

# PIHMgis: A *“Tightly Coupled”* GIS Framework for PIHM v2.2

Version: 3.5

## Users Guide



Hydrology Group  
Civil & Environmental Engineering  
Pennsylvania State University, University Park, USA.

## Table of Content

<b>1.</b>	<b>Introduction</b>	<b>3</b>
<b>2.</b>	<b>Initial Setup</b>	<b>3</b>
<b>3.</b>	<b>Understanding PIHMgis Framework</b>	<b>3</b>
<b>4.</b>	<b>Step [1]::Load/Create PIHMgis project settings</b>	<b>5</b>
<b>5.</b>	<b>Group [2]::Raster Processing</b>	<b>6</b>
5.1	Fill Pits: Step 1 of 7 for Raster Processing	6
5.2	Flow Grid: Step 2 of 7 for Raster Processing	7
5.3	Stream Grid: Step 3 of 7 for Raster Processing	8
5.4	Link Grid: Step 4 of 7 for Raster Processing	9
5.5	Catchment Grid: Step 5 of 7 for Raster Processing	10
5.6	Stream Polyline: Step 6 of 7 for Raster Processing	11
5.7	Catchment Polygon: Step 7 of 7 for Raster Processing	12
<b>6.</b>	<b>Group [3]::Vector Processing</b>	<b>13</b>
6.1	Dissolve Polygons: Step 1 of 5 for Vector Processing	13
6.2	Polygon to Line: Step 2 of 5 for Vector Processing	14
6.3	Simplify Polylines: Step 3 of 5 for Vector Processing	16
6.4	Polyline to Lines: Step 4 of 5 for Vector Processing	17
6.5	Merge Vector Layers: Step 5 of 5 for Vector Processing	19
<b>7.</b>	<b>Group [4]::Domain Decomposition</b>	<b>20</b>
7.1	Read Topology: Step 1 of 3 for Domain Decomposition	20
7.2	Triangulation: Step 2 of 3 for Domain Decomposition	21
7.3	TIN Shape Layer: Step 3 of 3 for Domain Decomposition	23
<b>8.</b>	<b>Group [5]::Data Model Loader</b>	<b>24</b>
8.1	Mesh (*.mesh) File	25
8.2	Attribute (*.att) File	26
8.3	River (*.riv) File	27
8.4	Soil (*.soil) File	28
8.5	Geology (*.geol) File	29
8.6	Land Cover (*.lc) Data File	30
8.7	Initial State Condition (*.init) Data File	31
8.8	Initial Boundary Conditions (*.ibc) Data File	32
8.9	Parameter (*.para) Data File	33
8.10	Calibration (*.calib) Data File	34
8.11	Forcing (*.forc) Data File	34
<b>9.</b>	<b>Group [6]::PIHM Simulation</b>	<b>35</b>
<b>10.</b>	<b>Optional::Hydro Informatics</b>	<b>36</b>
10.1	Temporal Plots	36
10.2	Spatial Plots	37
<b>11.</b>	<b>Debugging PIHMgis</b>	<b>38</b>

## 1. Introduction

Physically-based fully-distributed hydrologic models seek to simulate hydrologic state variables in space and time while using heterogeneous input data for climate, land use, topography and hydrogeology. In the process of incorporating several physical data layers in a hydrologic model requires intensive effort in data gathering, development as well as topology definitions. Traditionally Geographic Information System (GIS) has been used for data management, data analysis and visualization. Joint use and development of sophisticated numerical models and commercial GIS systems poses challenges that result from proprietary data structures, platform dependence, inflexibility in their data models and nondynamic data-interaction with pluggable software components. PIHMgis v3.5 is an open-source, platform independent, extensible and “tightly-coupled” integrated GIS interface to Penn State Integrated Hydrologic Model (PIHM v2.2). The tight coupling between the GIS and the model is achieved by developing the PIHMgis data-model to promote minimum data redundancy and optimal retrievability. Minimum data redundancy and optimal retrievability are facilitated through carefully designed data-model classes, relationships and integrity constraints.

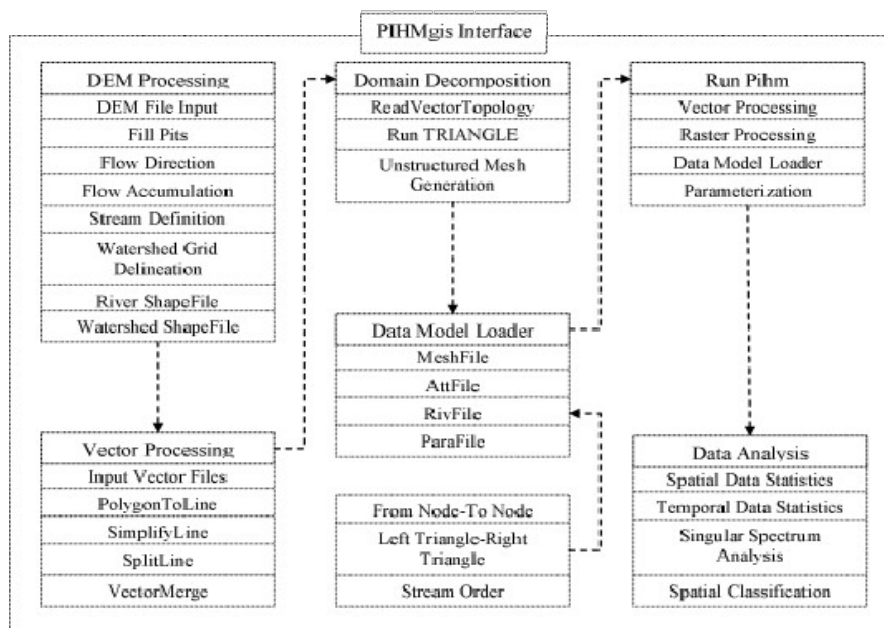
This tutorial has been designed to provide user with a step by step navigation. Starting scratch from a Digital Elevation Model (DEM) of any region of interest to model simulation and analysis based on model simulated results and observed data to help analyze the dynamics of different hydrologic processes and a better understanding of the parameters influencing the prediction variables. In this document it is also intended to provide user with brief internal operation taking place behind the steps performed.

## 2. Initial Setup

PIHMgis is platform independent (i.e. works on Windows, Linux, and MacX). Code and the PIHMgis v3.5 application (compiled and zipped with dependencies) are available to download from <https://github.com/leonard-psu/PIHMgis>. Visit this website for known build and installation issues. For example, Windows 10 users require windows-10-sdk from Microsoft.

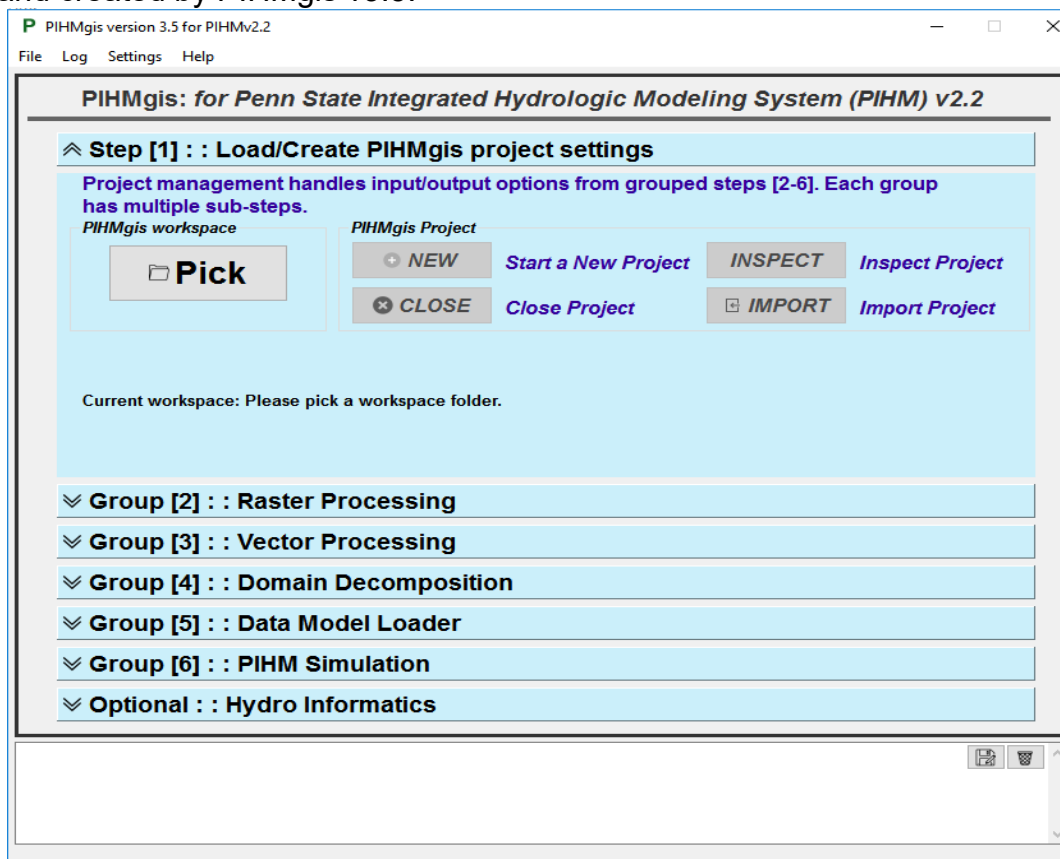
## 3. Understanding PIHMgis Framework

PIHMgis interface is interactive and procedural in nature. **Figure 3.1** shows the procedural framework of the interface. In the first step, **raster**-DEM-processing is facilitated for watershed delineation and stream definition. The **Vector** Processing module aids users in defining watershed properties and grid constraints using points (stream gauge, ground water observation-well locations), polygons (watershed and subshed boundary obtained in raster processing step, physiographic boundaries) and polylines (streams obtained from raster processing step). The **domain** constraints as well as internal and external boundaries are used to generate constrained Delaunay triangulations with certain restrictions on the minimum angle of each triangle. Topological and spatial data are assigned in an automated way in **Data Model** Loader Module. In the next step, data prepared in the previous steps are used by the PIHM v2.2 model. Data Analysis allows easier visualization of PIHM results.



**Figure 3.1: PIHMgis Procedural Framework**

The PIHMgis v3.5 interface (**Figure 3.2**) uses Qt as a standalone application. Users need to use their favorite GIS tool (i.e. QGIS, ESRI, GRASS) to examine and edit GIS files used and created by PIHMgis v3.5.



**Figure 3.2: PIHMgis v3.5 Interface**

## 4. Step [1]::Load/Create PIHMgis project settings

To get started with PIHMgis, you need to specify a workspace. To start a new project, you need to choose a base folder, by using the **Pick** button. The base folder will contain all the PIHMgis steps. A PIHMgis workspace contains folders for each of the group steps (e.g. Raster Processing, VectorProcessing). It is highly recommended to use this folder structure. If folders are missing from the workspace, error messages will be displayed in the log window (bottom portion of dialog) and the interface will be locked to prevent further action.

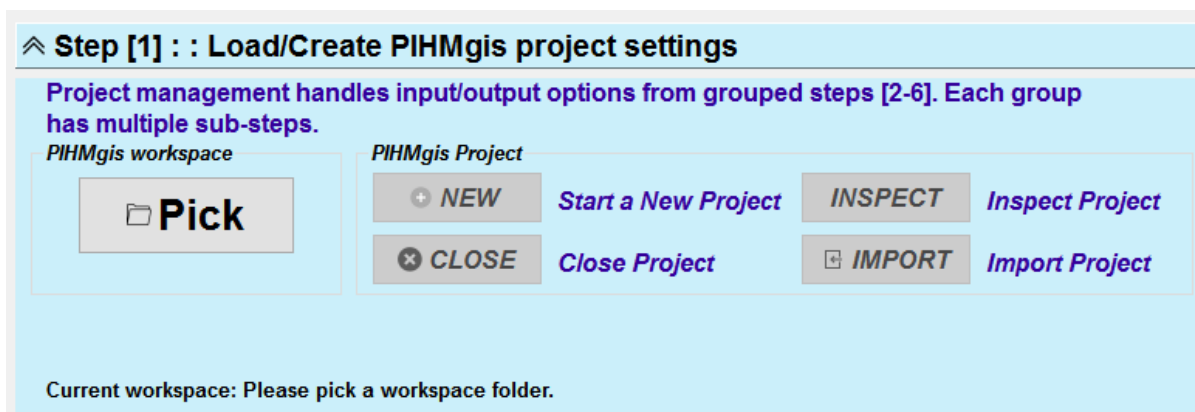
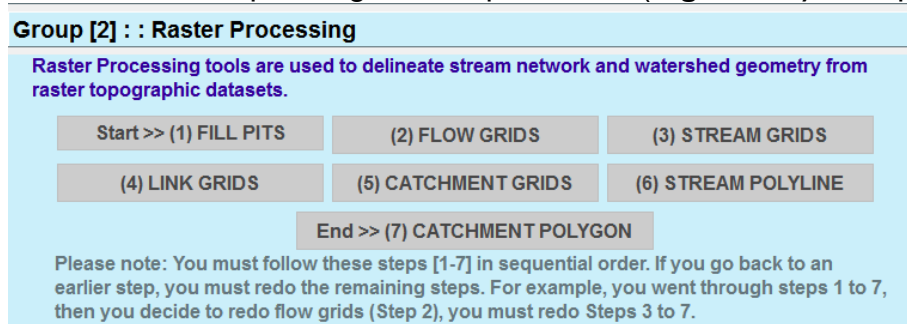


Figure 4.1: Picking a workspace

When picking a workspace, if a PIHMgis project already exists, the interface will be populated with the last user actions. If the workspace is empty, you will need to create a **New Project** which will create the workspace folders. If you have been using the older version of PIHMgis you can import the project using the **Import Project** button. To recall what steps have been completed, and which parameters used, the **Inspect Project** button provides the ability to search the current project. The **Close Project** button is used to close the current project to open a different workspace.

## 5. Group [2]::Raster Processing

Raster processing facilitates subshed/watershed delineation and stream definition from the Digital Elevation Model (DEM) of any region. In order to successfully complete raster processing, one needs to step through seven processes (**Figure 5.0**) in sequential order.

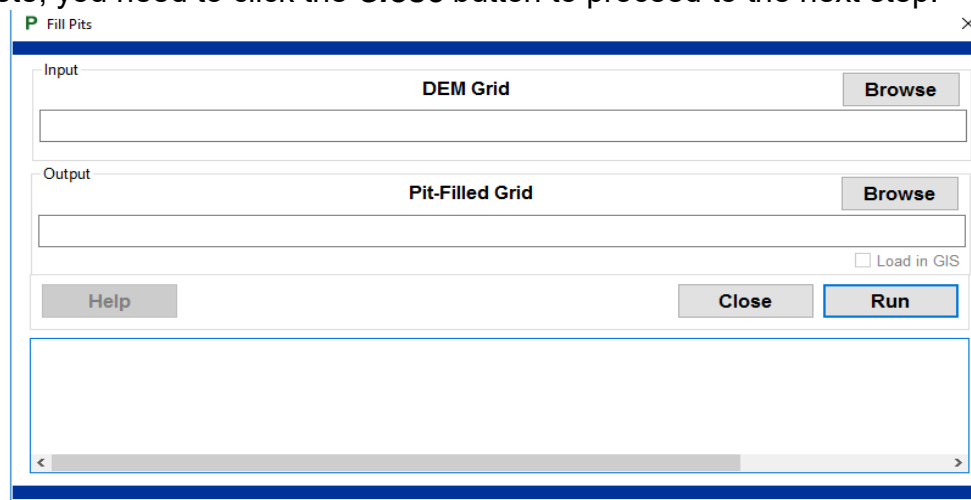


**Figure 5.0: The seven steps to complete raster processing for PIHMgis**

### 5.1 Fill Pits: Step 1 of 7 for Raster Processing

The Fill Pits step fills pits in a raster grid. If a cell is surrounded by higher elevation cells, the water is trapped in that cell and cannot flow. Pits are generally taken to be artifacts that interfere with the routing of flow across DEMs, so are removed by raising their elevation to the point where they drain off the edge of the DEM. Original pit locations can be identified and “protected” from getting modified by this function. (see TAUDem for more details, accessed 2006).

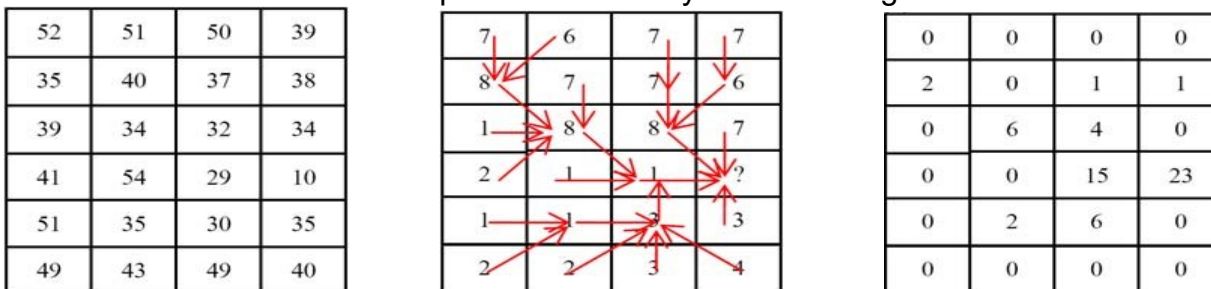
To get started, click on (1) **Fill Pits** button from the **Raster Processing** menu (**Figure 5.0**). This shows the Fill Pits modal dialog (**Figure 5.1**). In the Input section of the dialog, browse to the DEM file (ESRI binary (\*.adf) or Arc/Info ascii file (\*.asc)). In the Output section, browse a file name to where the pit-filled grid will be saved at. Next, click the **Run** button to start processing. Depending on the size and resolution of the DEM, the fill pit processing could take several minutes. The text browser at the bottom of the dialog provides information related to any error or processing steps. After the pit fills processing is complete, you need to click the **Close** button to proceed to the next step.



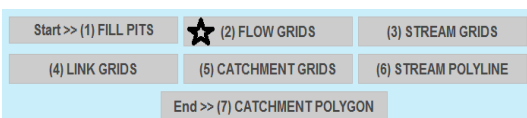
**Figure 5.1: Fill Pits Dialog (Raster Processing Step 1 of 7)**

## 5.2 Flow Grid: Step 2 of 7 for Raster Processing

**Flow Direction** outputs an encoded grid with the neighboring cell direction to which the steepest descent is found using D8 algorithm [O'Callaghan and Mark (1984)]. The encoding is 1 - east, 2 - North east, 3 - North, 4 - North west, 5 - West, 6 - South west, 7 - South, 8 - South east. **Flow Accumulation** outputs an accumulation grid that contains the accumulated number of cells upstream of a cell, for each cell in the input grid using a recursive procedure explained in (Mark, 1988). **Figure 5.2** shows the Flow direction and Flow accumulation calculations performed on a synthetic DEM grid.

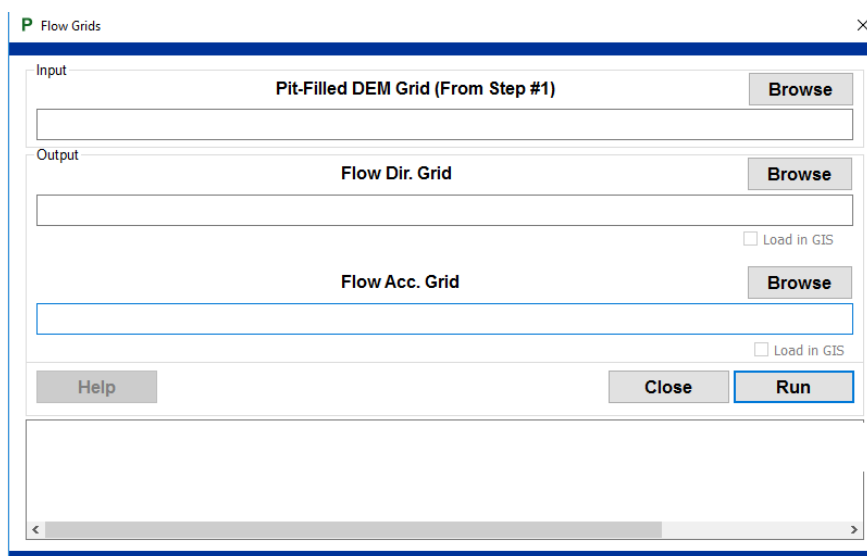


**Figure 5.2: Flow direction and Flow accumulation for synthetic grid.**



Click the (2) **Flow Grids** button from the **Raster Processing** menu. This shows the Fill Pits dialog (**Figure 5.3**).

In the Input section of the dialog, browse to the Pit Filled Grid generated by the step raster processing step one. In the Output section, browse to the file names to where the flow direction and flow accumulation grid will be saved at. Next, click on the **Run** button to start processing. Please be patient until processing completes. The text browser at the bottom of the dialog provides information related to any error or processing steps. After the processing step completes, you need to click on the **Close** button to proceed to the next step.



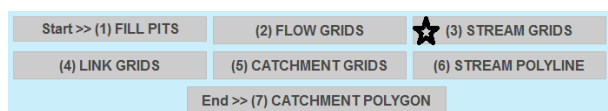
**Figure 5.3: Flow Grid Dialog (Raster Processing Step 2 of 7)**

### 5.3 Stream Grid: Step 3 of 7 for Raster Processing

Stream Grid is a raster equivalent of the stream network. Those Flow Accumulation Grid cells having value equal or greater than the threshold value user provides with, are marked 1. Physically threshold implies the number of cells draining to a cell is greater than the given value should be classified as a stream. Rest of the grid assumes No Data Value. **Figure 5.4** shows stream grid generated when a threshold value of 2 is applied to the flow accumulation grid produced in the previous section.

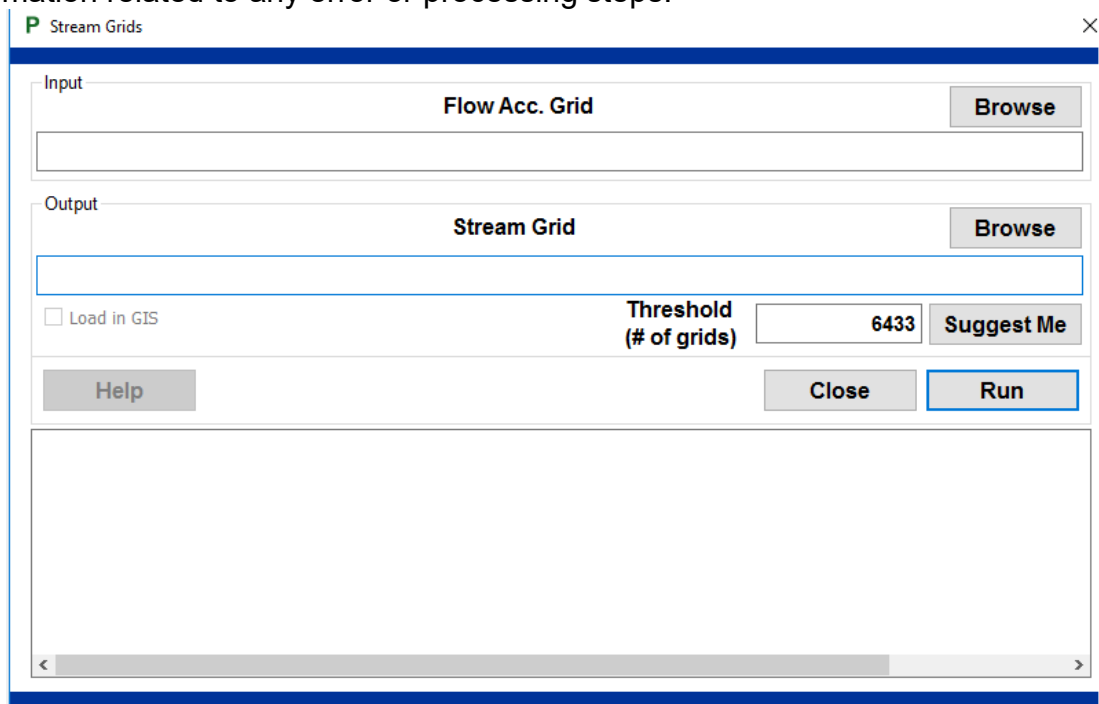
0	0	0	0
1	0	0	0
0	1	1	0
0	0	1	1
0	1	1	0
0	0	0	0

**Figure 5.4: Stream Grid for the synthetic grid.**



Click the (3) **Stream Grids** button from the Raster Processing menu. This shows the Stream Grid dialog (**Figure 5.5**).

In the Input section of the dialog, browse to the Flow Accumulation Grid generated by step 5.2. In the Output section of the dialog, browse to the file name to which the stream grid will be saved. An “integer” value for the grid threshold is required. Click on the **Suggest Me** button for a suggested threshold based on your grid. Any Flow Accumulation Grid cells with having value greater than threshold will be classified as a stream grid. Click on **Run** to begin processing. The text browser at the bottom of the dialog provides information related to any error or processing steps.



**Figure 4.5: Stream Grid Dialog (Raster Processing Step 3 of 7)**

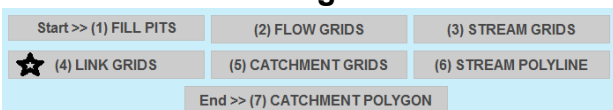


## 5.4 Link Grid: Step 4 of 7 for Raster Processing

Link Grid separates the stream grid segments at the junctions. Each Link Grid segment is assigned a unique integer value starting with 1. The rest of the grid assumes NoData value like that of Stream Grid. **Figure 5.6** shows the Link Grid generated corresponding to the Stream Grid obtained in the previous section.

0	0	0	0
3	0	0	0
0	3	3	0
0	0	1	1
0	2	2	0
0	0	0	0

**Figure 5.6: Link Grid for the synthetic grid.**



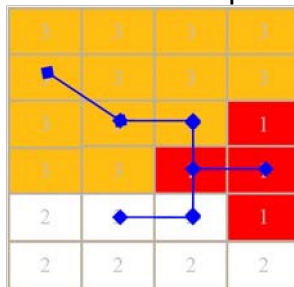
Click the (4) **Link Grids** button from the Raster Processing menu. This shows the Link Grid dialog (**Figure 5.7**).

In the Input section of the dialog, browse for the Stream and Flow Direction Grids generated by steps 5.3 and 5.2 respectively. In the Output section of the dialog, browse for the file name to where the Link Grid will be saved at. Now click on the **Run** button to begin processing.

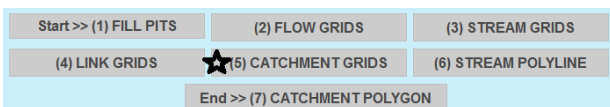
**Figure 5.7: Link Grid Dialog (Raster Processing Step 4 of 7)**

## 5.5 Catchment Grid: Step 5 of 7 for Raster Processing

All the grids draining to a stream polyline element are grouped into one type of catchment grid. Catchment grids are marked according to the stream polyline nomenclature with integer numbers starting with 1. **Figure 5.8** shows the catchment grid obtained for the synthetic grid using Stream and Flow direction grids discussed in sections 5.3 and 5.2 respectively. Different colors are used for clear representation of the catchment grid.



**Figure 5.8: Catchment grid for the synthetic grid.**



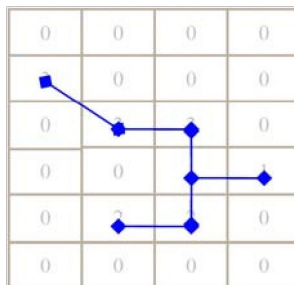
Click the (5) **Catchment Grids** button from the Raster Processing menu. This shows the Catchment Grid dialog (**Figure 5.9**).

In the Input section of the dialog, browse for the Link and Flow Direction Grids generated by steps 5.4 and 5.2 respectively. In the Output section of the dialog, browse for the file name to where the Catchment Grid will be saved at. Click on the **Run** button to start.

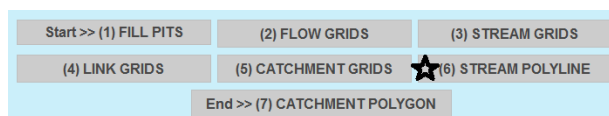
**Figure 5.9: Catchment Grid Dialog (Raster Processing Step 5 of 7)**

## 5.6 Stream Polyline: Step 6 of 7 for Raster Processing

Stream polylines are the drainage network for the region of interest obtained by the conversion of the link grid to the vector format from the raster. Each link segment forms an individual stream segment and connected at the junction points. Flow direction is used to ensure that the segments are topographically correct (i.e. FromNode and ToNode are consistent with the flow direction). **Figure 5.10** shows the stream polyline obtained corresponding to the link grid and flow direction grid obtained in the previous section for the synthetic grid.

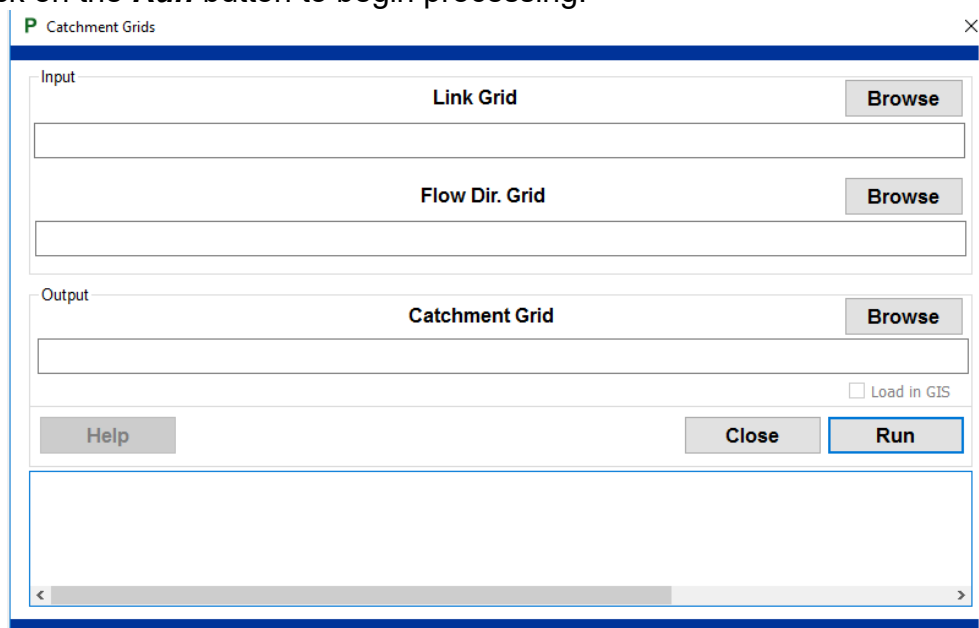


**Figure 5.10: Stream Polyline for the synthetic grid.**



Click the (6) **Stream Polyline** button from the **Raster Processing** menu. This shows the Stream PolyLine dialog (**Figure 5.11**).

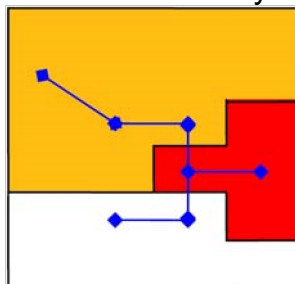
In the Input section of the **Catchment Grids** dialog, browse for the Link and Flow Direction Grids generated by steps 5.4 and 5.2 respectively. In the Output section of the dialog, browse for the shape file name to where the Stream Polyline will be saved at. Next, click on the **Run** button to begin processing.



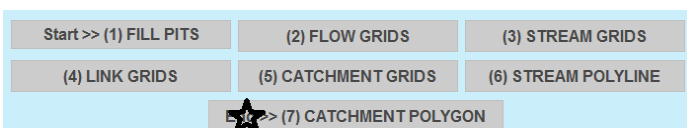
**Figure 5.11: Catchment Grids Dialog (Raster Processing Step 6 of 7)**

## 5.7 Catchment Polygon: Step 7 of 7 for Raster Processing

Catchment polygons are the vector representation of the catchment grid. Similar to the catchment grid, a catchment polygon bounds the region which has a single drainage outlet. **Figure 5.12** shows the catchment polygon obtained corresponding to the catchment grid obtained in the Section 5.5 for the synthetic grid.

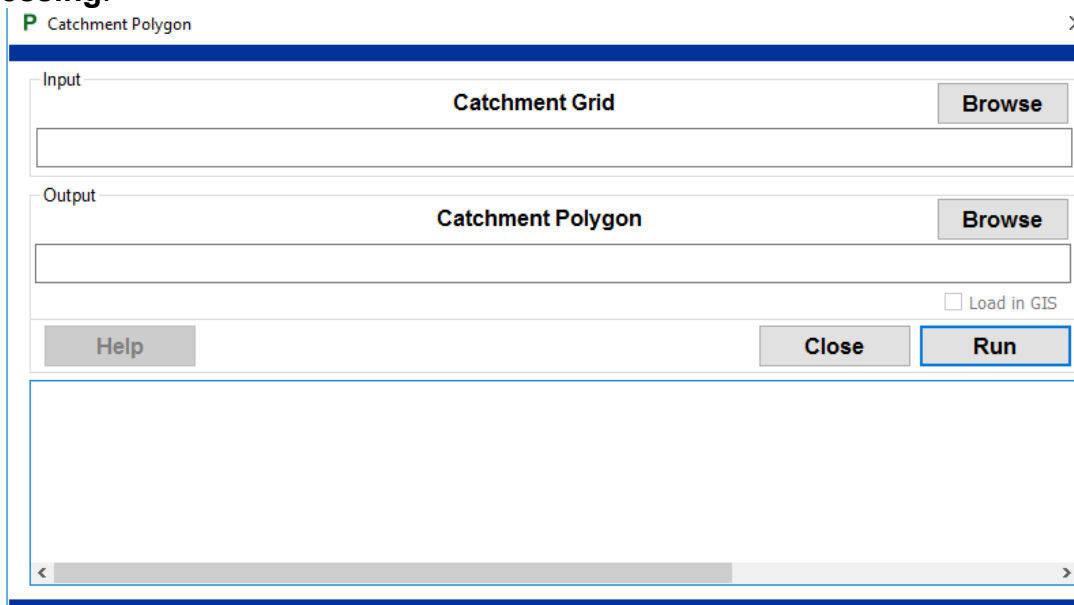


**Figure 5.12: Catchment Polygon for the synthetic grid.**



Click on the (7) **Catchment Polygon** button from the Raster Processing menu. This shows the Catchment Polygon dialog (**Figure 5.13**).

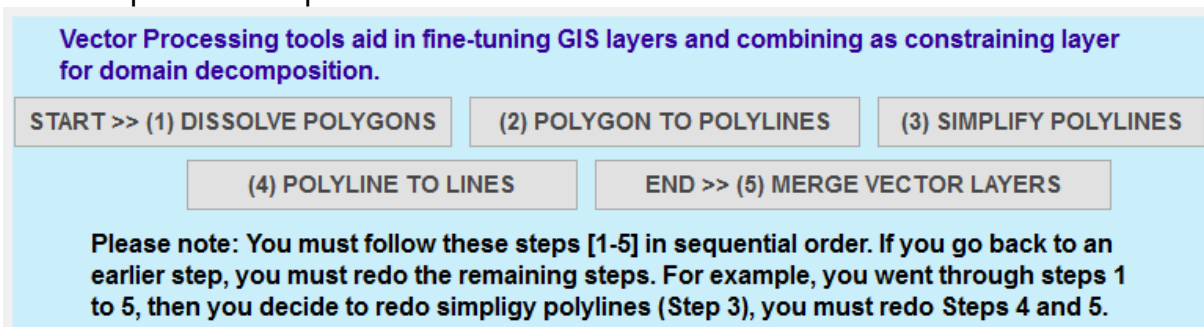
In the Input section of the **Catchment Polygon** dialog, browse to the Catchment Grid generated by step 5.5. In the Output section of the dialog, browse for the file name to where the Catchment Polygon will be saved at. Next click on the **Run** button to begin processing. Please be patient until processing completes. The text browser at the bottom of the dialog provides information related to any error or processing steps. After the processing is complete you need to press **Close** button to proceed to **Group 3 Vector Processing**.



**Figure 5.13: Catchment Polygon Dialog (Raster Processing Step 7 of 7)**

## 6. Group [3]::Vector Processing

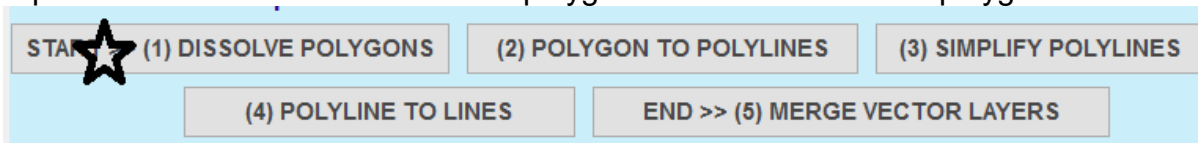
Vector processing consists of a set of operations which enables efficient discretization of the modeling domain. Stream polyline and Catchment polygon created in sections 5.6 and 5.7 are used primarily for this purpose. However, other hydrologic constraints such as soil coverage, land cover type can also be incorporated. Vector processing prepares a GIS layer which is used as an input constraint for domain decomposition (Group 4) of the watershed domain. **Figure 6** summarizes the five Vector Processing steps that need to be completed in sequential order.



**Figure 6: The five steps to complete vector processing for PIHMgis**

### 6.1 Dissolve Polygons: Step 1 of 5 for Vector Processing

Geo-data features can exist as point, line or polygon geometry types. In order to merge all the data features together, before the objects can be used by domain decomposition, the object properties for all the features need to be the same type of geometry. The first step is to dissolve the sub-watershed polygons into one catchment polygon.



**Click on the (1) Dissolve Polygons button from the Vector Processing menu.**  
**This shows the Dissolve Polygons dialog (Figure 6.1).**

In the **Input Layers** column of the **Dissolve Polygons** dialog (**Figure 6.2**), browse for the Catchment Polygon generated by step 5.7 using the **Add** button or entering text. In the **Dissolved layers** column of the dialog, specify where the dissolved catchment polygon will be saved at. You can **Add** more than one polygon to be dissolved. Use the **Remove** button to select a row to remove. Use the **Clear** button to remove all layers. Click on the **Run** button to begin processing.

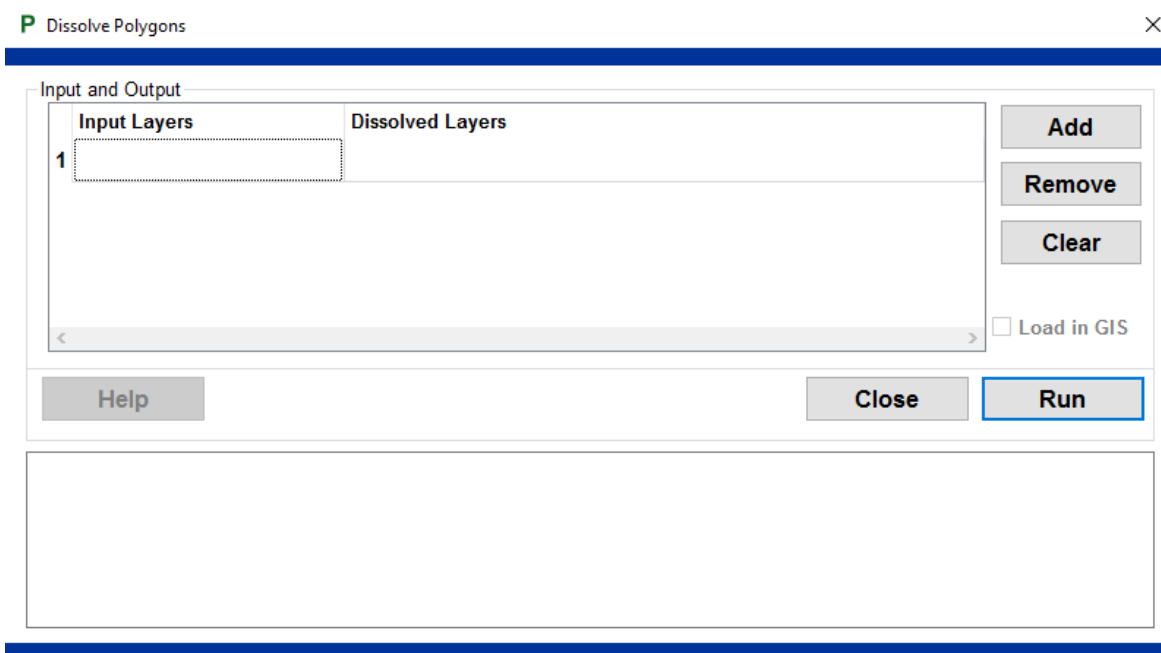


Figure 6.2: Dissolve Polygons Dialog (Vector Processing Step 1 of 5)

## 6.2 Polygon to Line: Step 2 of 5 for Vector Processing

Next, polygons need to be converted from polygon to polyline geometry. For example, lake features and catchments need to be converted to polylines before they can be merged with existing river line features. **Figure 6.3** shows a schematic of the steps used by the Polygon to Line algorithm.

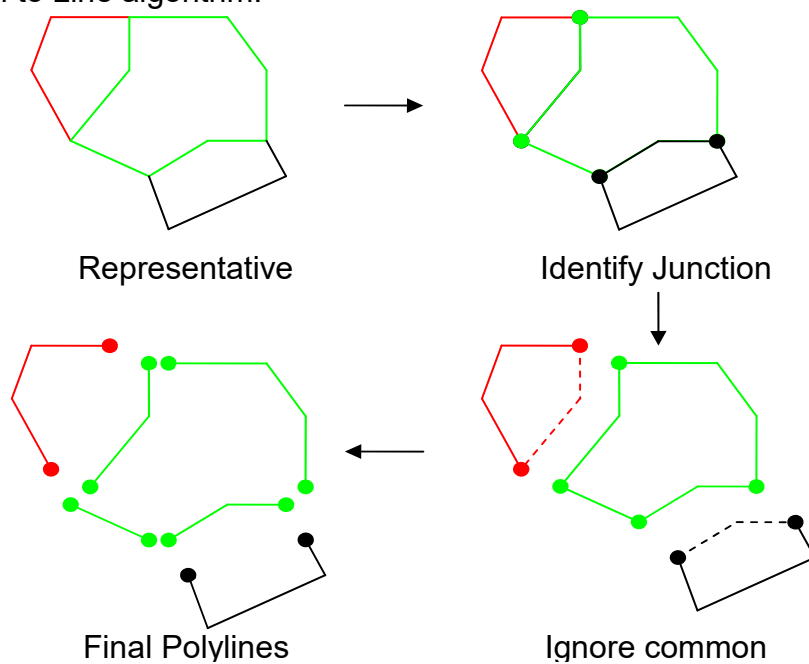
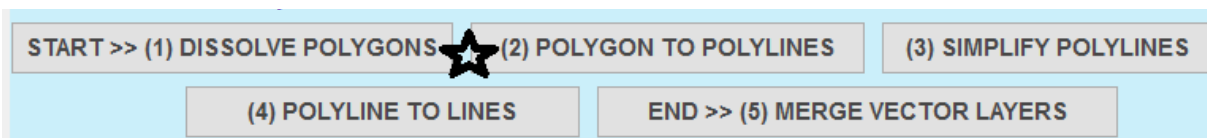
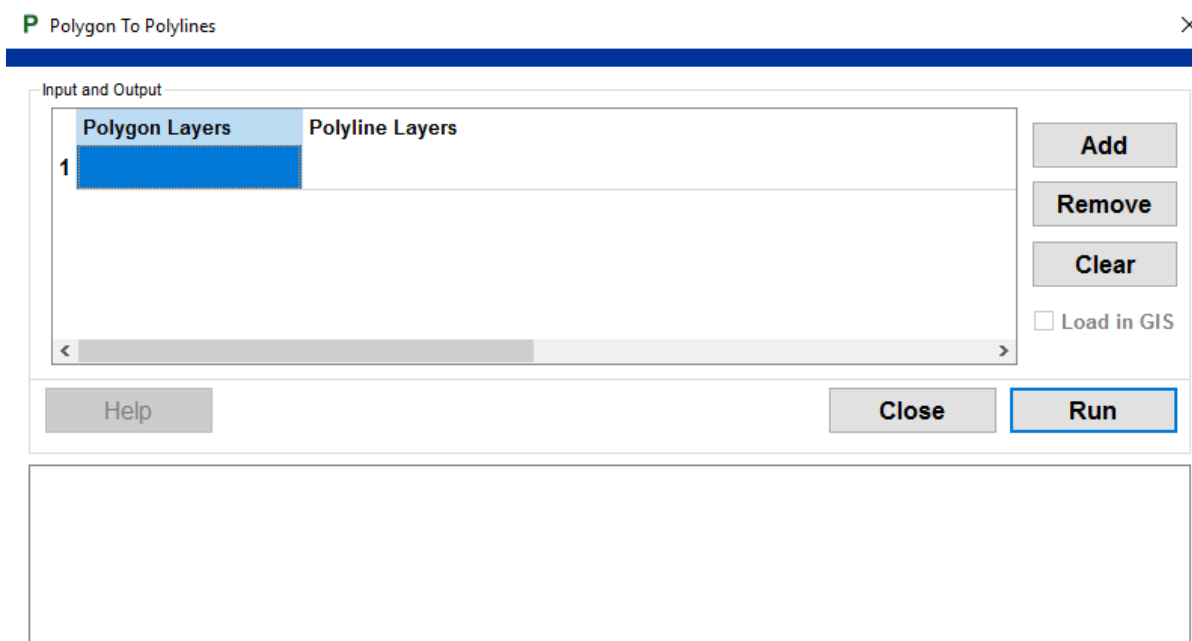


Figure 6.3: Steps for Polygon to Polyline



Click on the (2) Polygons to Polylines button from the Vector Processing menu. This shows the Polygons to Polylines dialog (Figure 6.4).

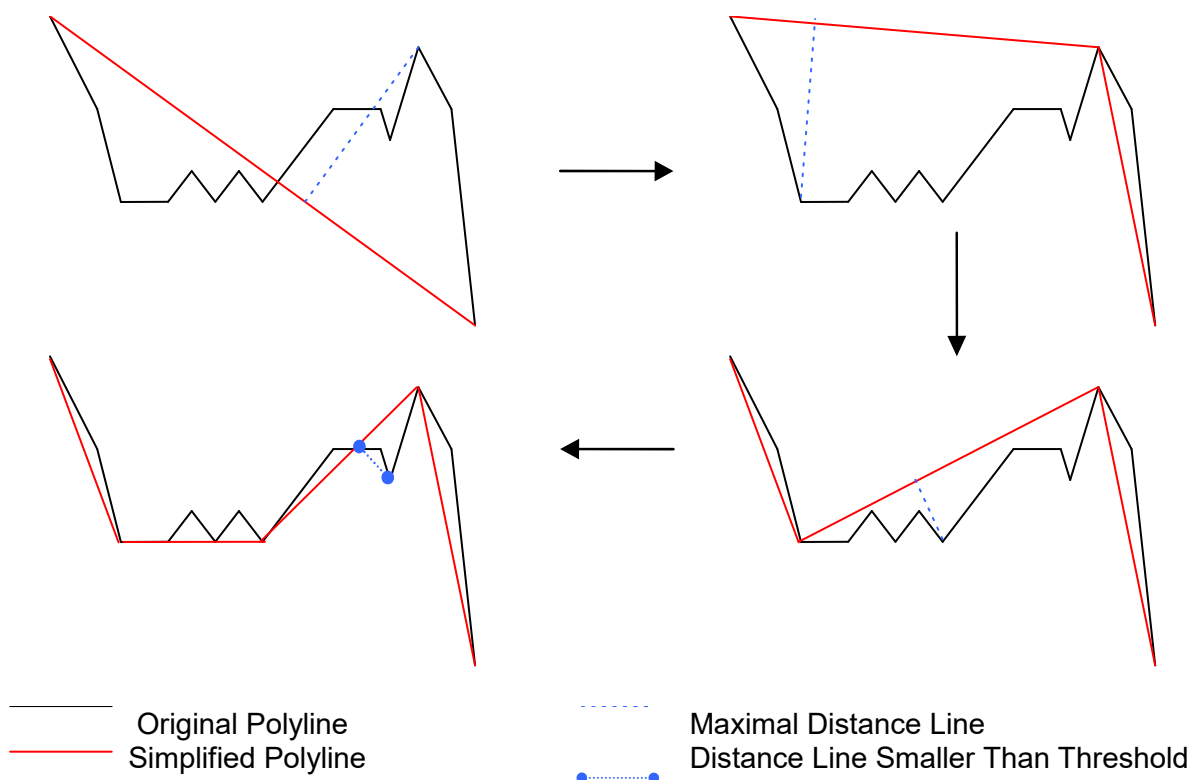


**Figure 6.5: Polygon to Polylines Dialog (Vector Processing Step 2 of 5)**

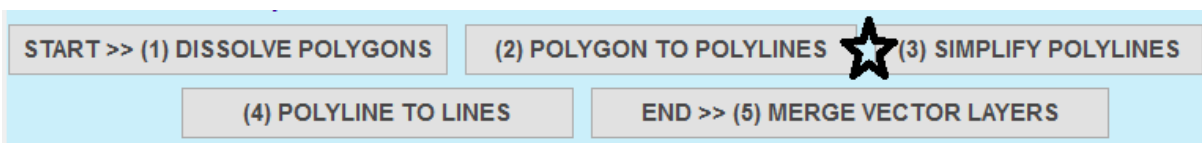
To add shape file(s) of “polygon” type, use the **Add** button on the far right of the dialog (**Figure 6.5**). A default output file name for the corresponding polyline file is generated which is editable if desired. The **Remove** button can be used to remove any file from the simplification routine by selecting the file from the left bar column of the spreadsheet. The **Clear** button clears everything from the dialog. After specifying polygon files to be converted, click on the **Run** button to proceed. The text browser provides information regarding any error or processing steps. Click on the **Close** button to proceed to the next step.

### 6.3 Simplify Polylines: Step 3 of 5 for Vector Processing

Simplify Polylines is used to simplify a polyline by removing small fluctuations or extraneous bends from it while preserving its essential shape. This step becomes particularly crucial for mesh quality and efficient domain decomposition as an unsimplified feature can have unnecessarily large number of nodes in it, which in turn determines the number of triangulations generated (Group 4 Domain Decomposition). Creating a larger number of decomposed triangle elements, increasing the computational requirements of numerical PIHM simulation. **Figure 6.6** shows intermediate steps in polyline simplification using the Douglas-Peucker algorithm.

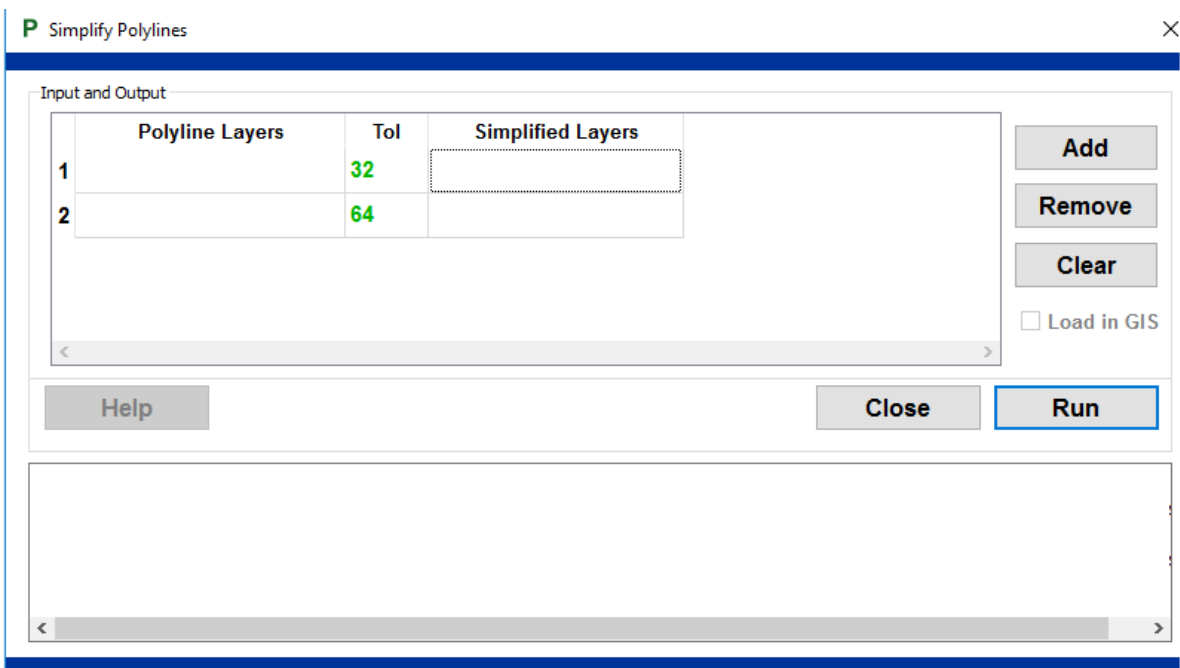


**Figure 6.6: Algorithm for Polyline Simplification**



Click on the (3) Simplify Polylines button from the Vector Processing menu. This shows the Simplify Polylines dialog (Figure 6.7).



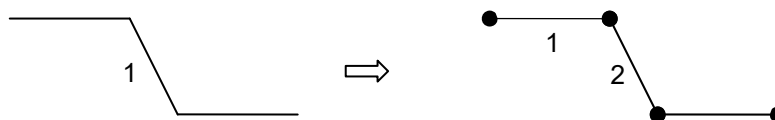


**Figure 6.8: Simplify Polyline Dialog (Vector Processing Step 3 of 5)**

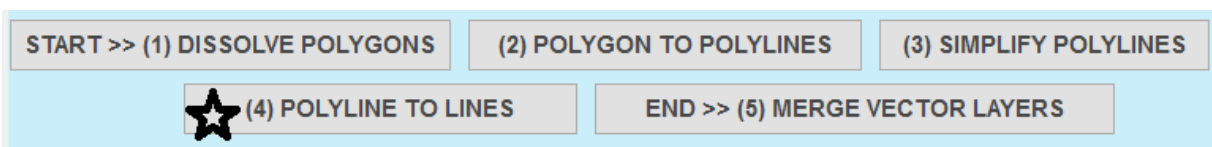
To add shape file(s) of “polyline” type, use the **Add** button on the far right of the dialog (**Figure 6.8**). A tolerance (double value) is necessary which acts as the maximum distance value (shapefile length unit) for the simplification algorithm. The **Remove** button can be used to remove any file from the simplification routine by selecting the file from the left bar column of the spreadsheet. The **Clear** button clears everything from the dialog. After specifying polyline files to be converted, click on the **Run** button to proceed. The text browser provides information regarding any error or processing steps. Click on the **Close** button to proceed to the next step.

#### 6.4 Polyline to Lines: Step 4 of 5 for Vector Processing

Before merging all the features together, it is necessary to have all features of the same geometry type. Polyline to lines splits polylines at each vertex. Therefore, this turns a single polyline feature into a multiple line feature depending upon the number of vertices present in the original polyline.



**Figure 6.9: Schematically describes split line.**



Click on the (4) Polyline to Lines button from the Vector Processing menu.  
This shows the Polyline to Lines dialog (Figure 6.10).

To add shape file(s) of “polyline” type (from section 6.2), use the **Add** button on the far right of the dialog (Figure 6.11). The **Remove** button can be used to remove any file from the conversion routine by selecting the file from the left bar column of the spreadsheet. The **Clear** button clears everything from the dialog. After specifying polyline files to be converted, click on the **Run** button to proceed. The text browser provides information regarding any error or processing steps. Click on the **Close** button to proceed to the next step.

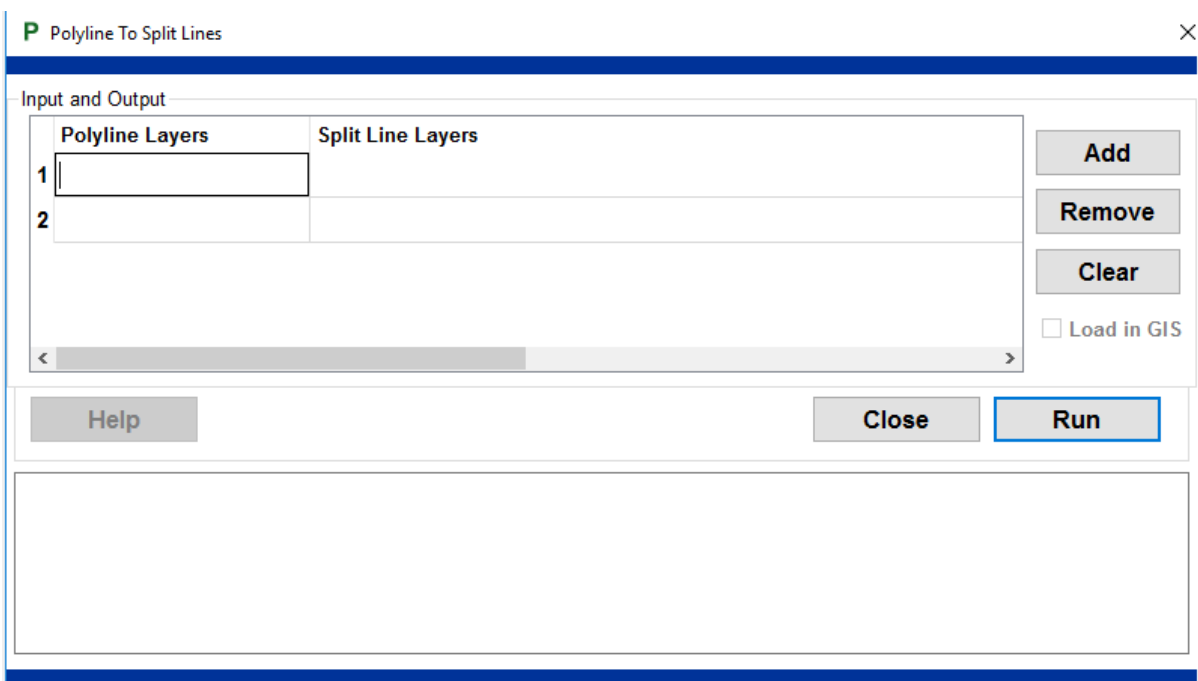


Figure 6.11: Polyline to Lines Dialog (Vector Processing Step 4 of 5)

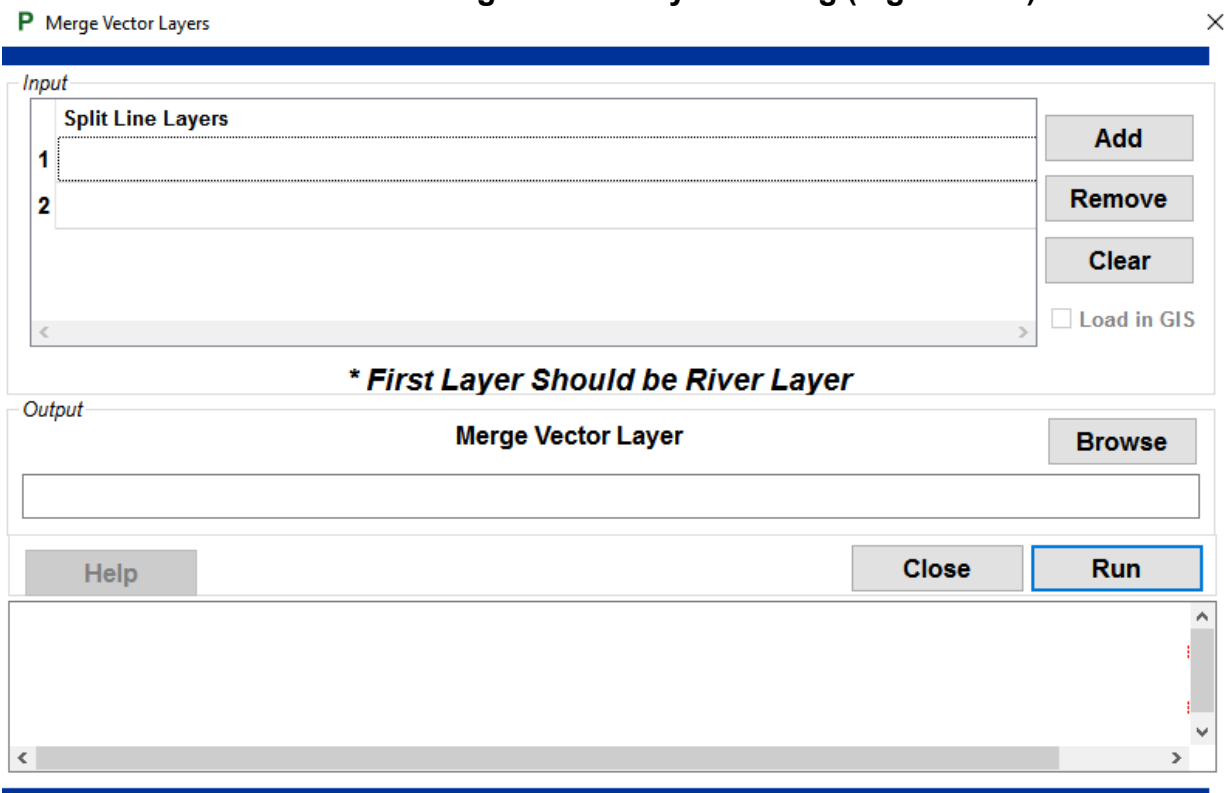
**Important Note:** Check that the stream network does not overlap the catchment boundary. Use your favorite GIS tool to “snap” the outlet node to the catchment boundary and/or edit the stream network.

## 6.5 Merge Vector Layers: Step 5 of 5 for Vector Processing

Vector Merge is the final step of Vector processing. It merges all the layers into one shape file. The merged shape file acts as constraints in the domain decomposition process.



Click on the (5) Merge Vector Layers from the Vector Processing menu.  
This shows the Merge Vector Layers dialog (Figure 6.12).



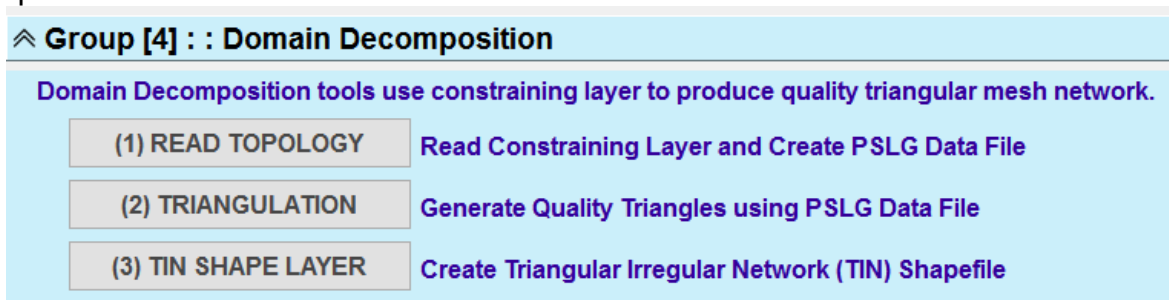
**Figure 6.13: Merge Vector Layers Dialog (Vector Processing Step 5 of 5)**

To add shape file(s) of “line” type (from section 5.4), use the **Add** button on the far right of the dialog (**Figure 6.13**). The **Remove** button can be used to remove any file from the merge routine by selecting the file from the left bar column of the spreadsheet. The **Clear** button clears everything from the dialog. After specifying files to be merge, click on the **Run** button to proceed. The text browser provides information regarding any error or processing steps. Click on the **Close** button to proceed to the next step.

**Important Note:** The outlet node must not overlap the catchment boundary. Use your favorite GIS tool to “snap” the outlet node to the catchment boundary.

## 7. Group [4]::Domain Decomposition

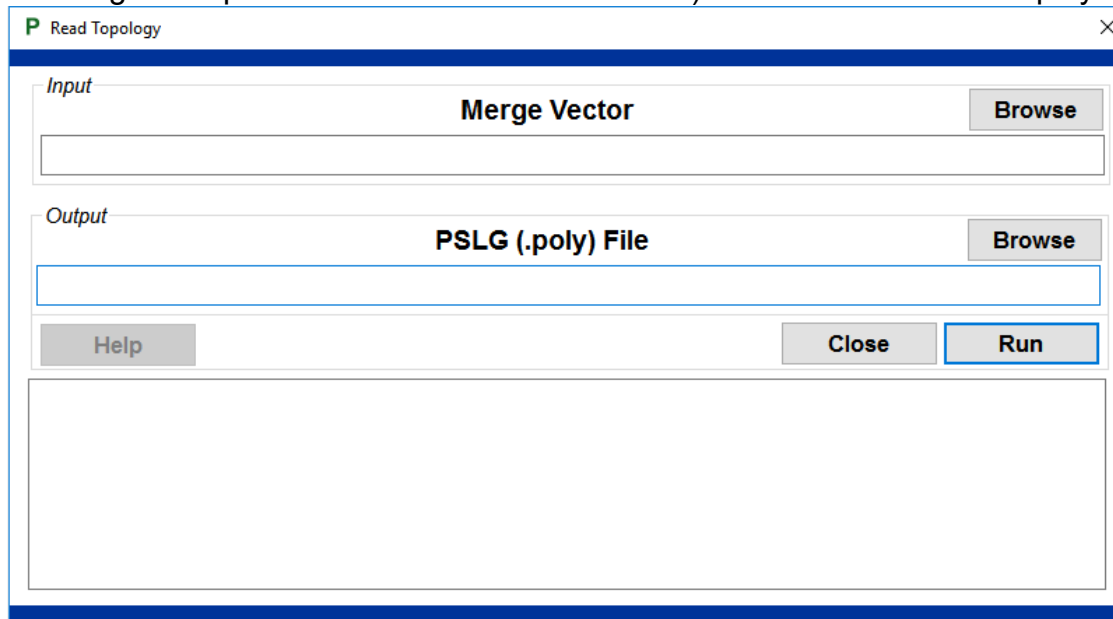
Domain decomposition applies Delaunay Triangulation [Delaunay, 1934] to decompose the modeling domain into triangular irregular mesh or triangular irregular network (TIN). A terrain can be better represented by an irregular mesh if all the critical terrain and hydrographic points are considered while performing domain decomposition. This may include watershed boundary, different types of contours (e.g. hypsometry, soil), stream network, hydraulic structures (e.g. dams, gages) for generating those points. **Figure 7** summarizes the three Domain Decomposition steps that need to be completed in sequential order.



**Figure 7: The three steps to complete Domain Decomposition for PIHMgis**

### 7.1 Read Topology: Step 1 of 3 for Domain Decomposition

This step prepares a \*.poly file, required input for running the **TRIANGLE** tool in the next section. All the geometry (nodes/vertices and lines) information from the input shape file (vector merged shape file obtained in the section 6.5) is transformed into the poly file.



**Figure 7.1: Read Topology (Domain Decomposition Step 1 of 3)**

In the input section of the dialog (**Figure 7.1**), browse for the shape file (merged shape file obtained in the section 6.5) which acts as constraining layer for domain decomposition. In the output section, browse to specify an output file name for the poly file. Click the **Run** button to proceed.

## 7.2 Triangulation: Step 2 of 3 for Domain Decomposition

TRIANGLE developed by Shewchuk [2001] takes planar straight line graph (PSLG) as input to generate a Delaunay triangulation, by inserting carefully placed vertices until the generated mesh meets a provided quality and size criterion.

The screenshot shows a software window titled "Delaunay Triangulations". It features a "Browse" button next to a text field for the "PSLG (.poly) File". Under "Input Options", there are three checked checkboxes: "Angle (degrees)" with a value of 20, "Area (sq. meters)" with a value of 10500, and "Other Options" with a value of -p. There are "Help", "Close", and "Run" buttons. At the bottom, there is a section for "Triangle Software Location" with a "Find" button and a large empty text area for specifying the tool and location.

**Figure 7.2: Delaunay Triangulation (Domain Decomposition Step 2 of 3)**

To use Triangle software, you need to specify the location of Triangle (i.e. Triangle.exe), using the **Find** button (**Figure 7.2**). Then specify the Input **poly** file created from Section 7.1. To specify the **minimum angle** constraint or not, use the Angle checkbox. To constrain the triangle area, specify the maximum **area** and enable the Area checkbox. The last input is **other options**. To use poly files, you will need to specify -p. This is not automated input to Triangle, as there are many controls for Triangle that you may use, as well as enable the user to specify different versions of Triangle.

Clicking the **Run** button executes TRIANGLE to generate four (4) output files: (1) **.poly** file; (2) **.node** file; (3) **.ele** file; and (4) **.neigh** file. They are stored in the same directory as the input poly file.

**Important note**, that the values provided for minimum angle and maximum area influence the number of triangles generated. It is recommended to minimize the number of triangles at first to validate all PIHM inputs. Then refine your TIN after checking PIHM is running.

**Table 7: TRIANGLE options**

Options	Description
-p	Triangulates a Planar Straight Line Graph (.poly file).
-r	Refines a previously generated mesh.
-q	Quality mesh generation. A minimum angle may be specified.
-a	Applies a maximum triangle area constraint.
-u	Applies a user-defined triangle constraint.
-A	Applies attributes to identify triangles in certain regions.
-c	Encloses the convex hull with segments.
-w	Weighted Delaunay triangulation.
-W	Regular triangulation (lower hull of a height field).
-j	Jettison unused vertices from output .node file.
-n	Generates a list of triangle neighbors.
-B	Suppresses output of boundary information.
-P	Suppresses output of .poly file. (.poly is required for PIHMgis)
-N	Suppresses output of .node file. (.node is required for PIHMgis)
-E	Suppresses output of .ele file. (.ele is required for PIHMgis)
-l	Suppresses mesh iteration numbers.
-O	Ignores holes in .poly file.
-X	Suppresses use of exact arithmetic.
-z	Numbers all items starting from zero (rather than one).
-o2	Generates second-order subparametric elements.
-Y	Suppresses boundary segment splitting.
-S	Specifies maximum number of added Steiner points.
-i	Uses incremental method, rather than divide-and-conquer.
-F	Uses Fortune's sweepline algorithm, rather than d-and-c.
-l	Uses vertical cuts only, rather than alternating cuts.
-s	Force segments into mesh by splitting (instead of using CDT).
-C	Check consistency of final mesh.
-Q	Quiet: No terminal output except errors.
-V	Verbose: Detailed information on what I'm doing.
-h	Help: Detailed instructions for Triangle.

**Note:** These options are for Triangle (**version 1.7.0\_acute**) A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Copyright 1993, 1995, 1997, 1998, 2002, 2005 Jonathan Richard Shewchuk. Your options may be slightly different.

### 7.3 TIN Shape Layer: Step 3 of 3 for Domain Decomposition

This step uses the .ele and .node files produced by TRIANGLE software (Step 2) and generates an unstructured mesh **shapefile** with all the triangle elements. In the input section of the dialog (**Figure 7.3**), browse for the input files (.ele file and .node files) generated by TRIANGLE in Section 7.2. In the output section, browse to a location where to save the unstructured mesh shape file. Click on the **Run** executes the routine. Any error or progress information is displayed in the text browser at the bottom of the dialog.

The screenshot shows a software dialog box titled "Delaunay Triangulation". It contains several input fields and buttons. Under the "Input" section, there are two rows: "Element File" and "Node File", each with a text box and a "Browse" button. Under the "Output" section, there is one row: "TIN Shape Layer" with a text box and a "Browse" button. Below these is a checkbox labeled "Load in GIS". At the bottom of the dialog are three buttons: "Help", "Close", and "Run". A large empty text area is at the very bottom of the dialog.

**Figure 7.3: TIN Shape Layer (Domain Decomposition Step 3 of 3)**

**Note:** It is recommended to check the unstructured mesh shapefile with your favorite GIS tool. Try to eliminate small triangles that often form along your stream network by changing values in Section 7.2.

## 8. Group [5]::Data Model Loader

The Data model loader steps convert GIS data into PIHM v2.2 input files. The files prepared constitute spatial and relational attributes of the modeling watershed domain. PIHM v2.2 requires 11 input files and are summarized in **Figure 8**. It is not necessary to complete these steps sequentially. However, often issues happen with the unstructured grid mesh and stream network. Therefore, completing these steps in order first (Steps 1-3) is recommended.

**⤴ Group [5] : : Data Model Loader**

**Data Model Loader tools perform automated assignment of topology and for mapping spatio-temporal watershed and climatological properties to mesh elements and river reaches.**

**Create PIHM Input Files:**

(1) MESH file	(2) ATT file
(3) RIV file	(4) SOILfile
(5) GEOL file	(6) LC file
(7) INIT file	(8) IBC file
(9) PARAM file	(10) CALIB file
(11) FORC file	

**Figure 8: The 11 input files required for PIHM v2.2**



## 8.1 Mesh (\*.mesh) File

The Mesh file contains the unstructured mesh geometry by storing nodes (vertices) and elements (triangles). Each node contains X, Y spatial locations, the surface elevation (Z) and the bedrock depth. Each triangle element contains the indices of the three nodes and topological relationships with neighboring elements.

**Figure 8.1: Mesh Data File Generation Dialog**

In the top section of the dialog (**Figure 8.1**) specify the location of the input files. The first three input files (element, node, and neighbor files) are those created by the TRIANGLE software in **Section 7.2**. In the Elevation section, specify the surface (pit-filled generated in Group 2:: Step 1) and bedrock elevation of the modeling domain. The user either needs to specify a bedrock elevation raster dataset (generated by your favorite GIS tool) or by specifying a subsurface thickness using the checkbox (by default is on). In the output section, specify the location where to save the mesh data file. Click on the Run button to generate the PIHM v2.2 mesh file.

**Important Note:** All the input files must have the same geographic coordinate system. **Any areas with no data will be assigned -9999 values, with warning messages.** At present, these are not considered errors.

## 8.2 Attribute (\*.att) File

An Attribute (\*.att) file contains physical parameters for each triangle (element) of the unstructured grid mesh. For example, soil, land cover, and several forcing types. There are three ways to specify these values: (1) Homogeneous class, (2) Raster layer classification, (3) Parameter value or raster layer.

**Att Data File**

Climate Classifications    Soil, Geology, Land Cover    Initial / Boundary Conditions

Precipitation <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>
Temperature <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>
Relative Humidity <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>
Wind Velocity <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>
Solar Radiation <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>
Vapor Pressure <sup>1</sup>	<input checked="" type="checkbox"/>	<input type="text" value="1"/>	<input type="button" value="Browse"/>

**TIN Layer**

**Output** **Att Data File**

<sup>1</sup>Homogeneous Class    <sup>2</sup>Classification Raster Layer  
<sup>3</sup>Parameter Value / Raster Layer

**Figure 8.2: Attribute File Generation Dialog**

In the input section of the dialog (Figure 8.2), specify the TIN shape file generated in **Section 7.3**. There are three general categories to specify values (1) Climate, (2) Soil, Geology, Land Cover, and (3) Initial/Boundary Conditions. To use raster classifications with any of these parameters, uncheck the checkbox to browse for the input file. Click the **Run** button to generate the attribute file.

**Important Notes:** With classifications, the class number should be an integer starting with 1. All grid coverage extents need to be larger than the TIN domain and use the same spatial reference as the input TIN.

### 8.3 River (\*.riv) File

Topological information related to river segments (such as Node information; Left and Right Elements) is stored in River (\*.riv) file. River segments are assigned different shape and material properties using Strahler order.

**P Riv Data File** [X]

**Input**

Element File	<input type="text"/>	Browse
Node File	<input type="text"/>	Browse
Neighbour File	<input type="text"/>	Browse
River Shape File	<input type="text"/>	Browse

**Outlet(s) Boundary Condition**

☐ Dirichlet   ☐ Neumann   ☒ Zero-depth   ☐ Critical-depth

**Output**

Riv Data File  Browse

Help Close Run

**Figure 8.3: River File Generation Dialog**

In the input section of the dialog (**Figure 8.3**), specify the River shape file (simplified) created in Section 6.3 (Group 3::Step 3 Simplified polyline). Use the identical element, node, and neighbor input files specified in Section 7.1 Mesh file generation. The last input is to specify the watershed outlet boundary condition. Click the **Run** button to generate the river file.

**Important Notes:** Users need to provide and edit shape, material and initial condition information properties. Use the PIHM v2 input file format file documentation to understand expected data structures. A common issue is not using the same mesh input files with both the River and Mesh file generation process.

## 8.4 Soil (\*.soil) File

To create the Soil (\*.soil) data file for PIHM v2.2, a soil texture file is necessary. The soil texture file contains a MUKEY (or unique numbers), silt (%), clay (%), organic matter (%), and bulk density (g/cm<sup>3</sup>). A soil texture file can be created manually (see PIHM v2 input file documentation for data structure) or downloaded using the HydroTerre software services. The soil texture file is transformed using pedo-transfer functions to create the soil data file.

The screenshot shows a software dialog box titled "P Soil Data File". It has a standard Windows-style title bar with a close button (X). The dialog is divided into two main sections. The first section, labeled "Input", has a sub-label "Soil Texture" and a "Browse" button. Below this is an empty text input field. The second section, labeled "Output", has a sub-label "Soil Data File" and a "Browse" button. Below this is another empty text input field. At the bottom of the dialog, there are three buttons: "Help", "Close", and "Run". The "Run" button is highlighted with a blue border. Below the buttons is a large, empty rectangular area, likely for displaying output or logs, with a horizontal scrollbar at the bottom.

**Figure 8.4: Soil File Generation Dialog**

In the input section of the dialog (**Figure 8.4**), specify the soil texture file. In the output section, specify the soil data file output. Click the **Run** button to generate the soil file.

**Important Notes:** Users need to validate that the soil classes (specified as raster dataset in Section 8.2) matches the soil texture rows.

## 8.5 Geology (\*.geol) File

To create the Geology (\*.geol) data file for PIHM v2.2, a geology texture file is necessary. The geology texture file contains a MUKEY (or unique numbers), silt (%), clay (%), organic matter (%), and bulk density (g/cm<sup>3</sup>). A geology texture file can be created manually (see PIHM v2 input file documentation for data structure) or downloaded using the HydroTerre software services. The geology texture file is transformed using pedo-transfer functions to create the geology data file.

The screenshot shows a software dialog box titled "P Geol Data File". It features a blue header bar with a close button (X) in the top right corner. The dialog is divided into two main sections: "Input" and "Output". The "Input" section has a label "Geol Texture" and a "Browse" button. Below this is a text input field. The "Output" section has a label "Geol Data File" and a "Browse" button. Below this is another text input field. At the bottom of the dialog, there are three buttons: "Help", "Close", and "Run". The "Run" button is highlighted with a blue border. Below the buttons is a large, empty text area with a horizontal scrollbar at the bottom.

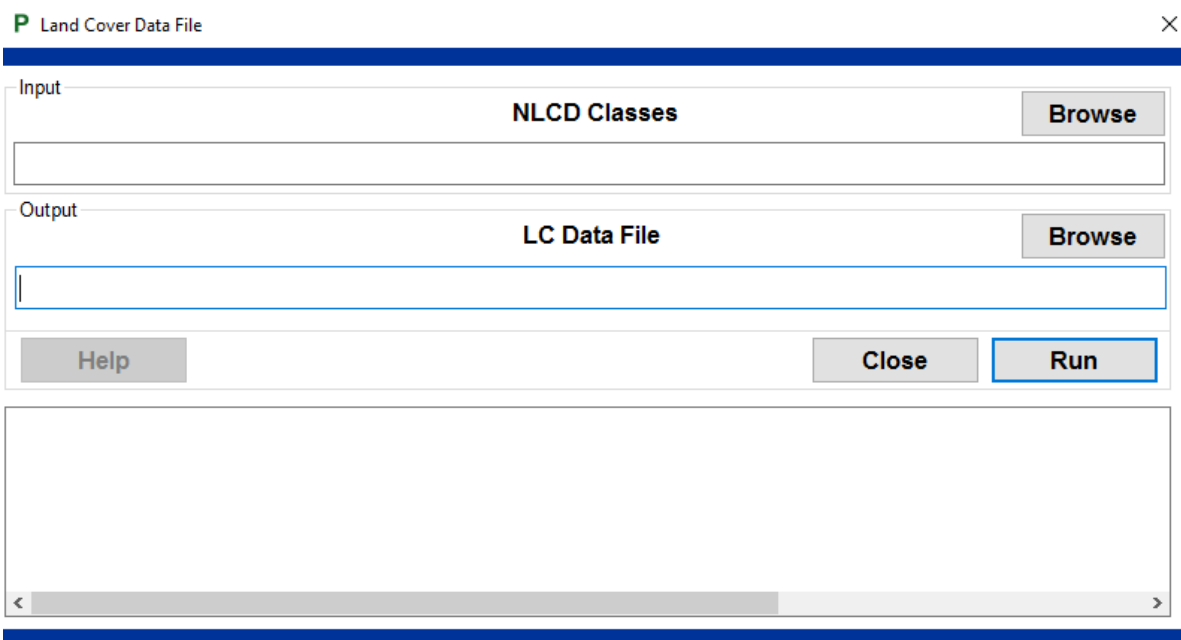
**Figure 8.5: Geology File Generation Dialog**

In the input section of the dialog (**Figure 8.5**), specify the geology texture file. In the output section, specify the geology data file output. Click the **Run** button to generate the geology file.

**Important Notes:** Users need to validate that the geology classes (specified as raster dataset in Section 8.2) matches the geology texture rows.

## 8.6 Land Cover (\*.lc) Data File

To create the Land-Cover (\*.lc) data file for PIHM v2.2, a NLCD classification file is necessary. The NLCD classes file contains a unique list of National Land Cover Dataset classification numbers. A NLCD classes file can be created manually (see PIHM v2 input file documentation for data structure) or downloaded using the HydroTerre software services. The NLCD classes file is transformed using a lookup table to specify the maximum LAI, Rmin, RSref, Albedo, VegFrac and Roughness values.



**Figure 8.6: NLCD File Generation Dialog**

In the input section of the dialog (**Figure 8.6**), specify the NLCD classification file. In the output section, specify the Landcover data file output. Click the **Run** button to generate the Landcover file.

**Important Notes:** Users need to validate that the NLCD classes (specified as raster dataset in Section 8.2) matches the Landcover types.

## 8.7 Initial State Condition (\*.init) Data File

To create the initial state condition variable (\*.init) data file for PIHM v2.2, the mesh (Section 8.1) and river (Section 8.2) data files are required inputs. If Surface, Interception and Snow Storage values (in meters) are known, specify these initial conditions on the top left of the dialog (**Figure 8.7**). To specify Soil Moisture, Groundwater, River and Riverbed initial conditions, these four values can be specified as percentage (default) or as meter values.

The screenshot shows the 'Init Data File' dialog box. It has a title bar with a green 'P' icon and a close button. The dialog is divided into two main sections: 'Input' and 'Output'.  
In the 'Input' section, there are two columns. The left column contains three input fields: 'Interception' (value 0), 'Snow' (value 0), and 'Surface' (value 0). The right column contains two radio buttons: '\* meters' (unselected) and '\* percent' (selected). Below the radio buttons are four input fields: '\*Soil Moisture' (value 40), '\*River' (value 40), '\*Groundwater' (value 40), and '\*Riverbed' (value 40).  
Below the 'Input' section, there are three file selection rows, each with a text field and a 'Browse' button: 'Mesh Data File', 'Riv Data File', and 'Init Data File'.  
At the bottom of the dialog, there are three buttons: 'Help', 'Close', and 'Run'. Below these buttons is a large, empty text area with a scrollbar.

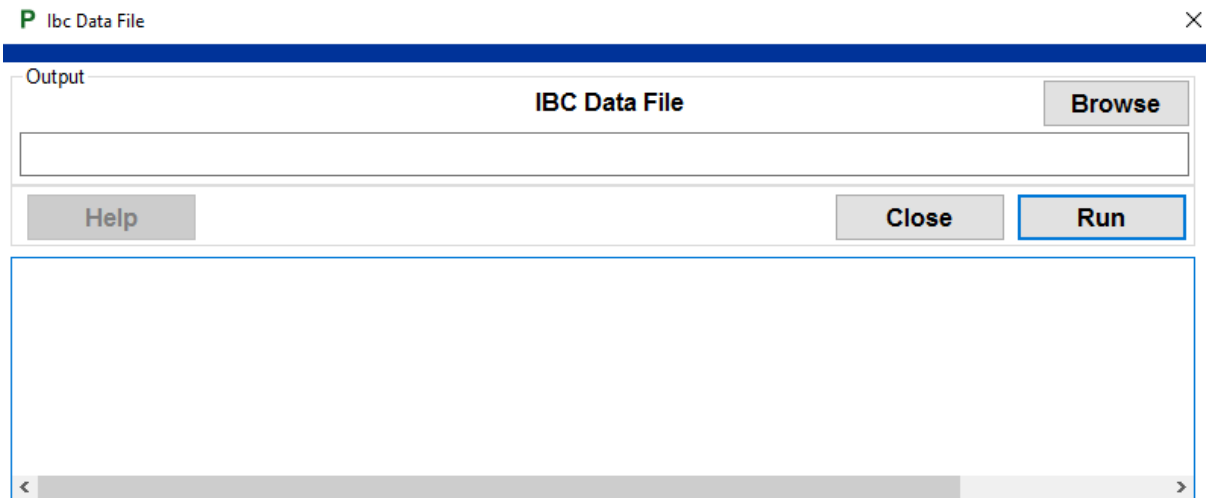
**Figure 8.7: Init Data File Generation Dialog**

In the output section, specify the Init data file output. Click the **Run** button to generate the Init data file.

**Important Notes:** A common issue is not using the same mesh input files with both the River and Mesh file generation process. Initial conditions will be invalid if the number of triangle elements do not match the mesh and river element count.

## 8.8 Initial Boundary Conditions (\*.ibc) Data File

To create a ***default blank*** initial boundary conditions (\*.ibc) data file for PIHM v2.2 mesh elements, specify the output location and then click the **Run** button.



**Figure 8.8: IBC Data File Generation Dialog**

See PIHM v2 input file documentation for IBC data structure to create an IBC data file ***manually*** for your mesh elements.

**Note:** PIHM v2.2 requires an IBC data file. This step is necessary.



## 8.9 Parameter (\*.para) Data File

The parameter data file specifies model output intervals, solver options, and process parameters. For new PIHM users, the default settings are appropriate to evaluate your PIHM model. It is recommended to examine a short simulation duration, two days or less, to determine if the PIHM solver converges or not, to generate a solution.

**Para Data File**

Model Process Parameters | Model Solver Parameters | **Model Output Parameters**

Start Time: 0 | Stop Time: 2 | Days: Days

Output Intervals in: Days

River	1	Riverbed	1	Interception	1	Infiltration	1
Riv u/d	1	Rivbed u/d	1	Snow	1	Recharge	1
Riv S l/r	1	Rivbed l/r	1	Surface	1	Canopy Evap	1
Riv B l/r	1	Rivbed s	1	Soil Moisture	1	Transpiration	1
				Groundwater	1	Ground Evap	1

Output: Para Data File **Browse**

**Help** **Close** **Run**

**Figure 8.9: Parameter Data File Generation Dialog**

In the output section of the dialog, browse to the location where to save the parameter file. Then click the **Run** button to generate the parameter file.

## 8.10 Calibration (\*.calib) Data File

The calibration data file is used to calibrate PIHM results. The values in the calibration file are used to multiply parameters calculated during simulation steps. For new PIHM users, the default settings (all ones) are appropriate to evaluate whether your PIHM model is converging.

The screenshot shows the 'Calib Data File' dialog box. It has a title bar with a green 'P' icon and a close button. Below the title bar are five tabs: 'Soils', 'Geology', 'Land Cover', 'River', and 'Forcings'. The 'Soils' tab is selected. Inside the 'Soils' tab, there are six input fields arranged in two rows. The first row contains 'Alpha', 'Porosity', and 'Ksat V'. The second row contains 'Beta', 'InfilDepth', and 'MP KsatV'. Each field has a text box with the value '1.000000' and a small up/down arrow icon. Below these fields is an 'Output' section with a text field and a 'Browse' button. At the bottom of the dialog are three buttons: 'Help', 'Close', and 'Run'.

Figure 8.10: Calibration Data File Generation Dialog

**Note:** It is recommended to change the calibration Soils and Geology infiltration values to large numbers. If this does not improve the PIHM converging process, it is likely the mesh and river datasets require refinement.

## 8.11 Forcing (\*.forc) Data File

See PIHM v2 input file documentation for forcing data structure to create a forcing data file **manually** from your climate data sources. The other method is to visit the HydroTerre website to access NLDAS.

## 9. Group [6]:PIHM Simulation

After completing steps in groups 2 to 5, you are **nearly** ready to execute a PIHM simulation. The first step is to copy the files required for PIHM (Section 8) into the **Input Data Folder** location (**Figure 9**). All the files must match the Data Key or **Unique Project Name**. For example, the Project Name is “AAA”, the input files will be AAA.mesh, AAA.att, etc. Users may be frustrated doing this manually, but it is necessary to force users to not mix and match files that will prevent PIHM from working (crashes). For example, if you used an attribute file from an older mesh with 500 triangles, and your new mesh has 1000 triangles, you are missing 500 attribute properties and PIHM will not operate correctly. Use the **Re-Check Inputs** button to check if the necessary input files exist. Once all “Unique Project Name” files exist within the same input data folder location, the **Run** button will be enabled. Click **Run** to execute PIHM. The **progress-bar** keeps track of PIHM progress. The bottom portion of the dialog provides messages (status, debug, error) information from PIHM v2.2. If you decide not to wait for PIHM to finish, click on the **Stop PIHM** button.

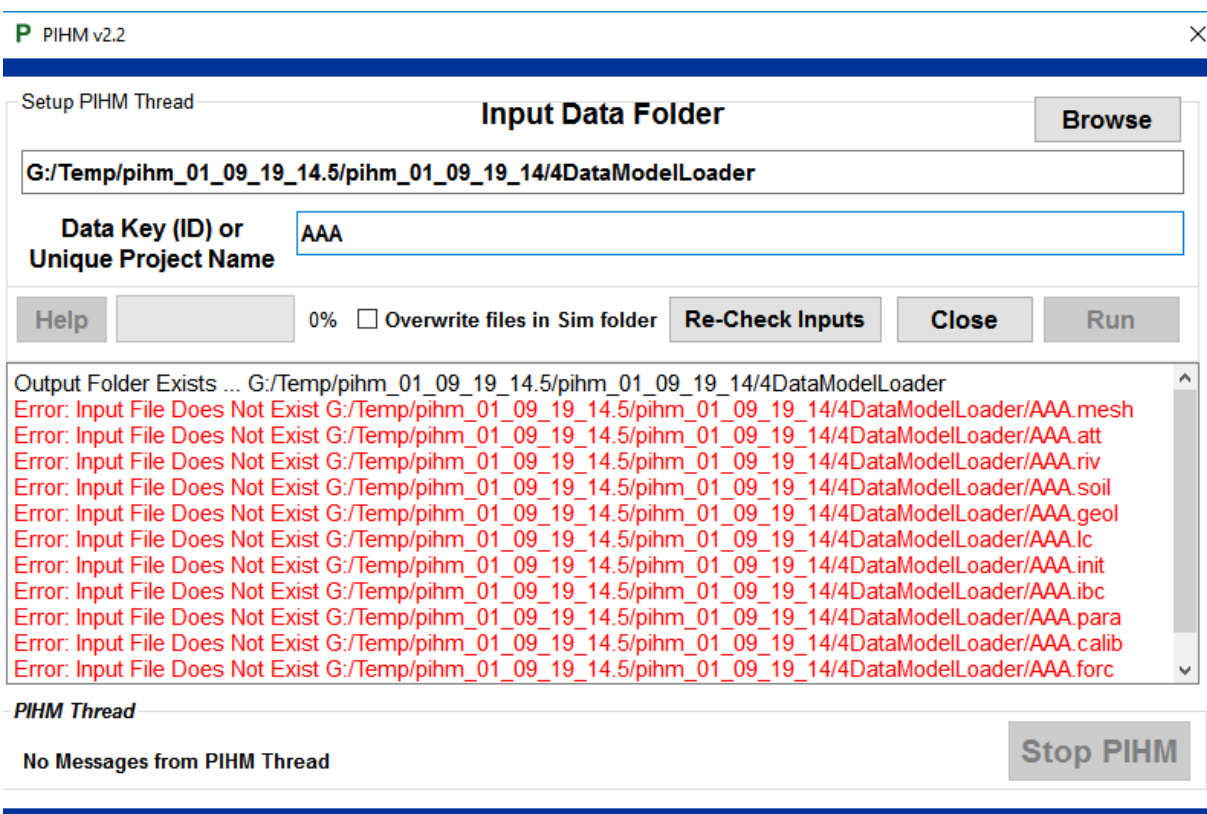


Figure 9: PIHM Dialog

**Note:** It is recommended to use folders with a meaningful name. For example, pihm\_500\_triangles. That way you remember what experiment you were evaluating, to avoid crashing PIHM by using files from other experiments.

## 10. Optional::Hydro Informatics

To help evaluate PIHM results, the Hydro Informatic steps provide **temporal** and **spatial analysis** of several state variables corresponding to the watershed domain.

### 10.1 Temporal Plots

The Time series dialog (**Figure 10.1**) is used for plotting temporal behavior of states or flux corresponding to either triangle elements or river segments. Users can visualize an element or river segment individually or as an average of all the feature elements within the watershed. To create temporal plots, both the data key and output data folder location from **Section 9**, containing the PIHM results need to be specified.

**Figure 10.1: Temporal Series Dialog**

Select Element feature or River Feature depending on the plot variable it corresponds to. **Select Feature by:** allows visualize time series of individual feature element (**ID:** individual element or stream segment) or an **Average** of all the feature elements. It is only required to input the ID value if not using All features has been selected. **Select Plot Variable:** provides options to choose from for the plot variable of interest. Click the **Run** button to begin plotting. The process might take a while for processing the model output file. After successful processing of the model output files, the time series plot is displayed using Qt widget.

## 10.2 Spatial Plots

The spatial plots dialog (**Figure 10.2**) is used for plotting spatial behavior of several state or flux variables corresponding to either elements or river segments. Users can visualize spatial distribution variables as a snapshot of time or as an average over a given period. To create spatial plots, both the data key and output data folder location from **Section 9**, containing the PIHM results need to be specified. As well as the mesh shape file generated from **Group 4: Step 3** Tin Shape Layer.

Mesh Spatial Analysis

Output Data Folder Browse

Data Key (ID)

Plot Variable **Surface Storage**

Time Interval 0 to 2 Minutes into 1

Mesh Shape File Browse

Help Close Run

**Figure 10.2: Spatial Analysis Dialog**

**Plot Variable** provides options to choose from for the plot variable of interest. **Time Interval** specifies the start and finish times, over which it is desired to obtain average spatial plot of the variable. Click the **Run** button to begin plotting. The process might take a while for processing the model output file. After successful processing of the model output files, the time series plot is displayed using Qt widget.

**Note:** The output data folder location is used to save output files from spatial analysis, including a shapefile.

## 11. Debugging PIHMgis

To help identify problems while using PIHMgis, the level of error detail shown by PIHMgis in the log files is controlled within the Log menu (**Figure 11.1**). The **default** setting is to show Important messages only to the console and log text files. Users can also specify **debug** messages to track location of the algorithms and interfaces. While **many** messages are used to show messages from recursive functions. Log files can be kept using the save button.

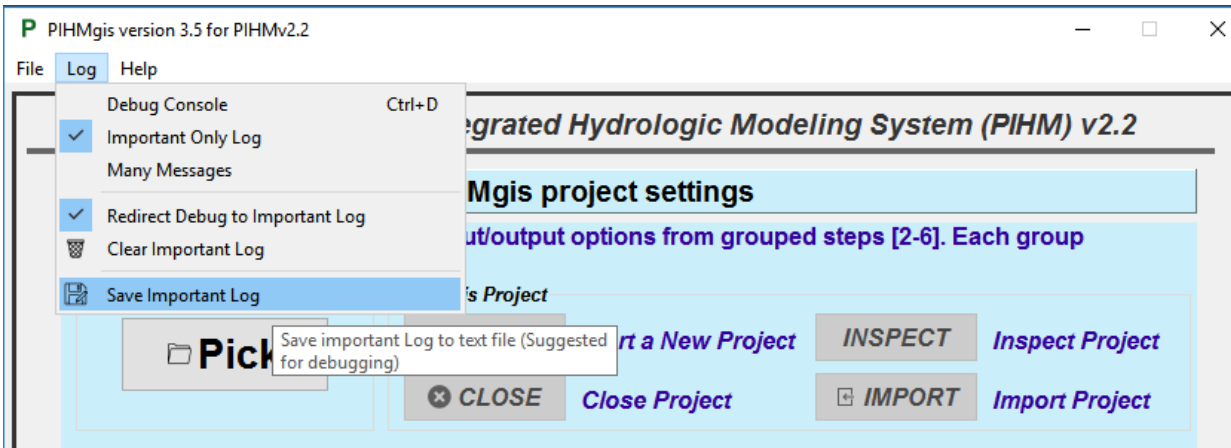


Figure 11.1: Log Menu Options

For situations where PIHMgis crashes (i.e. you can't identify where the crashed happened via the interface), or when you need to trace long messages in your favorite text editor, the autolog.txt file is automatically generated in the **.PIHMgis** folder in your workspace location.