

PIHMgis: A *“Tightly Coupled”* GIS Framework for PIHM

Version: 1.0

Users Guide



July 10, 2007

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

Table of Content

1.	Introduction	1
2.	Initial Setup	1
3.	Understanding PIHMgis Framework	1
4.	Raster Processing	4
4.1	Fill Pits	4
4.2	Flow Grid	5
4.3	Stream Grid	7
4.4	Link Grid	9
4.5	Stream Polyline	11
4.6	Catchment Grid	12
4.7	Catchment Polygon	14
5.	Vector Processing	16
5.1	Polygon to Line	16
5.2	Simplify Line	18
5.3	Split Line	21
5.4	Vector Merge	22
6.	Domain Decomposition	24
6.1	Read ShapeTopology	24
6.2	Run TRIANGLE	25
6.3	TIN Generation	27
7.	DataModel Loader	29
7.1	Mesh File	29
7.2	Att File	30
7.3	Riv File	32
7.4	Para File	34
8.	Run PIHM	36
9.	Analysis	37
9.1	Time Series Plots	37
9.2	Spatial Plots	38

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 non-dynamic data-interaction with pluggable software components. PIHMgis is an open-source, platform independent, extensible and “tightly-coupled” integrated GIS interface to Penn State Integrated Hydrologic Model (PIHM). 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. It can be downloaded from the PIHMgis website [<http://cataract01.cee.psu.edu/PIHMgis/>] under installation menu. You need to select the operating system on intend to use it for downloading the relevant files. The detailed installation instruction can also be found on the same page.

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 fed into the model (PIHM). Data Analysis allows easier visualization of model results.

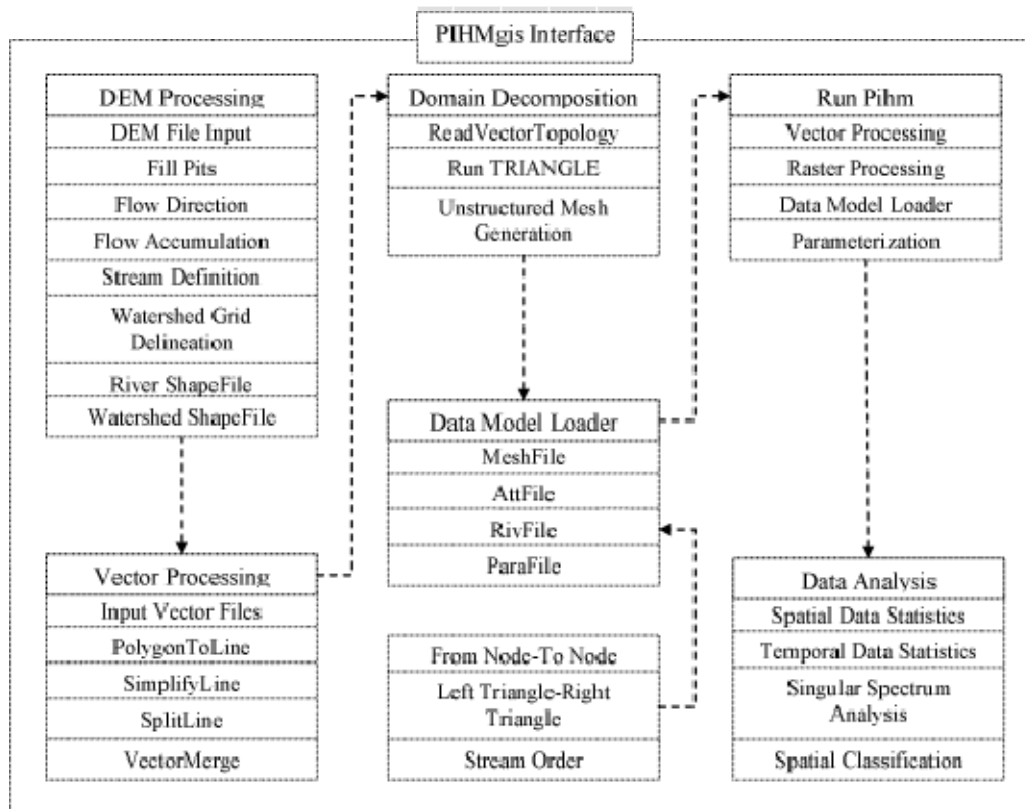


Figure 3.1: PIHMgis Procedural Framework

The PIHMgis interface has been shown in the Figure 3.2. PIHMgis uses QuantumGIS (QGIS) as base GIS framework. If the PIHMgis tool doesn't show up automatically it should be loaded manually by selecting **Plugins >> Plugin Manager** and then checking the box next to PIHMgis. Press OK to lead the PIHMgis plugin.

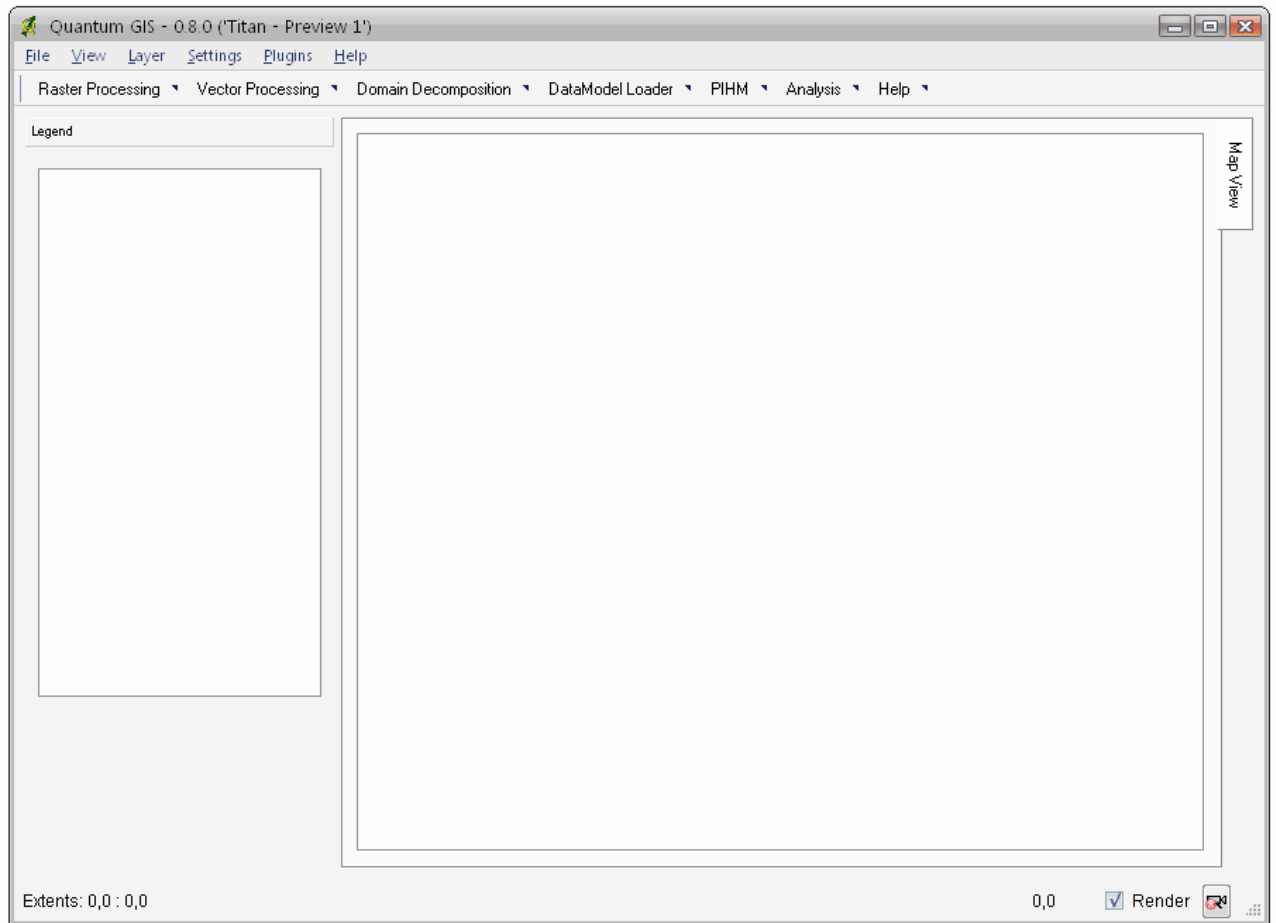


Figure 3.2: PIHMgis Interface

4. Raster Processing

Raster processing facilitates subshed/watershed delineation and stream definition from the Digital Elevation Model (DEM) of any region.

Any raster dataset can be loaded into the Qgis framework. This can be done by selecting **Layers >> Add Raster Layer** from the top menu. In order to successfully complete raster processing one needs to step through seven (7) processes. Those steps are accomplished by performing steps described in the following Raster Processing menu:



4.1 Fill Pits

Fill Pits fills pits in a grid. If a cell is surrounded by higher elevation cells, the water is trapped in that cell and cannot flow. They 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. (TAUDEM, accessed 2006).



Select **Fill Pits** from the **Raster Processing** drop down menu. This should bring Fill Pits dialog [Figure 4.1] on the screen.

In the Input section of the dialog browse to the raw DEM file. It could be either an ESRI binary (*.adf) or Arc/Info ascii file (*.asc). In the Output section browse a file name to which the pit-filled grid could be saved. At this point the module is ready for processing. Select **Run** to start processing. Depending on the size and resolution of DEM the processing could take several minutes. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the Pit-Filled Grid will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

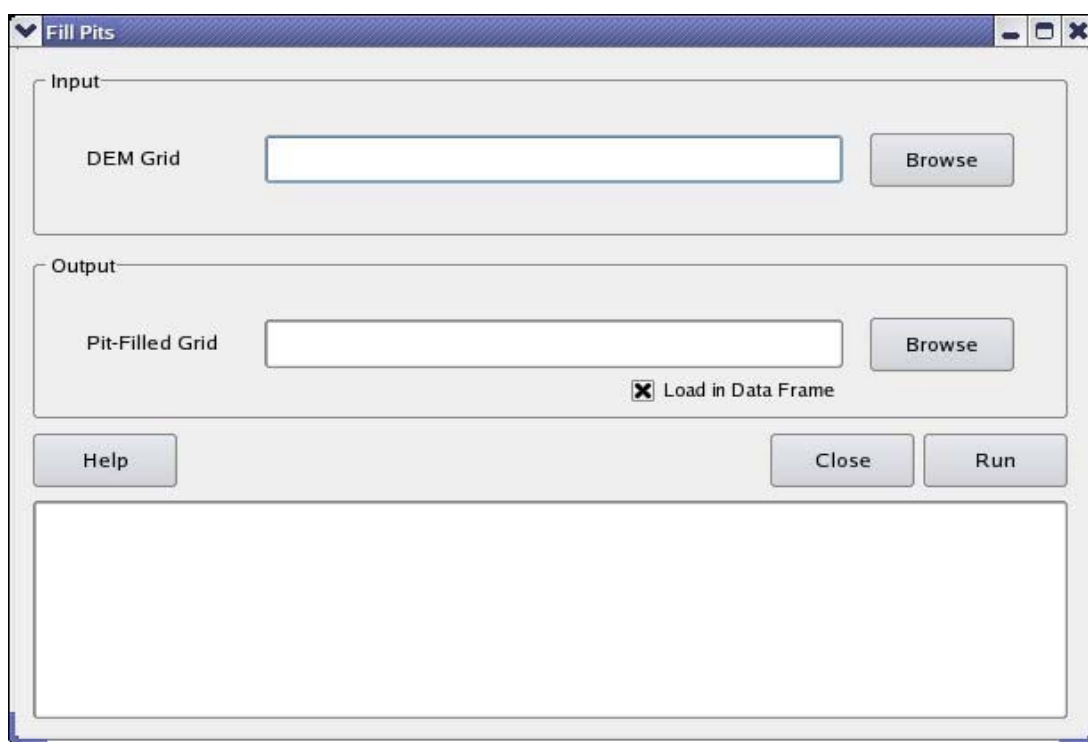


Figure 4.1: Fill Pits Dialog

4.2 Flow Grid

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 4.2 shows the Flow direction and Flow accumulation calculations performed on a synthetic DEM grid.

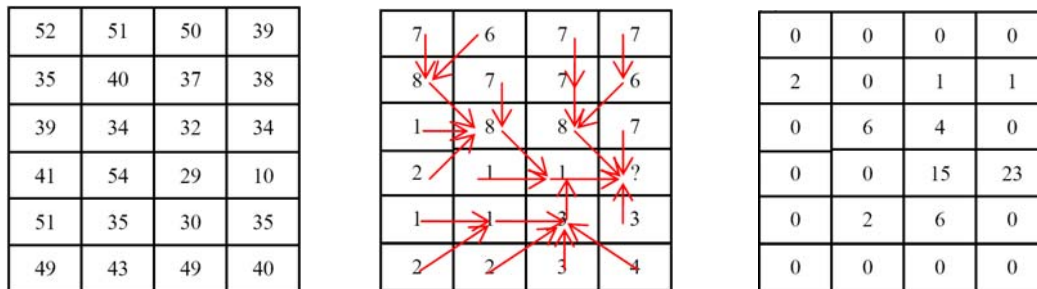


Figure 4.2: Flow direction and Flow accumulation for synthetic grid.



Select **Flow Grid** from the **Raster Processing** drop down menu. This should bring Fill Pits dialog [Figure 4.3] on the screen.

In the Input section of the dialog browse the Pit Filled Grid generated by the step 4.1. In the Output section browse the file names to which the flow direction and flow accumulation grid could be saved. At this point the module is ready to run; select **Run** to start processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the Flow Direction Grid and Flow Accumulation Grid will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.



Figure 4.3: Flow Grid Dialog

4.3 Stream Grid

Stream Grid is a raster equivalent of the stream network. Those Flow Accumulation Grid 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 particular cell is greater than the given value should be classified as a stream. Rest of the grid assumes No Data Value.

Figure 4.4 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 4.4: Stream Grid for the synthetic grid.



Select **Stream Grid** from the **Raster Processing** drop down menu. This should bring Stream Grid dialog [Figure 4.5] on the screen.

In the Input section of the dialog browse the Flow Accumulation Grid generated by the step 4.2. In the Output section of the dialog browse the file name to which the stream grid could be saved. Also it is required to provide an “integer” value for the threshold. Any Flow Accumulation Grid with having value greater than threshold will be classified as a stream grid. At this point module is ready to run; select **Run** to begin processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the Stream Grid will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

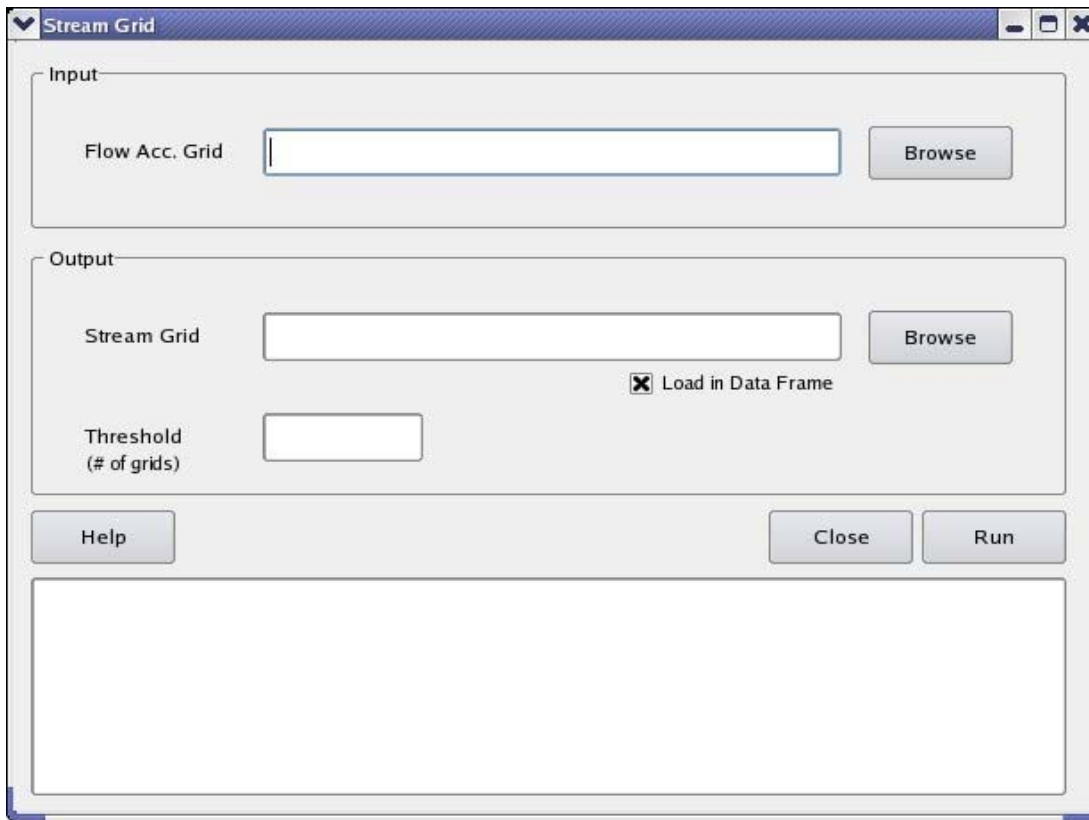
The image shows a software dialog box titled "Stream Grid". It has a standard Windows-style title bar with a minimize button, a maximize button, and a close button. The dialog is divided into two main sections: "Input" and "Output". In the "Input" section, there is a label "Flow Acc. Grid" followed by a text input field and a "Browse" button. In the "Output" section, there is a label "Stream Grid" followed by a text input field and a "Browse" button. Below the "Stream Grid" input, there is a checkbox labeled "Load in Data Frame" which is checked. Below the checkbox, there is a label "Threshold (# of grids)" followed by a text input field. At the bottom of the dialog, there are three buttons: "Help", "Close", and "Run". A large empty rectangular area is located at the very bottom of the dialog box.

Figure 4.5: Stream Grid Dialog

4.4 Link Grid

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 similar to that of Stream Grid.

Figure 4.6 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 4.6: Link Grid for the synthetic grid.



Select **Link Grid** from the **Raster Processing** drop down menu. This should bring Link Grid dialog [Figure 4.7] on the screen.

In the Input section of the dialog browse the Stream Grid and Flow Direction Grid generated by the step 4.3 and 4.2 respectively. In the Output section of the dialog browse the file name to which the Link Grid could be saved. At this point module is ready to run; select **Run** to begin processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the Link Grid will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

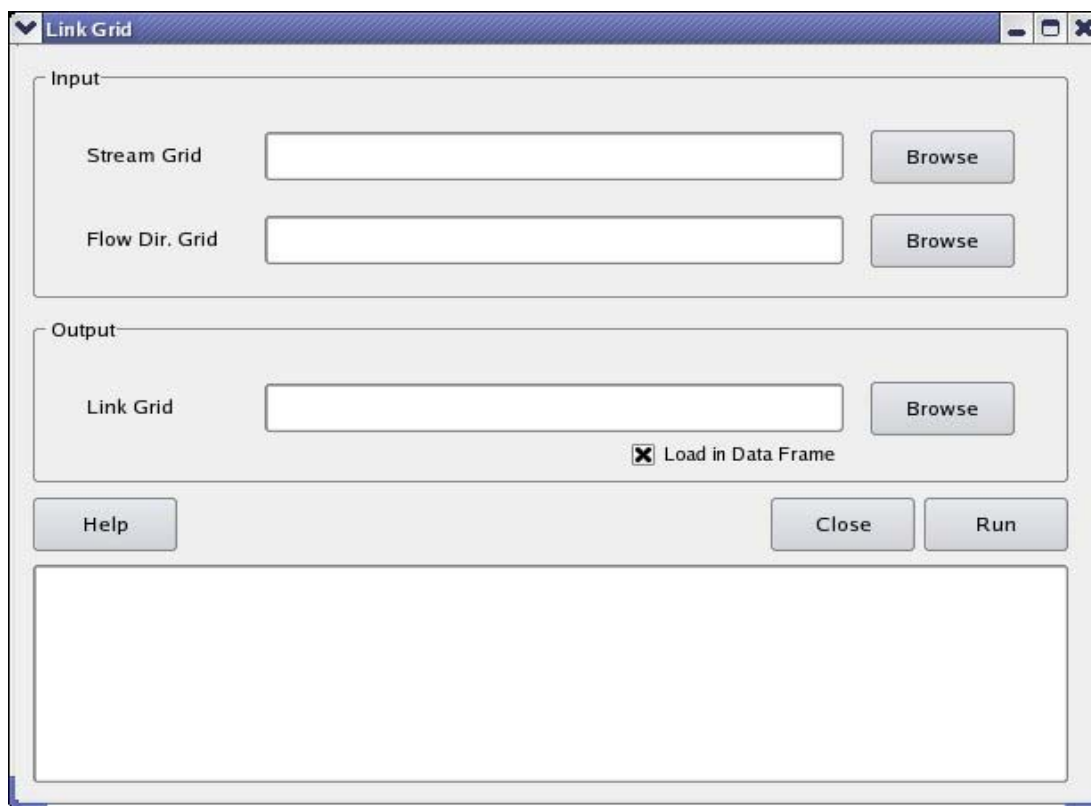


Figure 4.7: Link Grid Dialog

4.5 Stream Polyline

Stream polyline 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 4.8 shows the stream polyline obtained corresponding to the link grid and flow direction grid obtained in the previous section for the synthetic grid.

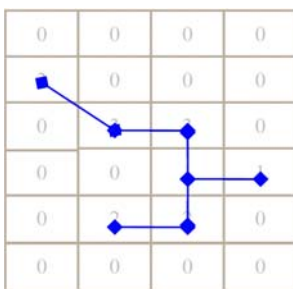


Figure 4.8: Stream Polyline for the synthetic grid.



Select **Stream Polyline** from the **Raster Processing** drop down menu. This should bring Stream PolyLine dialog [Figure 4.9] on the screen.

In the Input section of the dialog browse the Link Grid and Flow Direction Grid generated by the step 4.4 and 4.2 respectively. In the Output section of the dialog browse the shape file name to which the Stream Polyline could be saved. At this point module is ready to run; select **Run** to begin processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the generated Stream Lines will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

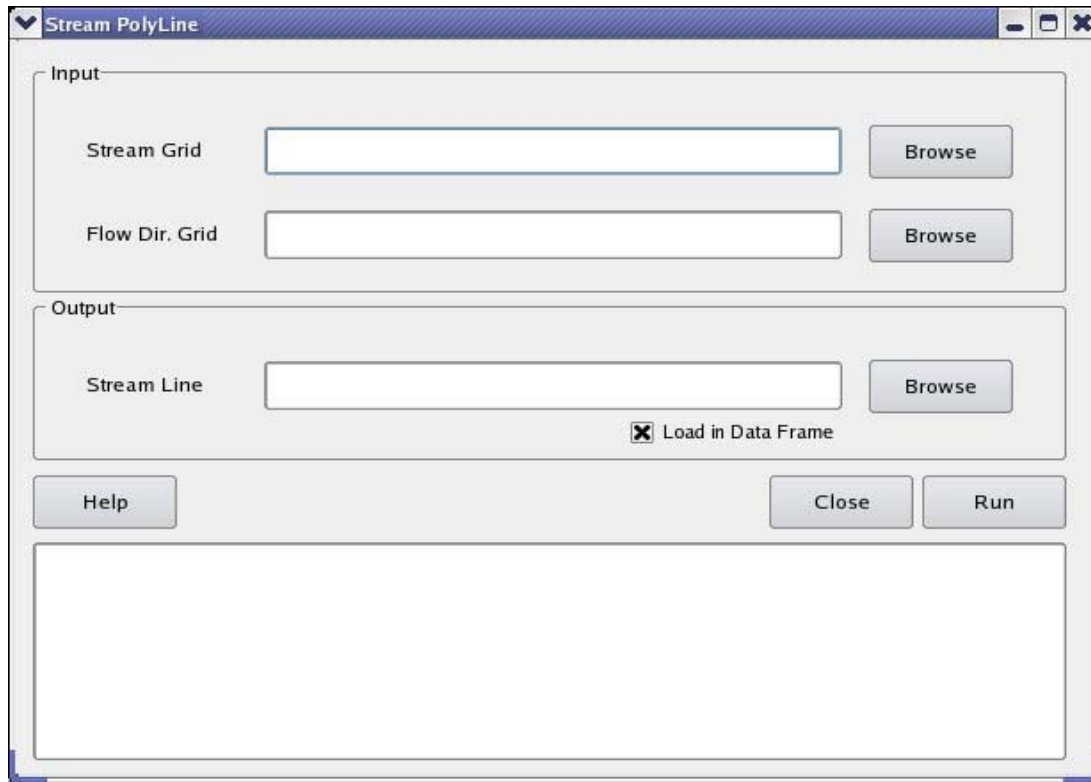


Figure 4.9: Stream Polyline Dialog

4.6 Catchment Grid

All the grids draining to a particular 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 4.10 shows the catchment grid obtained for the synthetic grid using Stream and Flow direction grid discussed in section 4.3 and 4.2 respectively. Different colors are used for clear representation of the catchment grid.

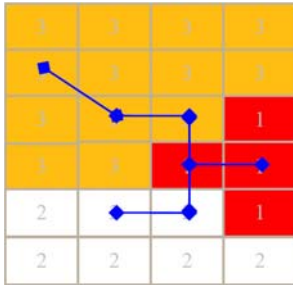


Figure 4.10: Catchment grid for the synthetic grid.



Select **Catchment Grid** from the **Raster Processing** drop down menu. This should bring Catchment Grid dialog [Figure 4.11] on the screen.

In the Input section of the dialog browse the Link Grid and Flow Direction Grid generated by the step 4.4 and 4.2 respectively. In the Output section of the dialog browse the file name to which the Catchment Grid could be saved. At this point the module is ready to run; select **Run** to begin processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the generated Catchment Grid will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

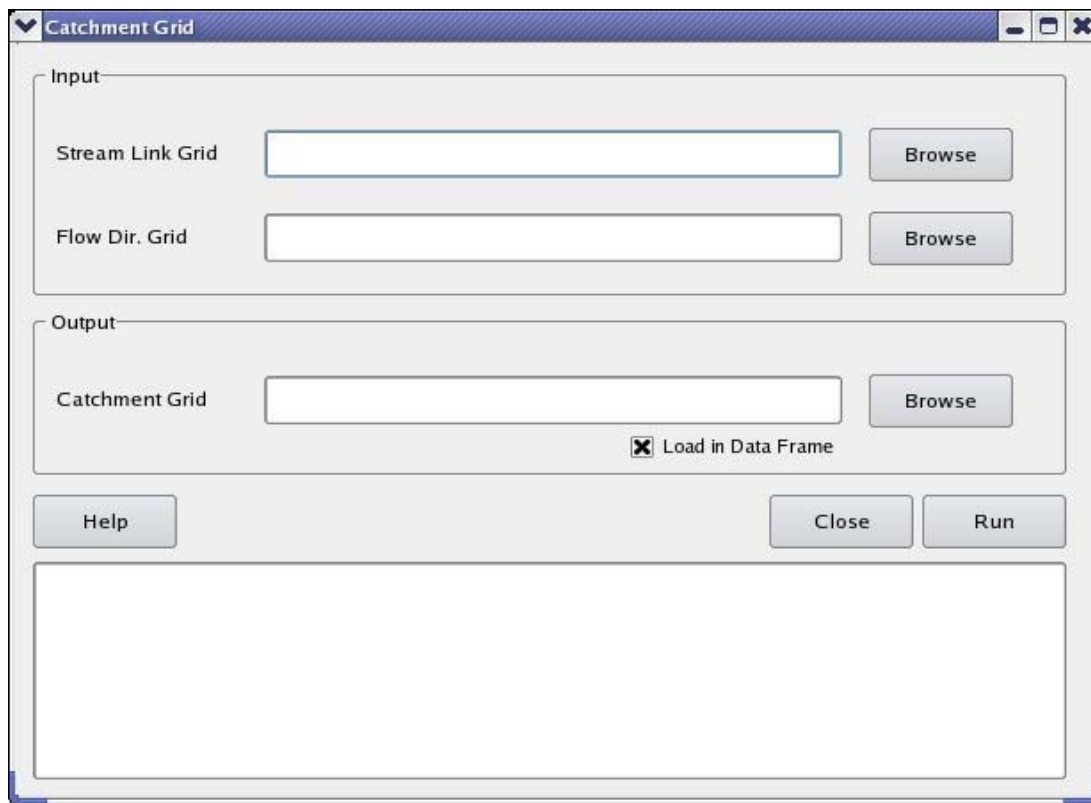


Figure 4.11: Catchment Grid Dialog

4.7 Catchment Polygon

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 4.12 shows the catchment polygon obtained corresponding to the catchment grid obtained in the section 4.6 for the synthetic grid.

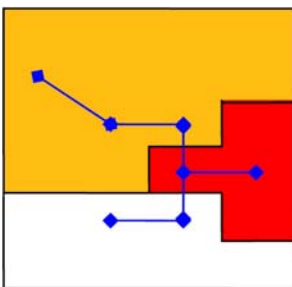


Figure 4.12: Catchment Polygon for the synthetic grid.



Select **Catchment Polygon** from the **Raster Processing** drop down menu. This should bring Catchment Polygon dialog [Figure 4.13] on the screen.

In the Input section of the dialog browse the Catchment Grid generated by the step 4.6. In the Output section of the dialog browse the file name to which the Catchment Polygon could be saved. At this point module is ready to run; select **Run** to begin processing. Please be patient until it completes processing. The text browser at the bottom of the dialog should provide information related to any error or processing. If the Load in Data Frame is checked the generated Catchment Polygon will be automatically loaded in the Qgis window. After the processing is complete you need to press **Close** to proceed to next step.

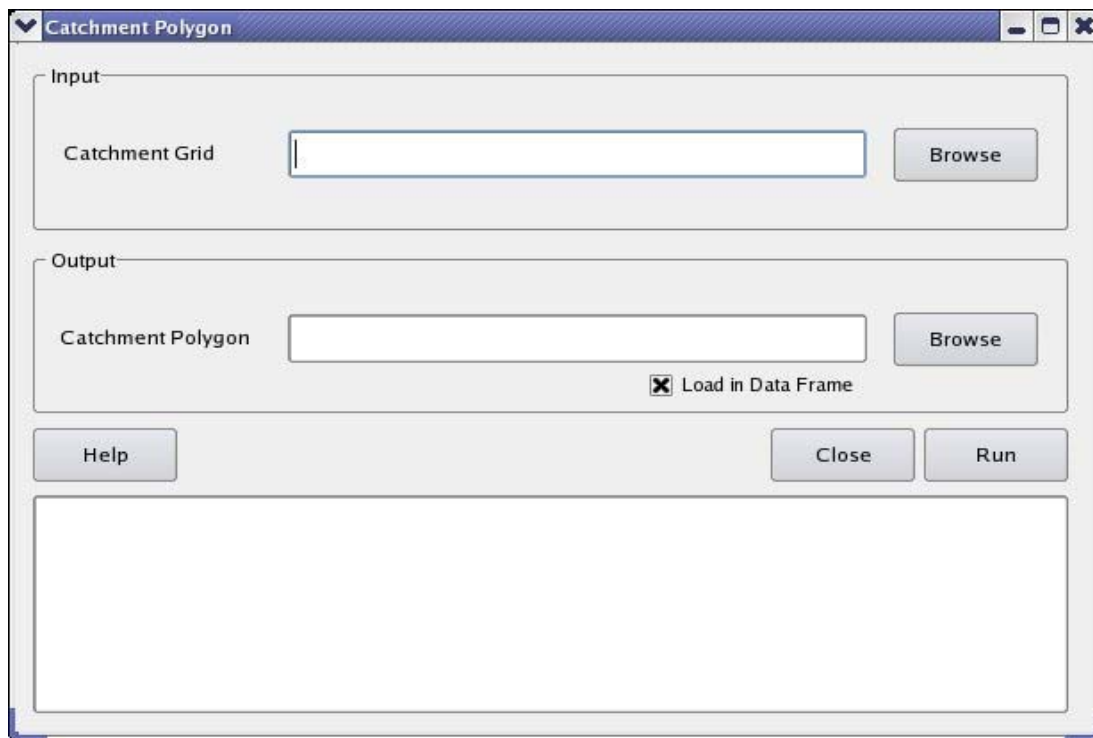
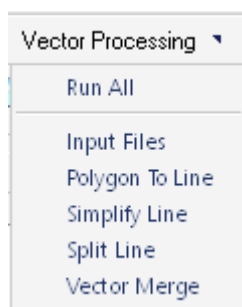


Figure 4.13: Catchment Polygon Dialog

5. Vector Processing

Vector processing consists of a set of operation which enables efficient discretization of the modeling domain. Stream polyline and Catchment polygon generated in section 4.5 and 4.7 can be used primarily for this purpose. However, other hydrologic constraint such as soil coverage, land cover type coverage can also be incorporated. Eventually, it prepares a GIS layer which is used as input constraint for the purpose of domain decomposition of the domain.

These steps are accomplished by performing steps described in the following Vector Processing menu:



5.1 Polygon To Line

The geo-data feature can exist as point, line or polygon objects. In order to merge all the data features together before it can be used by a domain decomposition code, the object properties for all the features should be same. So polygons like lake feature and subsheds are converted to polylines before they can be merged with already existing line features like river. Figure 5.1 shows a schematic of the steps involved in the algorithm.

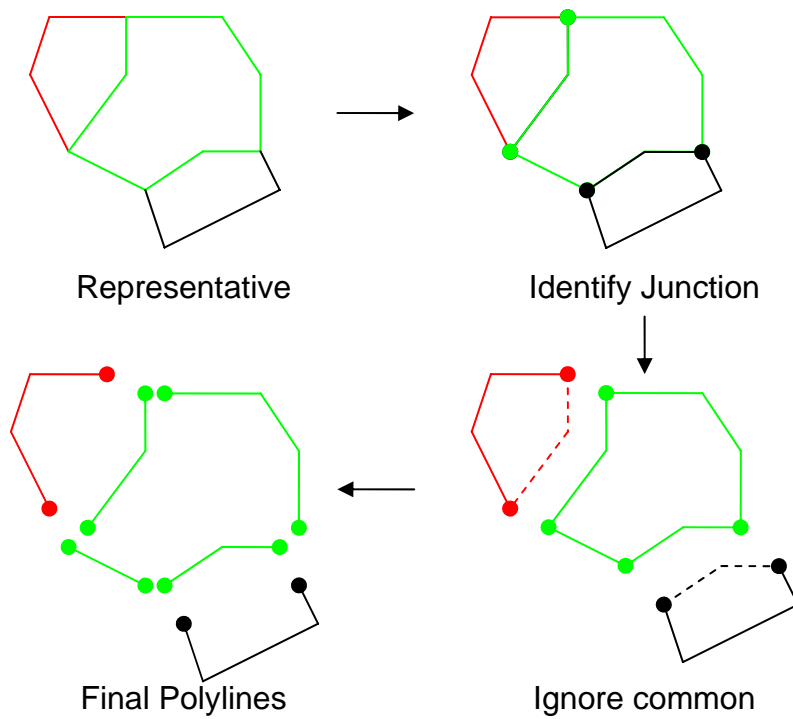
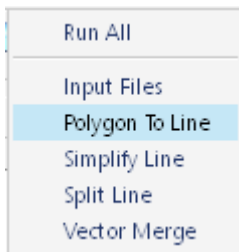


Figure 5.1: Steps for Polygon to Polyline



Select ***Polygon To Line*** from the ***Vector Processing*** drop down menu. This should bring Polygon To Line dialog [Figure 5.2] on the screen.

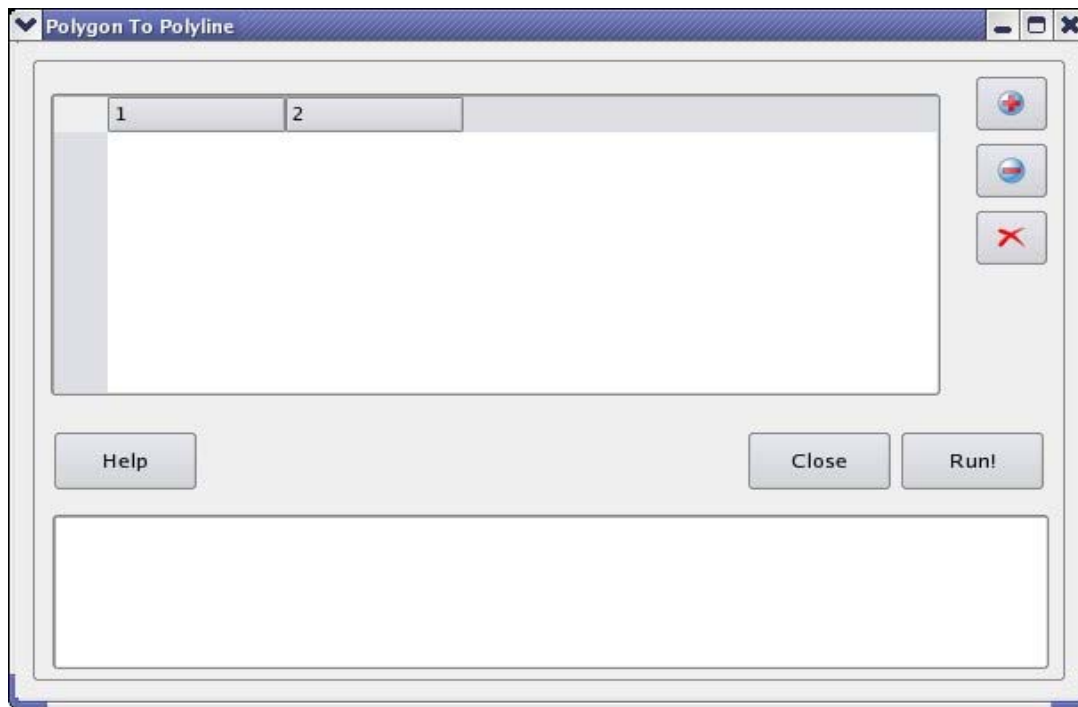





Figure 5.2: Polygon To Polyline Dialog

First of all add shape file(s) of “polygon” type using the  button on the top right of the dialog. A default output file name for the corresponding polyline file is generated which is editable if desired.  button can be used to remove any file from the simplification routine by selecting the file from the left bar of the spreadsheet.  button clears everything from the dialog setting it afresh. After adding the polygon files module is ready to execute using **Run** button. The text browser provides information regarding any error or processing. After the operation is complete for all the files it is required to **Close** the dialog to processed to next step.

5.2 Simplify Line

Simply Line 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 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. Needless to say, larger is the number of decomposed triangle elements, larger will be the computational requirement of the numerical model simulation. Figure 5.3 shows intermediate steps in polyline simplification using Douglas-Peucker algorithm.

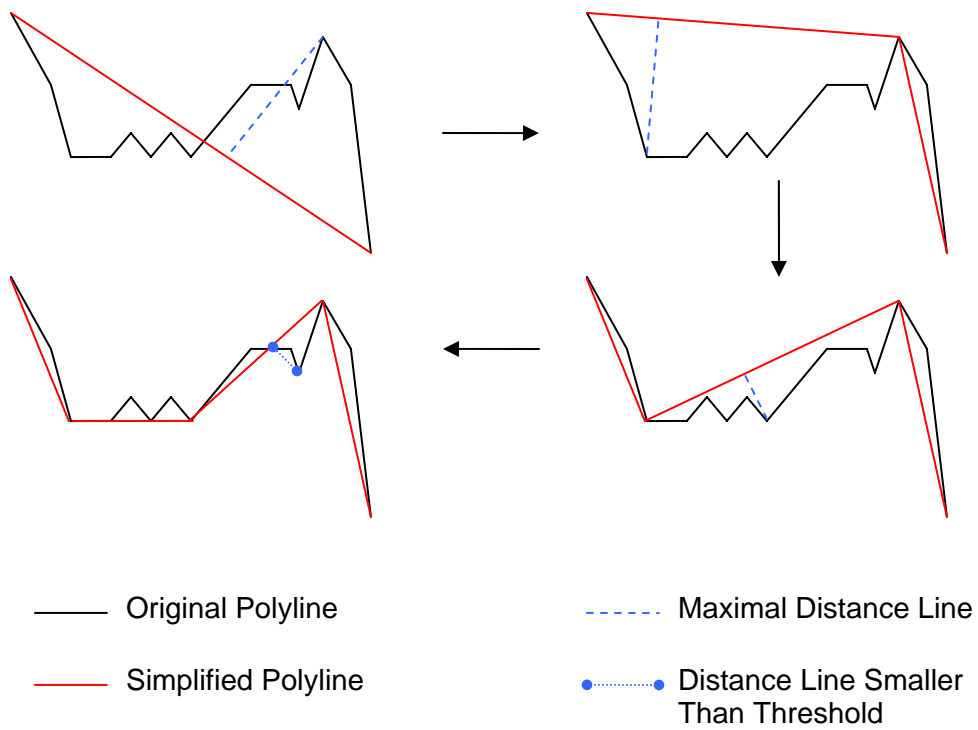
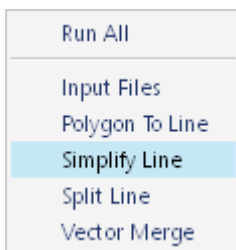


Figure 5.3: Algorithm for Simplification



Select ***Simplify Line*** from the ***Vector Processing*** drop down menu. This should bring Simplify Line dialog [Figure 5.4] on the screen.

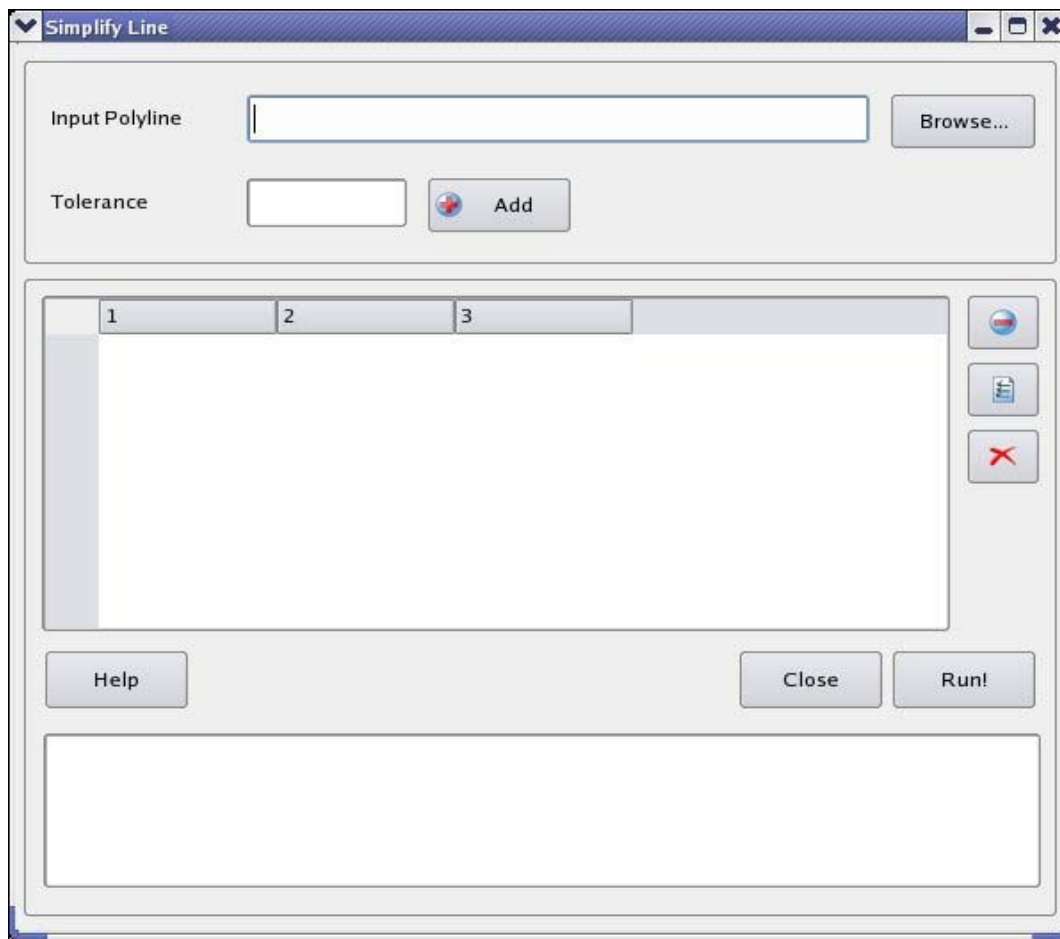






Figure 5.4: Simplify Line Dialog

First of all browse shape file(s) of “polyline” type. A tolerance (double type) should be provided which acts as the maximum distance value for the simplification algorithm. After that  button allows to add the file and the specified tolerance to the project. A default output file name for the corresponding simplified polyline file is generated which is editable if desired. The tolerance value is also editable.  button can be used to remove any file(s) from the simplification routine by selecting the file(s) from the left bar of the spread sheet.  button clears everything from the dialog setting it afresh.  button is provided for simplicity in editing the threshold value for a bunch of files at a time. After adding the polyline files module is ready to execute using **Run** button. The text browser provides information regarding any error or processing. After the operation is complete for all the files it is required to **Close** the dialog to processed to next step.

5.3 Split Line

Before merging all the features together it is necessary to have them all in one feature type. Split line 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 5.5 schematically describes split line.

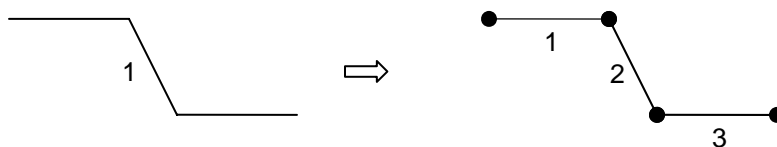
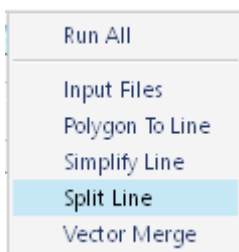





Figure 5.5: Split Line



Select ***Split Line*** from the ***Vector Processing*** drop down menu. This should bring Split Line dialog [Figure 5.6] on the screen.

First of all to add shape file(s) of “polyline” type (simplified polylines obtained in the section 5.2) using the  button. A default output file name for the corresponding splitted line feature file is generated for each input file. The output file name is editable if desired.  button can be used to remove any file(s) from the split line routine by selecting the file(s) from the left bar of the spread sheet.  button clears everything from the dialog setting it afresh. After adding the polyline files module is ready execute using ***Run*** button. The text browser provides information regarding any error or processing. After the operation is complete for all the files it is required to ***Close*** the dialog to processed to next step.

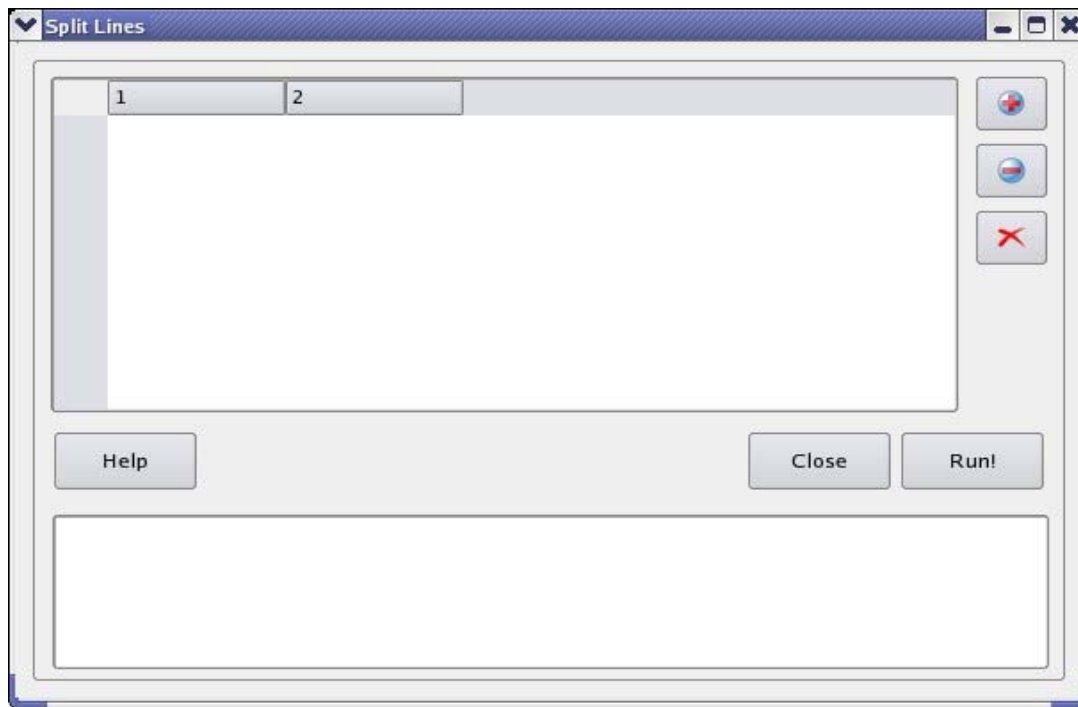
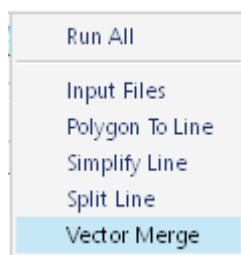




Figure 5.6: Split Line Dialog


5.4 Vector Merge

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 domain decomposition process.



Select **Vector Merge** from the **Vector Processing** drop down menu. This should bring Vector Merge dialog [Figure 5.7] on the screen.

First of all add shape file(s) those need to be merged, in the input section using the  button. User should be careful to provide only line features as input. In the output section browse a output shape file name to which the merged features will be saved. Cluster tolerance is a double type number. Any two nodes separated by the distance less than the cluster tolerance will be merged into one.  button

can be used to remove any file(s) from the routine by selecting the file(s) from the left bar of the spread sheet.  button clears everything from the dialog setting it afresh. After adding the layers module is ready to execute using **Run** button. The text browser provides information regarding any error or processing. After the operation is complete for all the files it is required to **Close** the dialog to processed to next step.

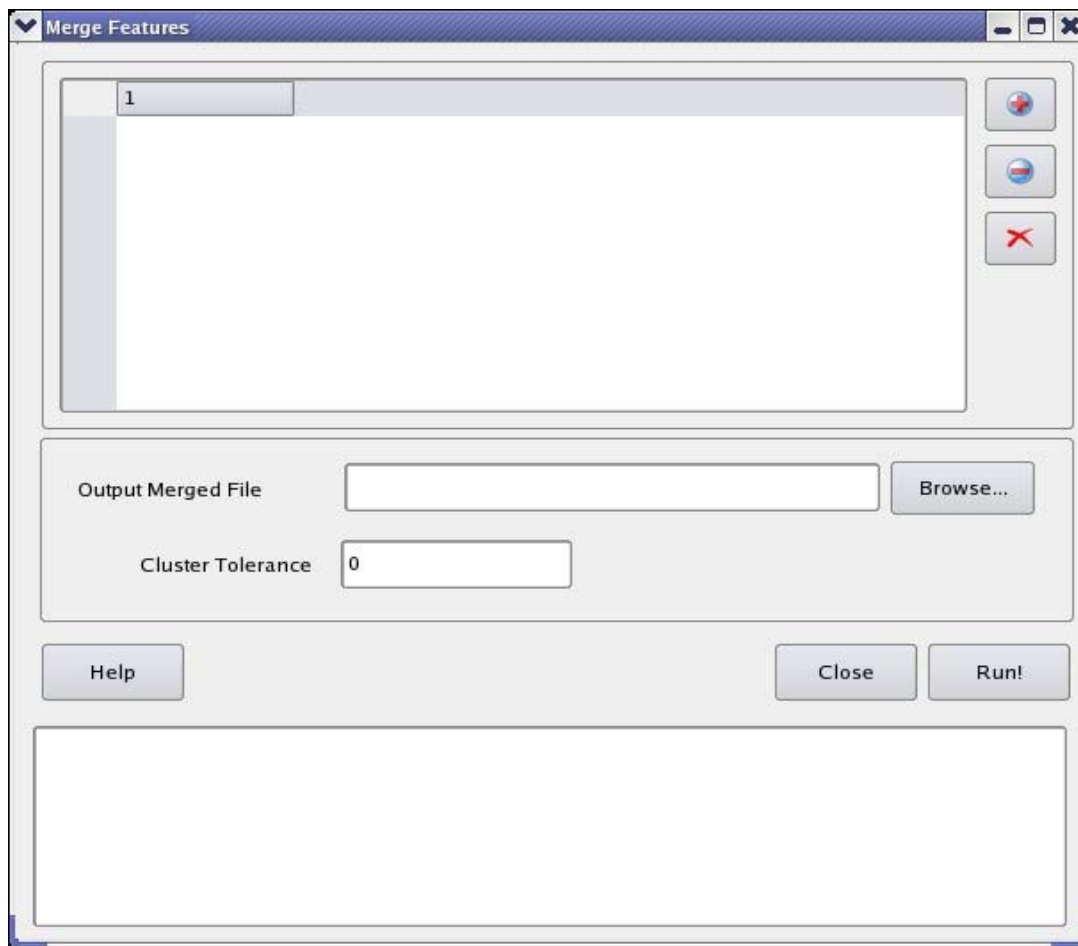
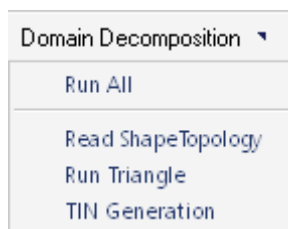


Figure 5.7: Merge Features Dialog

6. Domain Decomposition

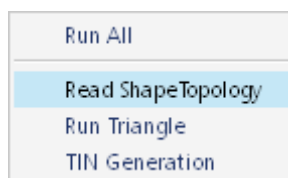
Domain decomposition applies Delaunay Triangulation [Delaunay, 1934] to decompose the modeling domain into triangular irregular mesh or triangular irregular network (TIN). A terrain can be better represented by an irregular mesh if all the critical terrain and hydrographic points are taken into account 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.

Domain Decomposition is performed by the following three steps described in the following Domain Decomposition menu:



6.1 Read ShapeTopology

This step prepares a .poly file which acts as input for running the TRIANGLE in the next section. All the node and line information from the input shape file (vector merged shape file obtained in the section 5.4) is transformed into the poly file.



Select ***Read ShapeTopology*** from the ***Domain Decomposition*** drop down menu. This should bring Read ShapeTopology dialog [Figure 6.1] on the screen.

In the input section of the dialog browse the shape file (merged shape file obtained in the section 5.4) which will act as constraining layer for domain decomposition. In the output section browse a output file name for the poly file. After this, the module is ready to execute using the ***Run*** button. The text browser provides information regarding any error or processing. After the operation is

complete for all the files it is required to **Close** the dialog to processed to next step.

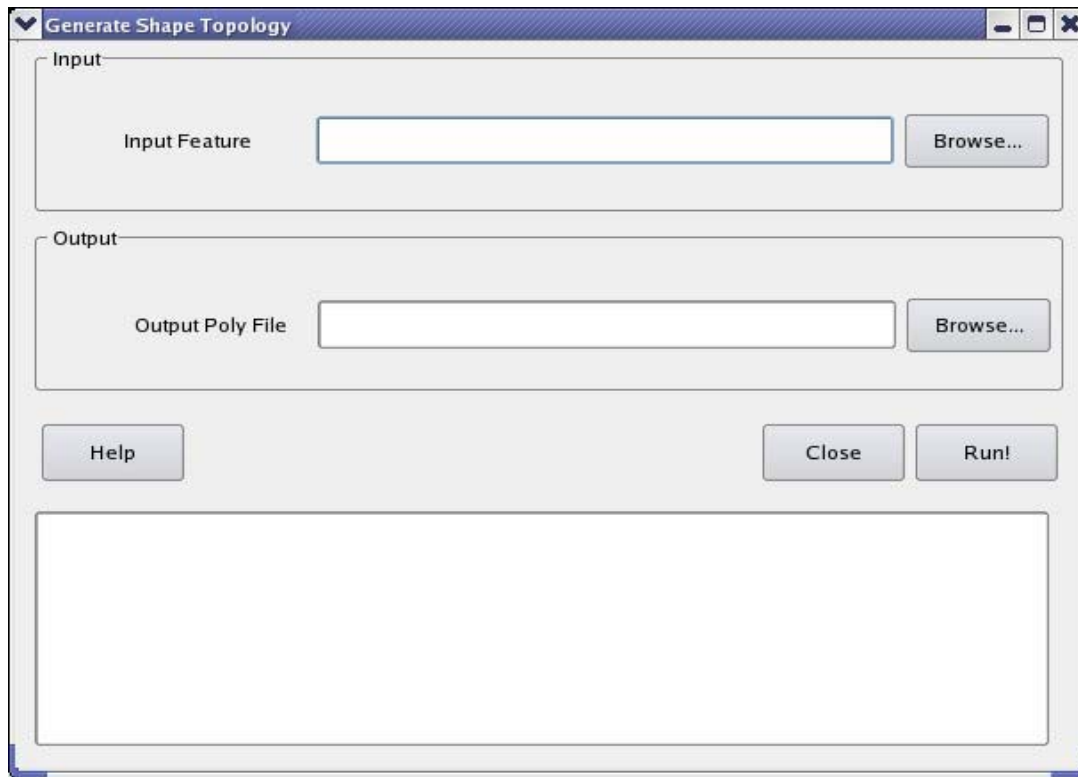


Figure 6.1: Read Shape Topology Dialog

6.2 Run TRIANGLE

'TRIANGLE' developed by Shewchuk [2001] is one of the efficient implementation of this algorithm which provides several flexibility and options to a user. It is also computationally efficient. It takes planner straight line graph (PSLG) as input. The algorithm works in such a way that it refines the Delaunay triangulation by inserting carefully placed vertices until the generated mesh meets a provided quality and size criterion.



Select **Run Triangle** from the **Domain Decomposition** drop down menu. This should bring Run TRIANGLE dialog [Figure 6.2] on the screen.

In the Options section of the dialog there are several options we can provide for the execution of the TRIANGLE. At the same time Other options can be used for the options which are not listed directly on the dialog. The available options and their purpose are listed in the table x.x. In the input poly file section browse the poly file generated in the section 6.1. A click on **Run** executes TRIANGLE. TRIANGLE generates 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. It is advised to note that the value provided as minimum angle and maximum area has tremendous influence on the number of triangles generated.

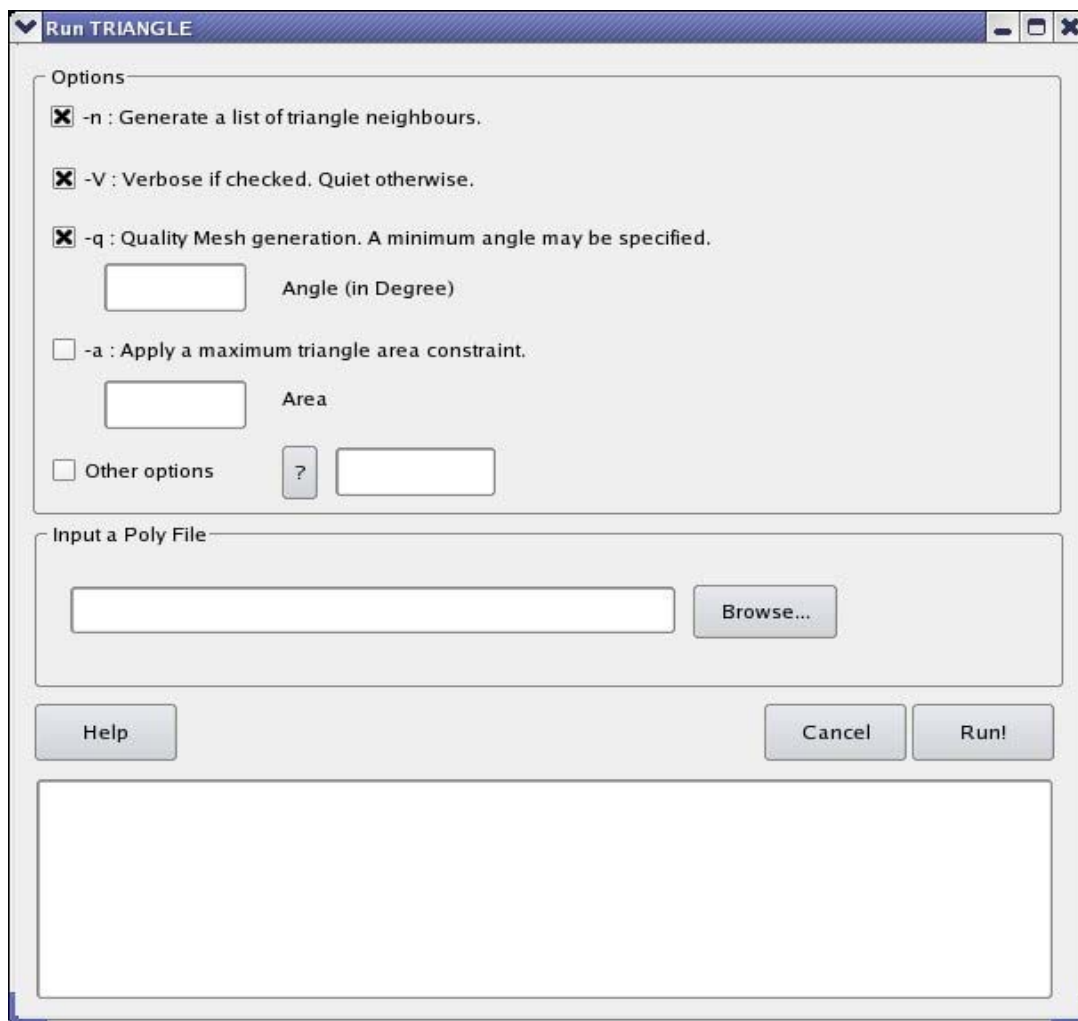


Figure 6.2: Run TRIANGLE Dialog

Table 6.1: 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.
-e	Generates an edge list.
-v	Generates a Voronoi diagram.
-n	Generates a list of triangle neighbors.
-g	Generates an .off file for Geomview.
-B	Suppresses output of boundary information.
-P	Suppresses output of .poly file.
-N	Suppresses output of .node file.
-E	Suppresses output of .ele file.
-I	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.
-L	Uses equatorial circles, not equatorial lenses.
-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).
-L	Uses Ruppert's diametral spheres, not diametral lenses.
-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.

6.3 TIN Generation

This step reads the .ele and .node file produced by the TRIANGLE and generates a shapefile with all the triangle elements.



Select ***TIN Generation*** from the ***Domain Decomposition*** drop down menu. This should bring Unstructured Mesh Generation dialog [Figure 6.3] on the screen.

In the input section of the dialog, browse input files (.ele file and .node file) generated by the TRIANGLE in the section 6.2. In the output section browse an output file name where the unstructured mesh shape file could be saved. Click on the ***Run*** executes the routine. Any error or progress information is displayed in the text browser at the bottom of the dialog.

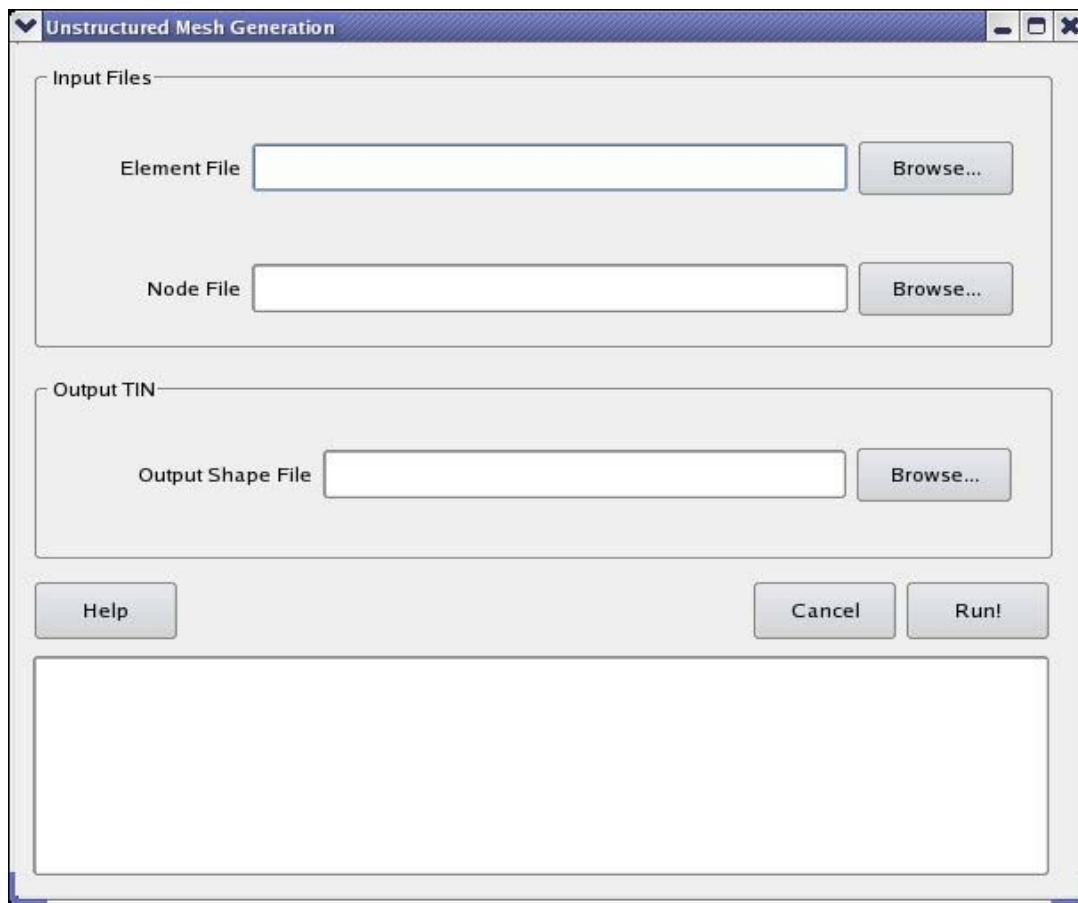
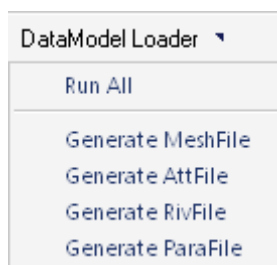


Figure 6.3: Unstructured Mesh Generation Dialog

7. DataModel Loader

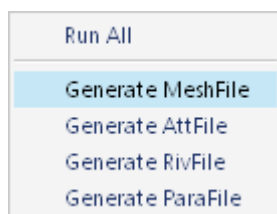
Data model loader reshapes the structure of GIS maps into a geo-database which is specifically designed according to the PIHM data model. The files prepared constitute spatial and relational attributes of the modeling domain. PIHM takes is required input in the form of 10 separate files: (1) mesh file; (2) att file; (3) soil file; (4) lc file; (5) riv file; (6) forc file; (7) ibc file; (8) para file; (9) calib file; and (10) prn file. More details about these files can be found in the PIHM Input File Formats [TODO: LINK]. Data model loader facilitates prepare 4 of those.

The steps are described in the following DataModel Loader dropdown menu:



7.1 Mesh File

Mesh file has all the irregular mesh (TIN) geometry information in it. It contains all the nodes and elements. For nodes, it records its location in space and for elements, it saves index of nodes of which elements comprise of and some topological relations in the form of its neighbor elements.



Select **Generate MeshFile** from the **DataModel Loader** drop down menu. This should bring Mesh File Generation dialog [Figure 7.1] on the screen.

In the input section of the dialog browse the 5 input files. First three: ele file, node file, and neighbor file are those obtained by the TRIANGLE in the section 6.2. Rest two files are the ESRI binary raster files corresponding to the surface and bedrock elevation of the modeling domain respectively. It should be noted that the files should be in the same geographic coordinate system. In the output section browse a mesh file to which the output will be saved. Execute the module

by clicking **Run**. Text browser should provide with any information regarding error or execution of the module. Click **Close** to close the dialog and proceed to next step.

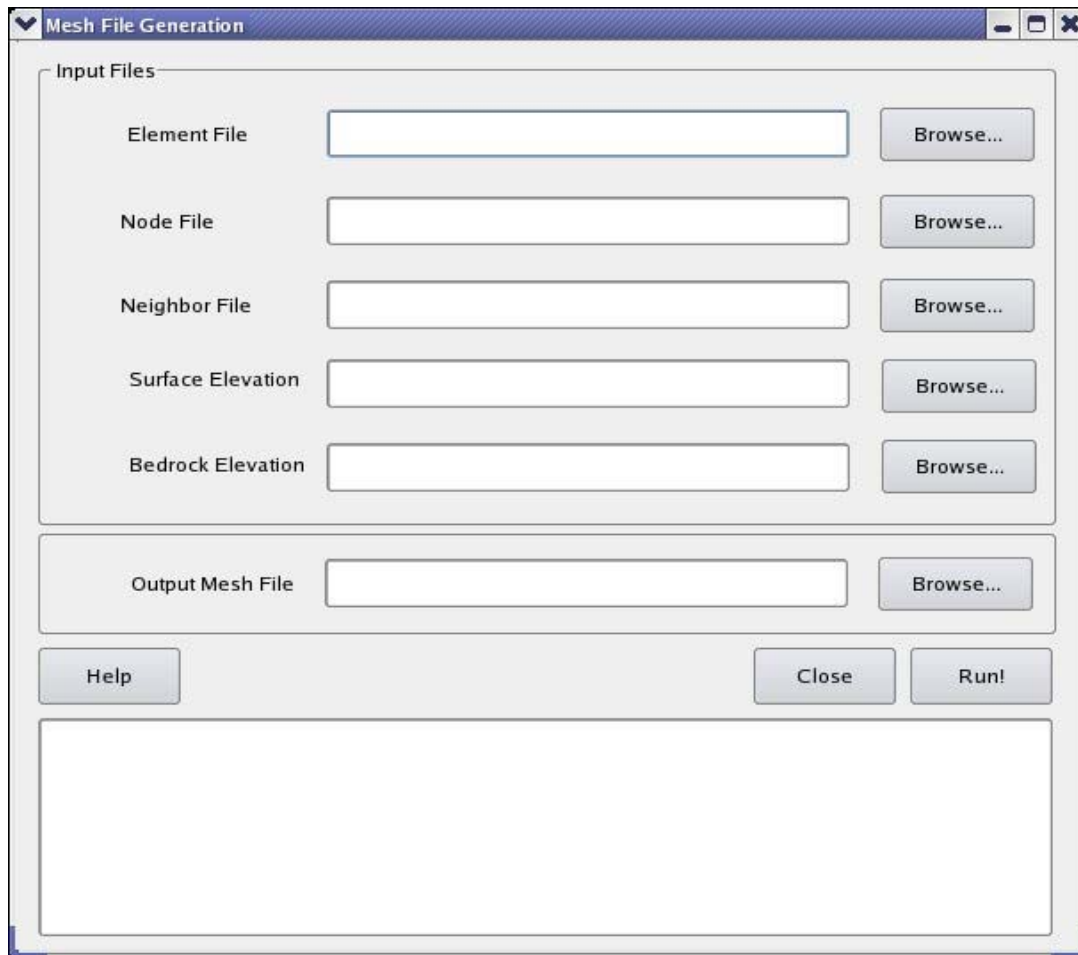
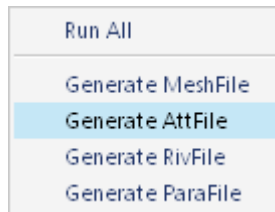
The image shows a software dialog box titled "Mesh File Generation". It has a standard Windows-style title bar with minimize, maximize, and close buttons. The dialog is divided into several sections. The top section is labeled "Input Files" and contains five rows, each with a text label, an empty text input field, and a "Browse..." button. The labels are "Element File", "Node File", "Neighbor File", "Surface Elevation", and "Bedrock Elevation". Below this section is another row with the label "Output Mesh File", an empty text input field, and a "Browse..." button. At the bottom of the dialog, there are three buttons: "Help" on the left, and "Close" and "Run!" on the right. Below these buttons is a large, empty rectangular text area for output or status messages.

Figure 7.1: Mesh File Generation Dialog

7.2 Att File

An att (attribute) file is a record which stores all the physical parameters class of each mesh elements such as soil type, land cover type, several forcing types. It allows efficient data storage.



Select **Generate AttFile** from the **DataModel Loader** drop down menu. This should bring Att File Generation dialog [Figure 7.2] on the screen.

In the input section of the dialog browse the TIN shape file generated in the section 6.3. Also browse the several forcing raster classification maps for the domain categorized in the three tabs. The class number should be integer starting with 1 for each forcing classification map. The data is extracted on the basis of selection of extraction method opted by the user.

It is advised to make sure that all the grid coverages span well enough to cover the decomposed domain or TINs generated. Also, the spatial reference used for each of the classification map must be the same. Click **Run** to begin processing. Any error or processing information is displayed on the text browser at the bottom. After successful completion of the routine press **Close** to close the dialog and proceed to the next step.

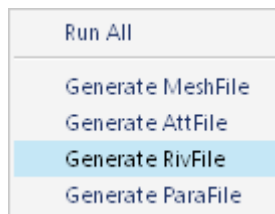


The dialog box is titled "Att File Generation". It features a "TIN File" input field with a "Browse..." button. Below this is a "Forcing" section with two tabs: "Landuse/Landcover" (selected) and "Miscellaneous". Under the "Landuse/Landcover" tab, there is an "Index File Input" section containing seven rows of input fields and "Browse..." buttons: "Precip.", "Temp.", "Humidity", "Wind Vel.", "G", "Rn", and "Pressure". Below the "Index File Input" section is an "Extract Using" section with three radio buttons: "Centroid" (selected), "Ortho-center", and "TIN Element". At the bottom of the dialog is an "Output Att File" input field with a "Browse..." button. The bottom-most section contains three buttons: "Help", "Close", and "Run!". A large empty rectangular area is located at the very bottom of the dialog box.

Figure 7.2: Att File Generation Dialog

7.3 Riv File

Topological information related to river segments (such as Node information; Left and Right Element) is stored in this file. Also different shape and material properties of river segments are provided. Other variables such as Initial and Boundary condition pertaining river segments are placed at the end of this file.



Select **Generate RivFile** from the **DataModel Loader** drop down menu. This should bring Riv File Generation dialog [Figure 7.3] on the screen.

In the input section of the dialog browse the River shape file (simplified) that was used in the section 5.4. Also, browse the ele, node and neigh files generated by the TRIANGLE in the section 6.2. In the output section of the dialog browse a riv file name where the output will be saved. Click **Run** to begin processing. Any error or processing information will be displayed in the text browser. Please note that the riv file generated by this module is not complete. User need to provide shape, material and initial condition information at the end of the file at the specified space. It is advised to refer the PIHM input file format [TODO: LINK] for more details. Click **Close** to close the dialog and proceed to next step.

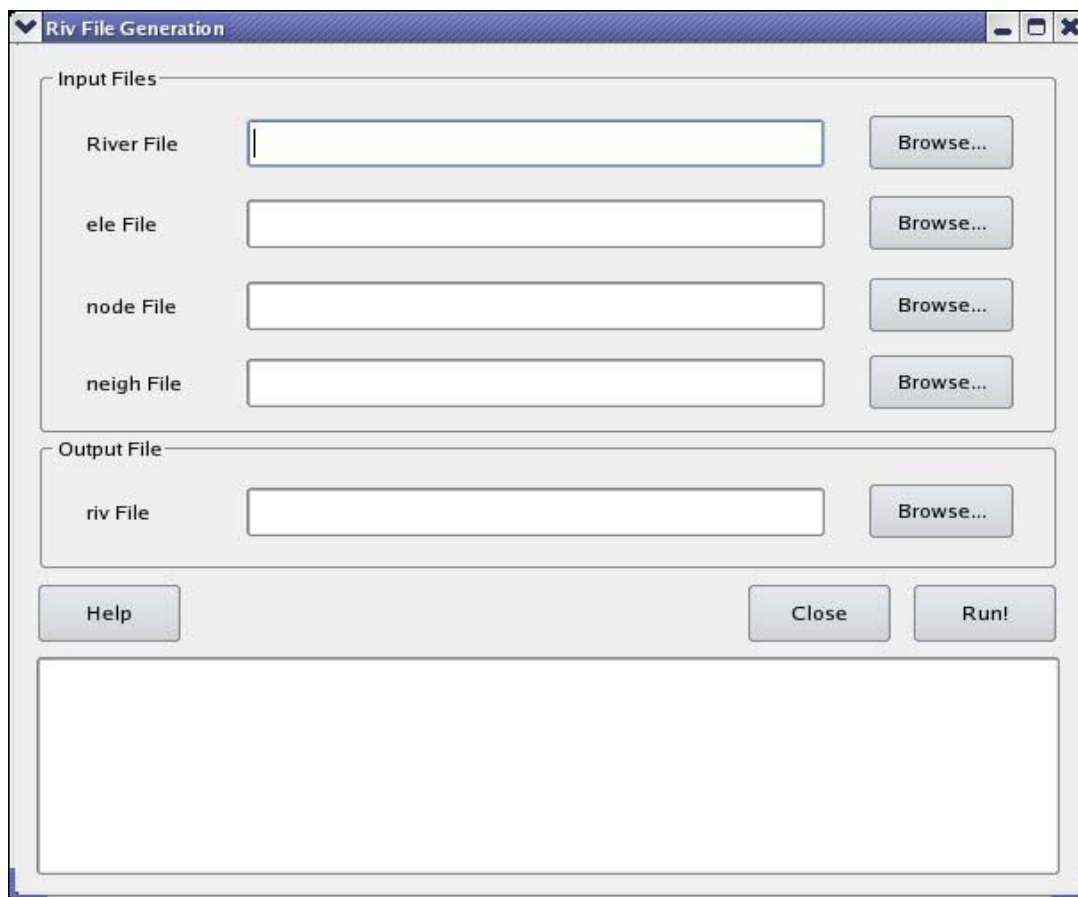
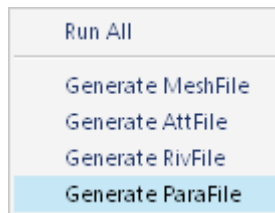


Figure 7.3: Riv File Generation Dialog

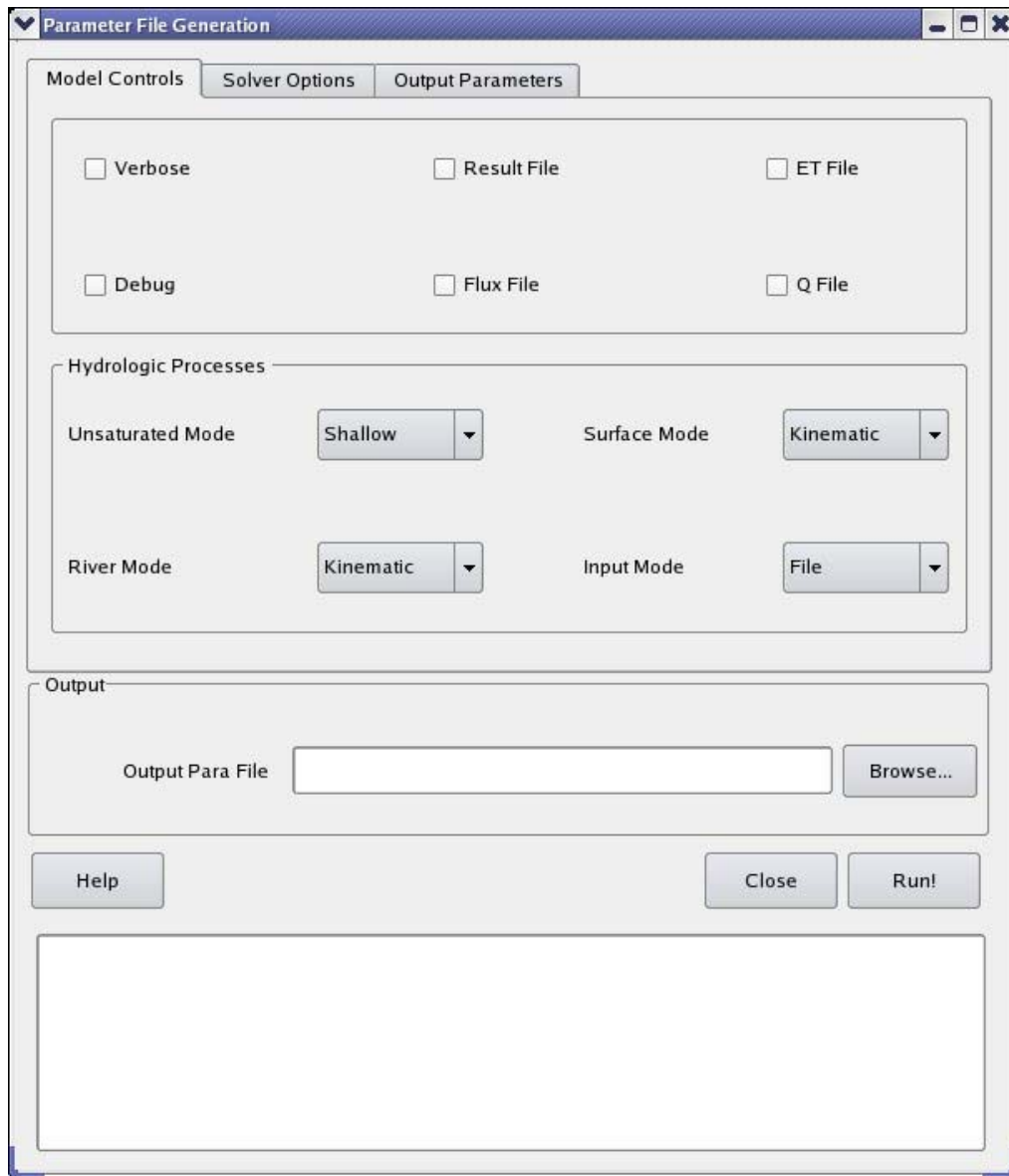
7.4 Para File

Para file provides all the control data to the model. It contains solver options; model modes; also parameters that govern model error.



Select **Generate ParaFile** from the **DataModel Loader** drop down menu. This should bring Parameter File Generation dialog [Figure 7.4] on the screen.

Select the several model and control parameter from the list provided on the dialog. In most of the situation user may leave the default values provided on the dialog. In the output section of the dialog browse output file name where the para file will be saved. Click **Run** to generate the output file with the specified options. Any error or processing information is displayed at the text browser. Click **Close** to close the dialog.



The dialog box is titled "Parameter File Generation" and features three tabs: "Model Controls", "Solver Options", and "Output Parameters". The "Output Parameters" tab is currently selected.

Model Controls

- ☐ Verbose
- ☐ Result File
- ☐ ET File
- ☐ Debug
- ☐ Flux File
- ☐ Q File

Hydrologic Processes

- Unsaturated Mode: Shallow
- Surface Mode: Kinematic
- River Mode: Kinematic
- Input Mode: File

Output

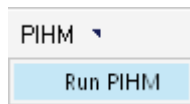
Output Para File:

A large empty rectangular area is located at the bottom of the dialog box.

Figure 7.4: Para File Generation Dialog

8. Run PIHM

RunPIHM is the module where PIHM has been integrated to Qgis. For a successful run of the model is required to have all the input files prepared in accordance with the file format of the PIHM. It should be noted that PIHMgis helps prepare only 4 files out of the 8 files needed for a model run.



Select **Run PIHM** from the **PIHM** drop down menu. This should bring PIHM dialog [Figure 8.1] on the screen.

In the input section of the dialog browse the folder where the input files for the model has been stored. The model assumes that each input file has the same first syllable for the file name followed by "." and file type (for e.g. domain.mesh; domain.att; domain.soil). Input File ID corresponds to that first syllable of the input files (for the "domain" example - input **domain** in the Input File ID textbox). Click **RUN** to begin processing. The interacting hydrologic processes of different timescales introduce stiffness into the system of ordinary differential equation. Therefore, the time a model run will take to complete depends on several factors which include: simulation duration, forcing data, system state. A progress-bar has been provided to keep track of progress of the simulation. After the completion of the model run **Close** the dialog.

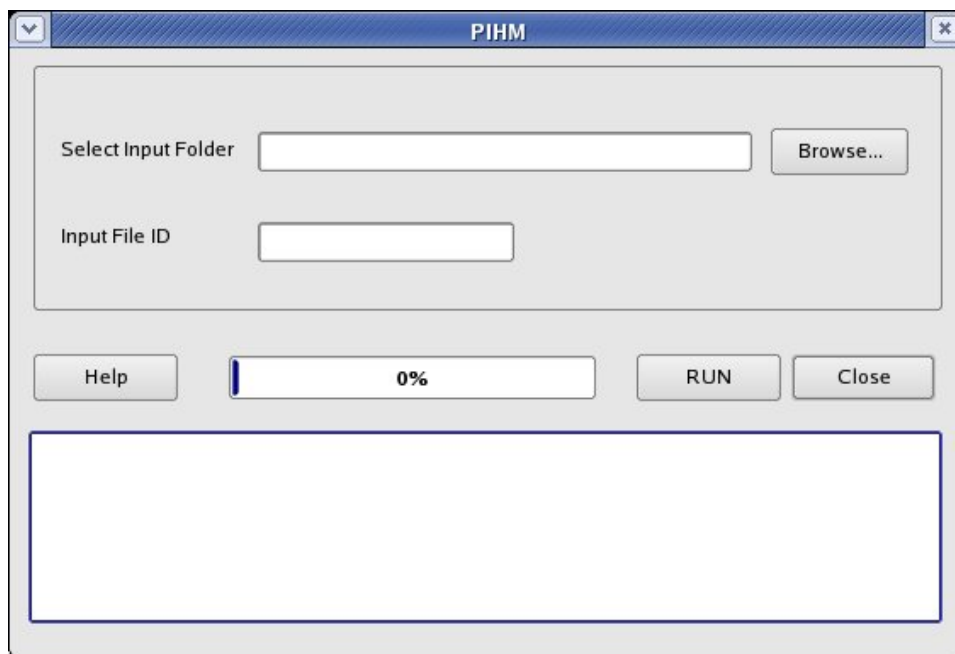


Figure 8.1: Run PIHM Dialog

9. Analysis

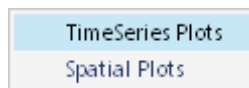
However there could be several ways of looking at the results and analyze them depending on the purpose of the modeling. Two basic types of analysis module have been integrated into this section. The enable analyze the time series and spatial pattern of several state variable corresponding to the model domain.

These modules can be accessed from the following Analysis drop down menu:



9.1 Time Series Plots

Time Series Plots can be used to plot temporal behavior of states or flux corresponding to either elements or river segments. It provides flexibility to visualize an element or river segment individually or as an average of all the feature elements.



Select **TimeSeries Plots** from the **Analysis** drop down menu. This should bring Time Series Plots dialog [Figure 9.1] on the screen.

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 “Select Feature by” ID has been selected. **Select Plot Variable:** provides option to choose from for the plot variable of interest. **Time-Average Interval** is used to average the model output by the value provided. User also needs to browse the model output file corresponding to variable of interest. Model saves the output files in the directory from where input files were fetched in the section 8. Click **Plot** to begin plotting. The process might take a while for processing the model output file. After successful processing of the model output file, time series plot is displayed using Qwt widget.

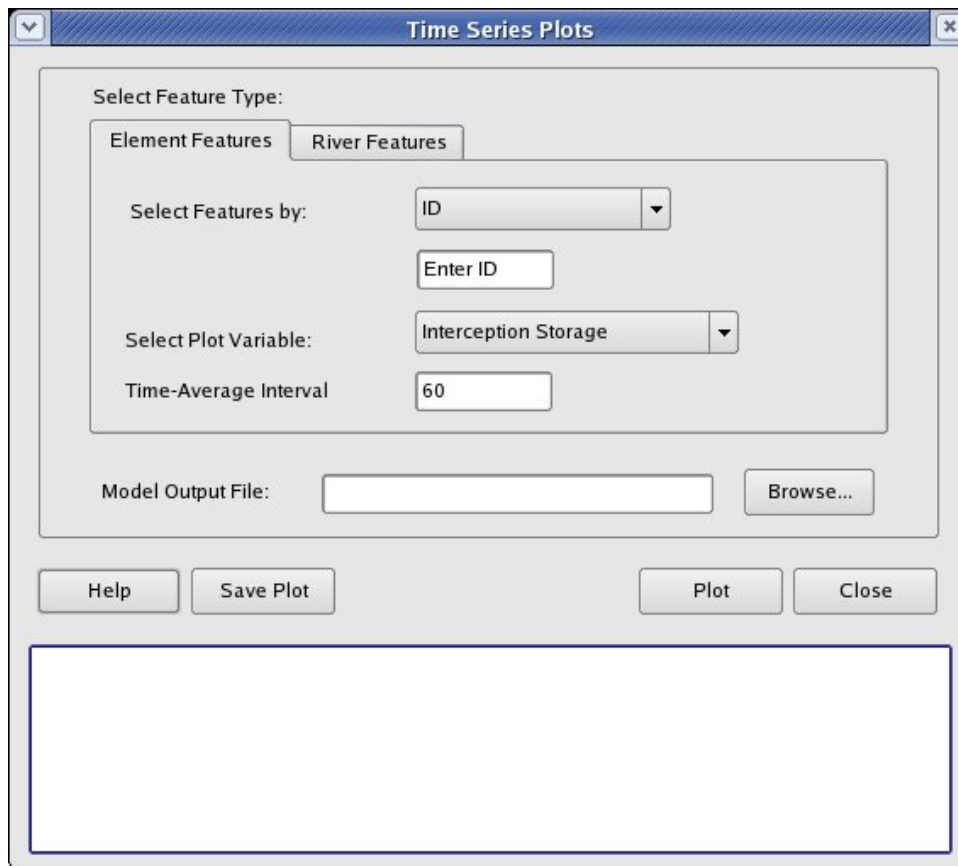
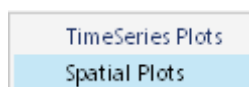


Figure 9.1: Time Series Plot Dialog

9.2 Spatial Plots

Spatial Plots can be used to plot spatial behavior of several state or flux variables corresponding to either elements or river segments. It provides flexibility to visualize spatial distribution of any of those variable as a snapshot of time or as an average over a give period of time.



Select ***Spatial Plots*** from the ***Analysis*** drop down menu. This should bring Spatial Plots dialog [Figure 9.2] on the screen.

Select Element feature or River Feature depending on the plot variable it corresponds to. Input **Start Time** and **Finish Time** over which it is desired to obtain average spatial plot of the plot variable. **Select Plot Variable:** provides option to choose from for the plot variable of interest. TIN shape file or River shape file also need to be browsed as the spatial map is generated based on them. User also needs to browse the model output file corresponding to variable of interest. Note that, model saves the output files in the directory from where input files were fetched in the section 8. Click **Generate** to begin processing. The routine might take a while for processing the model output file. After successful processing of the model output file, a shapefile named “tempShape.shp” is generated in the directory where PIHMgis.exe is located.

The image shows a software dialog box titled "Spatial Plots". It contains several input fields and buttons. At the top, there are two tabs: "Element Features" and "River Features". Below the tabs, there are two input fields for "Input Start Time" and "Input Finish Time", both containing the value "0". A dropdown menu for "Select Plot Variable:" is set to "Interception Storage". There are two "Browse..." buttons: one for "TIN Shape File" and one for "Model Output File". At the bottom, there are three buttons: "Help", "Generate", and "Close". A large empty rectangular box is at the very bottom of the dialog.

Figure 9.2: Spatial Plot Dialog