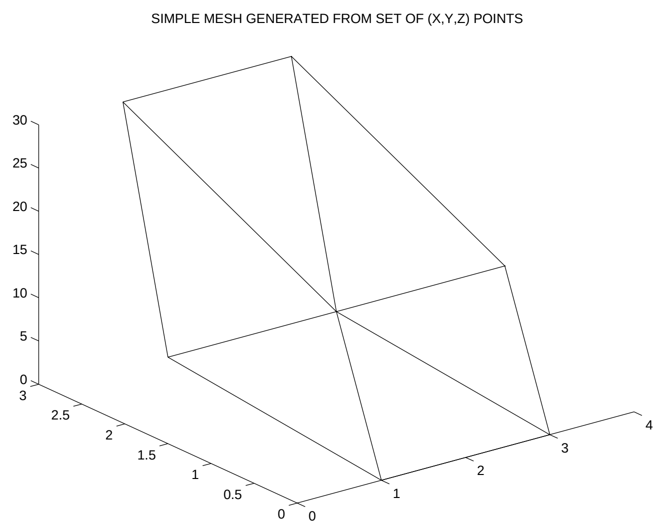

FIGURE 3. A seven-node mesh constructed from a set of input points.

Generating a Mesh from an Arc/Info Grid DEM

CHILD can also generate a mesh by interpolating a DEM. The DEM must be in Arc/Info's ascii grid format. In the present version, this capability is somewhat limited. The DEM must describe a single drainage basin, since only one outlet point is created. The resolution of the mesh will be the same as that of the original DEM, and will be uniform in space. The mesh can be either hexagonal, or “random.” In the latter case, the mesh starts with one mesh point per DEM point; the x and y coordinates of each point are then offset by a random amount. When a mesh point falls between DEM points, the four-point interpolation procedure of Tetzlaff and Harbaugh (1988) is used to derive the point’s elevation. To create a mesh with random offsets from a DEM, set OPTREADINPUT to 3; to create a hexagonal mesh from the DEM, set OPTREADINPUT to 4. Use the parameter ARCGRIDFILENAME to specify the name of the Arc/Info grid ascii file.

Setting Up Parameters and Options

This section describes how to set up the parameters in a CHILD run and how to activate and deactivate different options for landscape processes, hydrologic behavior, climatic variation, and so on.

Run Name, Duration, and Output

Use the OUTFILENAME parameter to specify a base name for the output files. Each output file will have this name followed by an extension that describes the contents of the file (see “Output Files” on page 23). The RUNTIME parameter sets the total duration of the run in years, and the parameter OPINTRVL specifies how often output should be written (also in years).

Climate Parameters

The parameters PMEAN, STDUR, and ISTDUR set the mean storm intensity, duration, and interstorm interval, respectively (in units of meters and years). To generate a sequence of random storms based on these parameters, set OPTVAR to 1; otherwise, each storm will have the mean values. The random number generator is initialized by the SEED parameter. Note that you can configure a run to operate as a continuous (constant-climate) model by simply turning off climate variation and, if desired, setting the interstorm interval to zero, in which case STDUR effectively becomes the time-step. Note also that for long simulations, it is usually convenient to accelerate performance by magnifying the mean storm and interstorm intervals. This preserves the distribution of event sizes while enhancing performance. However, use this approach with caution, since some parameters (such as soil storage capacity) depend on storm duration and would therefore need to be adjusted accordingly.

To invoke sinusoidal fluctuations in mean storm intensity, duration, and/or spacing, set OPTSINVAR to 1. You will then need to specify the period of variation in years (PERIOD) along with the maximum values of mean intensity (MAXPMEAN), duration (MAXSTDURNM), and spacing (MAXISTDURMN).

Runoff Generation and Flow Routing

The FLOWGEN parameter controls the model's runoff generation options. Possible values are:

0. Uniform Hortonian (infiltration-excess) runoff
1. Saturation-excess runoff using the O'Loughlin (1986) method
2. Saturation-excess runoff using the modified Beven and Kirkby (1979) method
3. Uniform soil-storage capacity ("bucket model")

The Hortonian runoff option is the simplest, and requires two additional parameters: the soil infiltration capacity (INFILTRATION) in meters per year, and an option for sinusoidal time-variation in infiltration capacity (OPTSINVARINFILT). The uniform storage capacity option also requires the INFILTRATION and OPTSINVARINFILT parameters, along with the parameter SOIL-

STORE, which represents the soil water-storage capacity in meters. The O'Loughlin saturation-excess model requires only a single parameter, the soil transmissivity (TRANSMISSIVITY) in square meters per year. The modified Beven-Kirkby saturation-excess model requires three parameters: the infiltration capacity (INFILTRATION), the infiltration variation option (OPTSINVARINFILT), and a unit soil deficit parameter (TRANSMISSIVITY). (Note that the Beven-Kirkby option uses the TRANSMISSIVITY parameter not as a transmissivity *sensu strictu*, but rather as a moisture deficit scaling parameter with units of meters.)

Sinusoidal Variation in Infiltration Capacity

Set the OPSINVARINFILT parameter to 1 to have infiltration capacity vary sinusoidally through time (this provides a way to model the hydrologic effect of changes in surface properties such as vegetation cover). Additional parameters required with this option include the period of variation (PERIOD_INFILT) and the maximum infiltration capacity (MAXICMEAN).

Lakes and Sinks

The CHILD model includes an algorithm to identify and route drainage through depressions in the terrain. To invoke this option set LAKEFILL to 1. Otherwise, water will be assumed to evaporate within sink points. It is usually a good idea to run the model with this option activated.

Adding an Inlet

An "inlet" is a point source of water and sediment that represents a through-flowing stream. The inlet option makes it possible to configure simulations that represent a segment of a valley or floodplain, or (as a larger-scale example) a large antecedent river crossing mountainous terrain. To activate an inlet, set OPTINLET to 1. The inlet location must then be specified with INLET_X and INLET_Y (with the coordinates given in meters). INDRAREA sets the drainage area at the inlet point (in m^2). The volumetric sediment load of each grain size is specified in the parameters INSEDLOAD1, INSEDLOAD2, etc. (one for each grain size fraction).

Hydraulic Geometry

CHILD calculates hydraulic geometry (width, depth, bed roughness, and bank roughness) using the empirical methods of Leopold and Maddock (1953). Parameters relating to hydraulic geometry must always be provided in the input file; their use in the model depends on which process options are used. For simulations that do not use the detachment-limited option, channel depth is computed and used as the effective vertical depth over which sediment is “seen” by flowing water (this will have an influence on erosion and transport only when the material is vertically heterogeneous). The stream meander module makes use of all of the hydraulic geometry parameters, and the floodplain deposition module uses bankfull depth and flood depth to compute water surface elevations. Bankfull parameters (width, depth, and roughness) are scaled to a bankfull event size (expressed as an bankfull-event-producing runoff rate in meters per year) which is specified in the parameter BANKFULLEVENT. Additional hydraulic geometry parameters are:

HYDR_WID_COEFF_DS: coefficient in the width versus bankfull discharge relation

HYDR_WID_EXP_DS: exponent in the width versus bankfull discharge relation

HYDR_WID_EXP_STN: exponent in the at-a-station width versus discharge relation

HYDR_DEP_COEFF_DS: coefficient in the depth versus bankfull discharge relation

HYDR_DEP_EXP_DS: exponent in the depth versus bankfull discharge relation

HYDR_DEP_EXP_STN: exponent in the at-a-station depth versus discharge relation

HYDR_ROUGH_COEFF_DS: coefficient in the roughness vs. bankfull discharge relation

HYDR_ROUGH_EXP_DS: exponent in the roughness versus bankfull discharge relation

HYDR_ROUGH_EXP_STN: exponent in the at-a-station roughness vs. discharge relation

BANK_ROUGH_COEFF: coefficient in the bank roughness vs. bankfull disch. relation

BANK_ROUGH_EXP: exponent in the bank roughness versus bankfull discharge relation

Note that the calculated channel width and depth are not used directly in the erosion and sediment transport equations, for the sake of generality. These relationships are, however, typically embedded in the terms of those equations, and care must be taken to ensure consistency between the erosion parameters and the hydraulic geometry parameters.

Erosion Parameters

Detachment and Sediment Transport

There are six parameters that govern particle detachment by running water: KR (the detachment coefficient for regolith), KB (the coefficient for bedrock), MB (the discharge exponent), NB (slope exponent), TAUCD (detachment threshold), and PB (excess stream power or shear stress exponent). Note that, if appropriate, these can be reduced to three effective parameters by setting KR=KB, TAUCD=0, and PB=1 (see for example Whipple and Tucker, 1999).

It is often advantageous to assume a detachment-limited condition, in which sediment transport capacity is not a limiting factor for surface erosion. Detachment-limited erosion will occur naturally under conditions of high transport capacity and/or low detachment capacity, but the overhead of computing transport rates at every point can involve a considerable computational burden that is often unnecessary. For such cases, the model provides an option for purely detachment-limited erosion, in which transport capacities are not calculated. To invoke this option, set

OPTDETACHLIM to 1.

Sediment transport can be modeled using either a simple power law or using the sand-gravel bedload transport equations of Wilcock (1998). For performance reasons, the choice between these two options must be made at compile-time rather at than run-time. The sediment transport option is specified in the header file `erosion.h`. To change it, change the defined constant `tSedTrans` as follows:

```
#define tSedTrans tSedTransPwrLaw
```

or

```
#define tSedTrans tSedTransWilcock
```

for the power-law and Wilcock (1998) models, respectively.

For the power-law sediment transport model, the required parameters are KF (transport coefficient), MF and NF (discharge and slope exponents, respectively). For the Wilcock sand-gravel equations, the mean diameter of the sand (GRAINDIAM1) and gravel (GRAINDIAM2) fractions must be given. (Note that the appropriate sediment transport parameters must be included in the input file even when running in detachment-only mode).

Diffusive Hillslope Transport

Diffusive sediment transport involves two parameters. The first of these is the diffusivity constant (KD) in m^2/yr. The second, OPTDIFFDEP, is an option that controls whether diffusive transport contributes to both erosion and deposition, or to erosion only. In some cases, there may be reason to apply diffusion only to convex (erosional) elements of topography, with any deposited sediment assumed to be removed rapidly by water erosion (see, e.g., Moglen and Bras, 1994). Set OPTDIFFDEP to 1 to deactive the deposition component hillslope diffusion.

Sediment Size and Sorting

Use the NUMGRNSIZE parameter to specify the number of grain size fractions, but beware: only the single-size (power-law) and dual-size (Wilcock) sediment transport models are currently supported, so that NUMGRNSIZE must be either 1 or 2. (However, the model data structures are configured to handle any number of grain size classes, so the addition of transport formulae using more than two size-fractions would be straightforward). For each size fraction, you will need to specify the median diameter in meters (GRAINDIAMn, where n is the size class), the volumetric proportion of that size class in the regolith substrate (REGPROPORTIONn; see below), and the volumetric proportion of that size class that would be generated from erosion into bedrock (BRPROPORTIONn). You will also need to specify the depth of the active layer within which sediment mixing and selective transport occur (MAXREGDEPTH).

Setting Up Initial Stratigraphy

In general, each model node may include any number of layers of variable thickness and sediment composition. By default, the model initializes with a simple, spatially uniform stratigraphy that consists of a column of bedrock overlain by a column of regolith. The initial regolith thickness may be zero. The total column thickness must be deeper than the maximum potential erosion depth. The initial thickness of the bedrock and regolith columns are set by the parameters BEDROCKDEPTH and REGINIT, respectively, and their erodibility factors are KB and KR. (See "Erosion Parameters" on page 15.) Alternatively, the model can be initialized with a variable sequence of layers of different thickness, composition, and erodibility at each node. This is the method used to restart a run in which stratigraphy matters, and it simply involves setting the OPTREADLAYER parameter to 1. Layer data are then read from a file having the name specified in INPUTDATAFILE followed by the extension `.lay.` (This file must have the format as the layer output file; typically it is a layer output file from a previous run. The file format is summarized in Table 2).

Configuring Uplift

"Uplift" here refers to any vertical motion relative to the model boundaries, whether positive or negative. Two uplift functions are provided: (1) spatially uniform uplift at a specified rate and duration, and (2) uniform uplift on one side of a fault that runs parallel to the model's x-axis at a specified y-position. The parameter UPTYPE specifies the type of uplift to be applied, with 0 meaning no uplift, 1 meaning uniform uplift, and 2 meaning fault-bounded uplift. Uplift options 1 and 2 require you to specify the uplift duration in years (UPDUR) and the rate in m/yr (UPRATE). For fault-bounded uplift, an additional parameter specifying the y-coordinate of the fault in meters (FAULTPOS) must be given.

Stream Meandering

Stream meandering is activated by setting the parameter OPTMNDR to 1. Meandering will only be applied to nodes that have a discharge value greater than CRITICAL_FLOW (in m^3/yr). Typically, this threshold would be used in conjunction with an inlet to set up a primary meandering stream (see "Adding an Inlet" on page 13). Additional parameters that need to be included in order to simulate meandering are:

DEF_CHAN_DISCR: default channel discretization for meandering channels (widths)

FRAC_WID_MOVE: the maximum fraction of channel width a node is allowed to move in a step

FRAC_WID_ADD: distance a channel point moves before a new node is added in its wake, as a fraction of channel width

VEG_ERODY: erodibility of vegetated banks

LATADJUST: ratio of bank to bed erodibility

CHNGPROB: (not currently used)

BNKHTDEP: degree to which bank erodibility depends on bank height; 0=none, 1=all

To invoke layer interpolation when points are moved, set the parameter OPTINTERPLAYER to 1 (in the current version this must always be set to 1 to avoid errors). Further information on the meander model is given by Lancaster (1998).

Overbank Deposition

The overbank deposition module is invoked with the OPTFLOODPLAIN parameter. Parameters used by the overbank deposition module include the deposition rate constant (FP_MU, m/yr), the distance decay constant (FP_LAMBDA, meters), the minimum drainage area for a node to be considered a flood-initiation point (FP_DRAREAMIN, m^2), and the magnitude of a runoff event large enough to generate an overbank flood (FP_BANKFULLEVENT, m/yr).

Eolian Deposition

Set the OPTLOESSDEP parameter to invoke the eolian deposition module, and specify the deposition rate in m/yr with the LOESS_DEP_RATE parameter.

Tracking Exposure Ages

To keep track of the effective surface exposure ages of individual layers, set OPTEXPO-

SURETIME to 1.

Table 1: List of required model parameters

Name	Symbol	Units	Description	Remarks
OPTREADINPUT	-		*Option for initial mesh input or generation*	*Five possible options*
NUMGRNSIZE	-		*Number of grain size classes used in run*	*1 or 2*
MAXREGDEPTH	-	*m*	*Depth of active layer, and maximum depth of a deposited layer*	
GRAINDIAMi	D_i	*m*	*Mean diameter of grain size class i*	*One for each size class; must be at least one*
OPTREADLAYER	-		*Option for reading layers from input file*	*0 or 1; set to 0 when generating new mesh*
REGPROPORTIONi	-	*ND*	*Volumetric proportion of size i in regolith*	*One for each size class; range zero to one*
BRPROPORTIONi	-	*ND*	*Volumetric proportion of size i generated from eroded bedrock*	*“*
REGINIT	-	*m*	*Starting thickness of regolith layer*	
BEDROCKDEPTH	-	*m*	*Starting thickness of bed-rock layer*	
KR	K_r	*vary*	*Erodibility coefficient for regolith*	*If layers are read in, values from layer file are used instead*
KB	K_b	*vary*	*Erodibility coefficient for bedrock*	*“*
SEED	-		*Seed for random number generation*	*Must be an inte-ger*
OPTINTERPLAYER	-		*Option for layer interpola-tion when points are moved or added*	*Must be 1 in cur-rent version*

Table 1: List of required model parameters

Name	Symbol	Units	Description	Remarks
OUTFILENAME	-		*Base name for output files*	
OPTVAR	-		*Option for random rainfall variation*	
PMEAN	P	*m/yr*	*Mean storm rainfall intensity*	
STDUR	T_r	*yr*	*Mean storm duration*	
ISTDUR	T_b	*yr*	*Mean time between storms*	
OPTSINVAR	-		*Option for sinusoidal variation in rainfall parameters*	
OPTINLET	-		*Option for an external water and sediment input*	
FLOWGEN	-		*Runoff generation option*	*Four possible settings (0,1,2, or 3)*
OPTMNDR	-		*Option for stream meandering*	
BANKFULLEVENT	-	*m/yr*	*Runoff rate associated with bankfull flood event*	*Used to compute hydraulic geometry*
HYDR_WID_COEFF_DS	k_{wb}	*vary*	*Coefficient in bankfull width-discharge relation*	
HYDR_WID_EXP_DS	m_{wb}	*ND*	*Exponent in bankfull width-discharge relation*	
HYDR_WID_EXP_ST	m_w	*ND*	*Exponent in at-a-station width-discharge relation*	
HYDR_DEP_COEFF_DS	k_{hb}	*vary*	*Coefficient in bankfull depth-discharge relation*	
HYDR_DEP_EXP_DS	m_{hb}	*ND*	*Exponent in bankfull depth-discharge relation*	
HYDR_DEP_EXP_ST	m_h	*ND*	*Exponent in at-a-station depth-discharge relation*	
HYDR_ROUGH_COEFF_DS	k_{rb}	*vary*	*Coefficient in bankfull roughness-discharge relation*	

Table 1: List of required model parameters

Name	Symbol	Units	Description	Remarks
HYDR_ROUGH_EXP_DS	m_{rb}	*ND*	*Exponent in bankfull roughness-discharge relation*	
HYDR_ROUGH_EXP_ST	m_r	*ND*	*Exponent in at-a-station roughness-discharge relation*	
BANK_ROUGH_COEFF	k_{br}	*vary*	*Coefficient in bank roughness-discharge relation*	
BANK_ROUGH_EXP	m_{br}	*ND*	*Exponent in bank roughness-discharge relation*	
MB	m_b	*ND*	*Discharge exponent in detachment capacity equation*	
NB	n_b	*ND*	*Slope exponent in detachment capacity equation*	
PB	p_b	*ND*	*Excess power/shear exponent in detachment capacity equation*	
TAUCD	τ_{cd}	*vary*	*Detachment threshold (e.g., critical shear stress)*	
KD	k_d	m^2/yr	*Hillslope diffusivity coefficient*	
UPTYPE	-		*Type of uplift to be applied*	*0 = no uplift*
RUNTIME	-	*yr*	*Duration of run*	
OPINTRVL	-	*yr*	*Frequency of output to files*	
OPTDETACHLIM	-		*Option for detachment-limited erosion only*	
OPTDIFFDEP	-		*Option to deactivate deposition by hillslope diffusion*	
OPTFLOODPLAIN	-		*Option for floodplain overbank deposition*	
OPTLOESSDEP	-		*Option for eolian (loess) deposition*	

Output Files

Data output from CHILD include elevation, slope, drainage area, surface flow directions, surface water discharge, surface sediment composition, layer data, and information on the mesh triangulation. Output is written to a series of ascii files at selected intervals beginning with the initial state of the run (see OPINTRVL, Table 1). Each file contains one type of information (e.g., node elevations) and includes output from the start to the end of the run. The format of each file varies somewhat according to the type of data it contains, but all output files share the same header format: each time-slice is preceded by the simulation time (this line always begins with a space for easy identification) and the number of data elements for the current time-slice (equal to the number of nodes, edges, or triangles, depending on the contents of the file). The data for each node (or edge or triangle, as the case may be) then follow, separated by carriage returns. The format is illustrated below with a hypothetical node elevation file containing two time slices for a mesh with only three nodes:

```
 0                 // time 0
3                  // 3 nodes
1.1                // elevation of node 0 at time 0
2.2                // elevation of node 1 at time 0
3.3                // elevation of node 2 at time 0
 100               // time 100
3                  // still 3 nodes
0.1                // elevation of node 0 at time 100
1.2                // etc.
2.3
```

Note that data elements are always listed in order by ID number (e.g., node ID), starting from zero. Note also that the number of elements can change between successive time-slices if dynamic remeshing is used (e.g., when points are moved, added, or deleted by the stream meandering routines), as can the order in which the elements appear (node ID's are reordered when nodes are sorted by network position, so node 1 at one time slice is not necessarily the same as node 1 at another). Each output file bears the name specified in the input file (OUTFILENAME; see Table 1) followed by a suffix that indicates something about what the file contains. Output files and their suffixes are listed in Table 2, below. (Further description of the format used by the triangulation files is given in "Creating a Mesh from Scratch" on page 4).

Table 2: Summary of CHILD output files

File Suffix	Contents	No. of data items per time slice	Notes
`.nodes`	*Node (x,y) coordinates, ID one spoke, and boundary code*	*# of nodes*	*Spoke ID is the ID of any edge having node as origin*
`.edges`	*ID numbers of origin and destination nodes and ID of the counter-clockwise edge*	*# of edges*	*Complementary listed together, e.g. edge B->A follows edge A->B*
`.tri`	*ID numbers of vertex nodes (CCW order), ID numbers of neighboring triangles opposite the vertex nodes, and ID numbers of clockwise-oriented edges that originate with the vertex nodes*	*# of triangles*	*-1 is used when no triangle lies opposite a given vertex*
`.z`	*Node elevations in meters*	*# of nodes*	
`.area`	*Drainage areas in square meters*	*# of non-boundary nodes*	
`.net`	*ID of downstream node to which each node drains*	*# of non-boundary nodes*	
`.slp`	*Gradient of edge along which flow from each node is directed*	*# of non-boundary nodes*	*For meandering nodes, a weighted average of several node-slopes is used*

Table 2: Summary of CHILD output files

File Suffix	Contents	No. of data items per time slice	Notes
`.q`	*Discharge at nodes in m3/yr*	*# of nodes*	*Represents discharge of most recent storm*
`.lay`	*Layer data: number of layers at each node and, for each layer, creation time, recent activity time, surface exposure age, vertical thickness, erodibility (KB or KR), sed/rock flag, and volumetric proportion of each grain size fraction in layer*	*# of non-boundary nodes*	
`.tx`	*Volumetric proportion of sand in active layer*	*# of nodes*	*Only applicable to sand-gravel transport model*

Error Messages

Being a research tool, CHILD does relatively little error-checking. The burden is on you the user to make sure that the format of your inputs is correct and that the parameter settings are reasonable. However, a limited amount of error handling is provided, and it is common to see error messages when a parameter is missing in the input file, the input mesh contains an error, etc. If you should happen to receive an “assertion failed” message, please report it to one of the following people along with the text of the error message and the input file that for the run that caused the error:

Greg Tucker	gtucker@mit.edu
Nicole Gasparini	nmgaspar@mit.edu
Stephen Lancaster	lancasts@ccmail.orst.edu

Rafael Bras rlbras@mit.edu

References

Beven, K.J., and Kirkby, M.J., 1979, A physically based variable contributing area model of basin hydrology: Hydrological Sciences Bulletin, v. 24, no. 1, p. 43-69.

Gasparini, N.M., 1998, Erosion and Deposition of Multiple Grain Sizes in a Landscape Evolution Model, unpublished M.Sc. thesis, Massachusetts Institute of Technology.

Gasparini, N.M., Tucker, G.E., and Bras, R.L., 1999, Downstream fining through selective particle sorting in an equilibrium drainage network: Geology, submitted.

Lancaster, S.L., 1998, A nonlinear river meander model and its incorporation in a landscape evolution model, unpublished Ph.D. thesis, Massachusetts Institute of Technology.

Leopold, L. and Maddock, T., 1953, The hydraulic geometry of stream channels and some physiographic implications: Professional Paper 252, United States Geological Survvey.

Moglen, G.E., and Bras, R.L., 1994, Simulation of observed topography using a physically-based basin evolution model: Ralph M. Parsons Laboratory, Hydrology and Water Resources Report Number 340, MIT, Cambridge, MA.

O'Loughlin, E.M., 1986, Prediction of surface saturation zones in natural catchments: Water Resources Research, v. 22, p. 794-804.

Tetzlaff, D.M., and Harbaugh, J.W., 1989, Simulating Clastic Sedimentation, New York, Van Nostrand Reinhold, 202pp.

Tucker, G.E., and Bras, R.L., 1998, Hillslope processes, drainage density, and landscape morphology: Water Resources Research, v. 34, p. 2751-2764.

Tucker, G.E., Gasparini, N.M, Lancaster, S.L., and Bras, R.L., 1997, An integrated hillslope and channel evolution model as an investigation and prediction tool, Technical report prepared for U.S. Army Corps of Engineers Construction Engineering Research Laboratory.

Tucker, G.E., Lancaster, S.T., Gasparini, N.M., Bras, R.L., and Rybarczyk, S.M., 1999, An object-oriented framework for distributed hydrologic and geomorphic modeling using triangulated irregular networks: submitted to Computers and Geosciences.

Whipple, K.X., and Tucker, G.E., 1999, Dynamics of the stream power river incision model: implications for height limits of mountain ranges, landscape response timescales and research needs: J. Geophys. Res., in press.