

**Tutorial for using Titan2D 4.0.0,  
online simulation tool on Vhub**

**Sylvain Charbonnier**

# Introduction

The Titan2D toolkit is used for modeling of geophysical mass flows over natural terrain (volcanic flows, avalanches, landslides etc.). Titan2D uses an adaptive finite volume scheme, and assumes that a large class of granular flows of geophysical origin can be modeled as an incompressible continuum satisfying a Mohr-Coulomb law (see References on the Titan2D <https://vhub.org/tools/titan2d> for detailed mathematical formulation). The solution is parallelized using the message-passing interface (MPI), and simulations for difficult terrains can be relatively time consuming even on large computational clusters. An essential input is the digital elevation model (DEM), since the flow simulation requires elevation, slope, curvature, and material information at every cell to be computed. This tool fronts a Titan2D simulation, but does not prepare the necessary DEM input that Titan2D relies upon, which is a prerequisite. (Note that the preparation of these DEMs can be done using the DEM converter tool for Titan at <http://vhub.org/resources/dem4titan>).

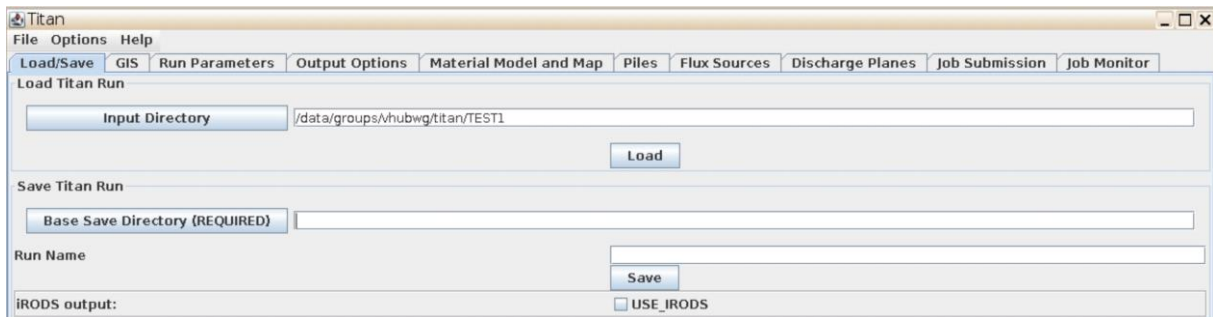
The Titan2D User Guide (<http://vhub.org/resources/300>) contains a detailed description of Titan2D and some information on the preparation of the inputs. This particular tutorial is intended to describe how to run Titan2D in the VHub environment. Within VHub, Titan2D and its Java-based GUI (graphical user interface) are already installed, along with the supporting cast of necessary software packages. At this time the actual execution of the Titan2D code can occur either within the limitations of the VHub server (local execution in the GUI job submission tab), or remotely on a large cluster (the Hub-submit option in the GUI). For local execution the runs must necessarily be limited in their size to a few (less than 8) processing cores, at most 1 GB or so of memory, and at most a few hundred MB of disk. When using the remote cluster there are fewer restrictions, you can utilize more cores and the associated memory (currently about 1GB per core). Storage is still an issue until the VHub storage pool is increased – you will need to ensure that your calculation will not generate more data than you can easily transfer back to VHub (at most a few hundred MB). You can approximate how much storage you need by the size of your DEM – multiply the size of your DEM by the number of time steps for which you output results of the flow. Decreasing the number of output steps will greatly reduce the storage requirements.

**During this tutorial, we will go through a stock example and (1) learn how to enter the different input parameters for running a Titan2D simulation on Vhub, (2) run the simulation and (3) visualize the results using the freely available Paraview software. If there is some time available, we will run a second simulation by changing only one parameter and investigate the effects on the simulated flow through a direct comparison of the results. We have a user group on vhub.org to help you using Titan2D. Visit the group page and join it here: <https://vhub.org/groups/titan2dusers>.**

## Walk-Through Example

The stock example comes as a prepared GIS/DEM for Merapi Volcano (Java, Indonesia) and can be found in the `/apps/tmp/iugg/titan2d/` directory on VHub, which also contains two folders called TEST1 and TEST2. In case of network problems to run the following example, these folders regroup all the documents needed to run the simulation and visualize the results.

## Load/Save Tab

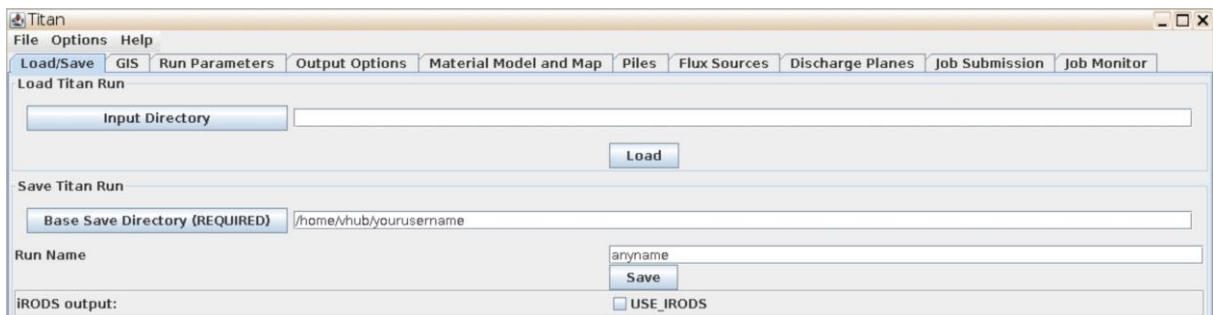


The screenshot shows the Titan GUI with the 'Load/Save' tab selected. The 'Load Titan Run' section has an 'Input Directory' field containing '/data/groups/vhubwg/titan/TEST1' and a 'Load' button. The 'Save Titan Run' section has a 'Base Save Directory (REQUIRED)' field which is empty, a 'Run Name' field which is empty, and a 'Save' button. At the bottom, there is an 'iRODS output:' section with a checkbox labeled 'USE\_IRODS' which is unchecked.

The Load/Save tab helps you to load a set of previously saved Titan2D input parameters, or save the current set, which can save you a lot of time/effort for frequently used set of run parameters.

**Load Titan Run:** (this step is not necessary) To load a previous run you need to enter in the field next to Input Directory the full path, including the name of the run, to the files with parameters for running a simulation. You can also click on the Input Directory button and navigate to that directory within a folder browser that pops up.

**Save Titan Run:** Here you have to specify the path (directory) where you want to store your new run. The directory name must begin with /home/vhub/username where *username* corresponds to your personal Vhub username (shown at the very top of the web page). You can also click on the Base Save Directory button and navigate to the folder where you save your Titan2D runs.

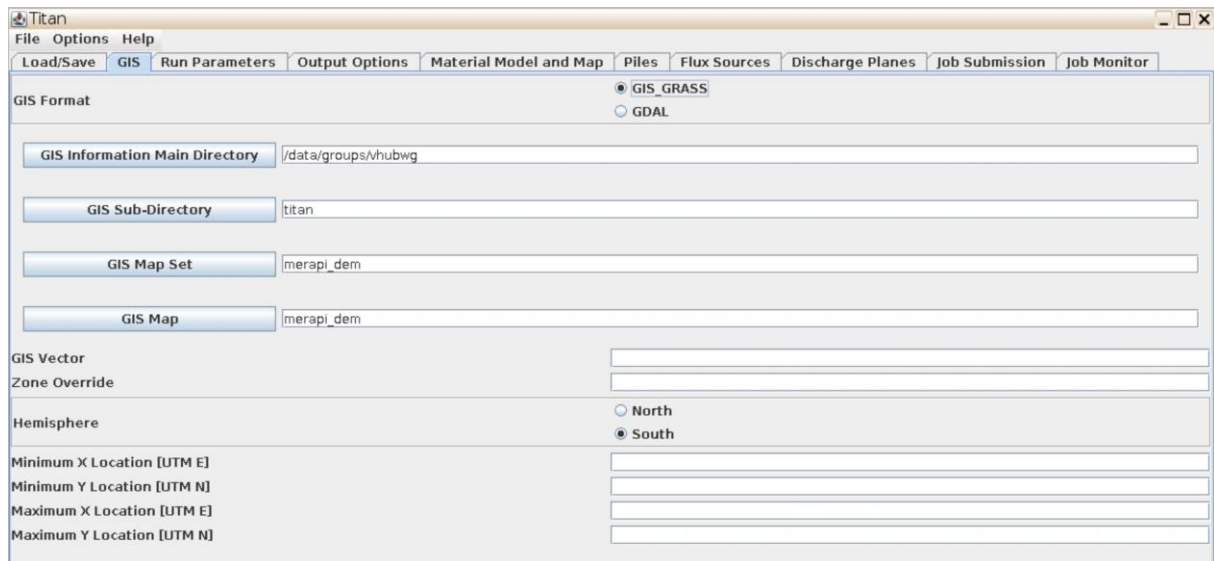


The screenshot shows the Titan GUI with the 'Load/Save' tab selected. The 'Load Titan Run' section has an empty 'Input Directory' field and a 'Load' button. The 'Save Titan Run' section has a 'Base Save Directory (REQUIRED)' field containing '/home/vhub/yourusername', a 'Run Name' field containing 'anyname', and a 'Save' button. At the bottom, there is an 'iRODS output:' section with a checkbox labeled 'USE\_IRODS' which is unchecked.

**Run Name:** The Run Name is actually a folder inside the previous directory where all the files with the parameters and results are saved. You need to press the Save button AFTER entering all other input parameters and just BEFORE submitting the job. The tool creates the folder (if it doesn't exist already) and save the parameters that you have entered in a series of files. To run the simulations the tool reads the input parameters from these files (not from the GUI). Following the example of the figure, the directory where the input files are saved would be /home/vhub/username/TEST1.

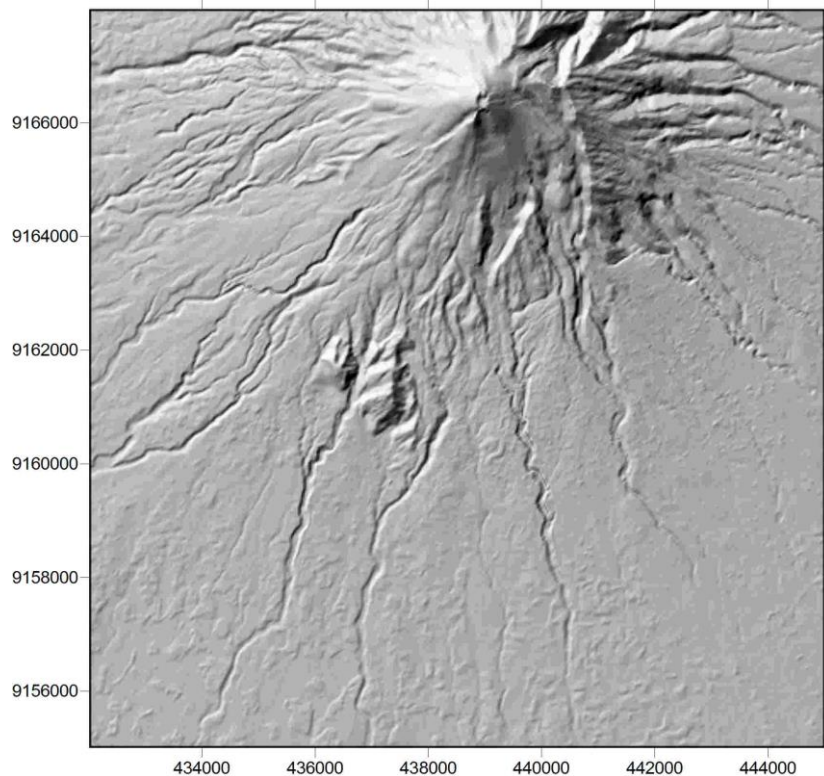
**NOTE:** After running a Titan2D simulation, the output files are put into a directory that is named by the time stamp of when it was run. You can find this output directory inside the directory you specified in the field above, that is /home/vhub/username/TEST1.

## GIS Tab



Titan2D performs flow simulations on a digital elevation model (DEM) of a user-defined region. The DEM data file, containing elevation data, must be formatted to operate in a GRASS (Geographic Resources Analysis Support System) GIS environment or in any GDAL Raster format (see [http://www.gdal.org/formats\\_list.html](http://www.gdal.org/formats_list.html) for a list of examples). GRASS GIS is an open source GIS with raster, topological vector, image processing, and graphics production functionality. You can download the software at <http://grass.fbk.eu/>.

For this example we'll be using a DEM of Merapi Volcano (Java, Indonesia). These data are located in a temporary directory created for this workshop session. The Main Directory is /data/groups/vhubwg and the Sub-Directory is 'titan'. Both the GIS Map Set and GIS Map folder names are 'merapi\_dem'. The 'GIS vector' tab allows you to enter the outline of an actual event in a SPRING GIS format (<http://www.dpi.inpe.br/spring/english/index.html>). You can also specify the hemisphere where your volcano is located. The minimum and maximum X and Y locations can be used if a computational region smaller than the entire GIS region is desired, the user can input the minimum and maximum x and y location of the desired computational region in UTM coordinates.



## Run Parameters Tab

The screenshot shows the Titan software interface with the 'Run Parameters' tab selected. The window title is 'Titan' and it has a menu bar with 'File', 'Options', and 'Help'. The 'Run Parameters' tab is active, showing various settings for simulation parameters. The 'Restart' section has radio buttons for 'True' and 'False', with 'False' selected. Below it is a 'Restart File' input field. The 'Maximum Number of Time Steps' is set to 10000, 'Maximum Time [s]' is 80, and 'Number of Computational Cells Across Smallest Pile/Flux-Source Diameter' is 20. The 'AMR' section has radio buttons for 'True' and 'False', with 'True' selected. The 'Order Method' section has radio buttons for 'First' and 'Second', with 'Second' selected. The 'Scale Parameters' section has radio buttons for 'Scale Simulation' set to 'True'. Below this are input fields for 'Length Scale [m]' (3000), 'Gravity Scale [ms<sup>-2</sup>]' (9.8), and 'Height Scale [m]'. The 'Stat Props' section includes a table for 'Run ID', 'Height used to define flow outline (>0) [m]', 'Test if flow Reaches Height [m]', 'Flow Height Test Point X Location [UTM E]', 'Flow Height Test Point Y Location [UTM N]', and 'Stat Props Prefix'. The 'Outline Props' section has radio buttons for 'Outline Props Enabled' set to 'True'. Below this are input fields for 'Max Linear Size' (1024), 'Init Size' (radio buttons for 'AMR' and 'DEM', with 'AMR' selected), and 'Outline Props Prefix'.

**Restart:** Titan has the capability to restart from a saved run. This may be necessary if the maximum number of iterations has been reached or simulation time has expired or the computer simply crashes before you would like the simulation to end. Instead of running the entire simulation over again, you can continue from the point where a previous run has left off.

**Maximum Number of Time Steps/Maximum Time:** The next two input parameters concern the run time of the simulation. The user must specify the Maximum Number of Time Steps (on the order of several thousand) and Maximum Time (in seconds). Fractions of seconds may be used if desired (e.g. 2.5 seconds). When the job is submitted for processing, the simulation will stop when it has reached the specified maximum allowed number of time steps or has simulated the specified amount of time (whichever comes first).

**Note:** Determining an acceptable stopping criterion for the simulation is difficult when considering general cases. Because of this, the stopping criteria for the simulation must be set by the user. The two criteria that are used are Maximum Time and Maximum Number of Time Steps. Both of these values should be set high enough such that the geologic event being simulated has ended (e.g. the material has come to rest). If either of these values is set too low, the simulation will end before the material has reached static equilibrium. If both of these values are set excessively high, wasted computation will be performed dynamically simulating a pile that has already come to rest. The simulation will end when either the number of time steps computed is equal to the Maximum Number of Time Steps or the simulation time has reached the Maximum Time. The number of computed time steps needed to simulate a geologic event will vary depending on the amount of computational mesh points used, the friction parameters, the use of grid adaptation, the simulation order, and initial pile geometry and location.

**Number of Computational Cells Across Smallest Pile/Flux-Source Diameter:** The amount of cell refinement within Titan is determined by the smallest pile/flux source diameter and the number of cells the user wants across it. The default case as an example (a value of 20) can be used. Then, at the beginning of a simulation, the grid-cell containing the centroid of each pile is successively refined (i.e. split into 4 'son' cells) until its size is no larger than 1/20th of the smallest pile or flux source diameter. This is the size of the smallest computational cell allowed on the map during the course of the simulation. It is important to choose a value here that strikes a balance between computation time and calculation accuracy.

**AMR (Adaptive Mesh Refinement):** This refers to the computational grid and reduces the computational cost while maintaining the simulation's accuracy. However, this can also introduce some instability into the computation. Also note that if an adaptive grid is not selected, then the entire computational domain will be a uniform maximally refined grid with a cell-size determined by the user's specification in the Number of Computational Cells Across Smallest Pile/Flux-Source Diameter box above. Due to the major savings in computation time, it is recommended to select it unless instability is detected in the output.

**Order Method:** Clicking on the Second button for this option allows you to select the 2nd order method for calculating the values in a computational grid cell (the 1st order method is the default). Under the first order method, the values for pile height, momentum etc. that are calculated by the model are approximated as constant across the entire cell. This may mean that there is a jump up or down to the value of the same parameter in the neighboring cell. Under the 2nd order method, the values of the parameters are assumed to vary linearly across the cell. The 2nd order method takes into account the value of the neighboring cells and uses those values to calculate the value for the cell in question. For example, if the neighboring cells up-flow of the cell in question have values lower than the cells down-flow of the cell in question, the value of the cell is going to increase (have a slope) in the down-flow direction. If there is no difference in values between the neighboring cells, then the cell in question will also stay constant. Selecting the 2nd order method will produce slightly more accurate results, but may also increase the computation time as the code has to perform more calculations.

**Scale Simulation:** Titan2D allows for several properties of the simulation to be scaled. Clicking the 'True' button allows the governing equations to be non dimensionalized using pile height, gravity and length scaling factors (among others, such as velocity and time scales, that are secondarily derived from them). The height scale used by titan is the cube root of the total volume from all piles and flux sources. It is calculated at the beginning of the simulation and then is a constant from that point on. The gravity scaling factor is taken as  $9.8 \text{ ms}^{-2}$ .

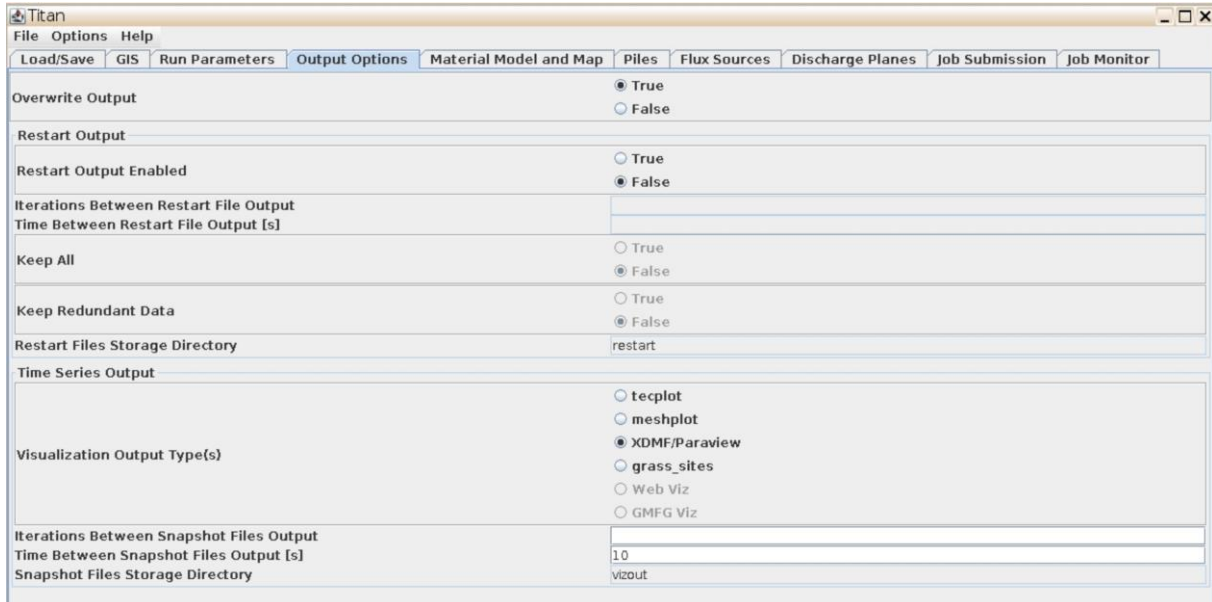
**Scale Length:** This scale factor refers to the expected runout length of the flows and need only be specified if the simulation is to be scaled, however for all real-terrain calculations it is highly recommended that the simulation be scaled. Note: Scaling will help reduce the occurrence of round-off error and is critical for the proper control of propagating thin layers (i.e. the boundaries of a flow), which otherwise may have a tendency to develop unrealistically high velocities.

**Height used to define flow outline (> 0):** This box sets the 'boundary' of the mass flow. The user can define a pile boundary (i.e. ignore material less than a specified pile height) by entering a value greater than zero here. This will be used to, among other things, compute the spread of the pile in the x and y directions. Spread in each direction is defined as the maximum minus minimum coordinate where the pile height is greater than the value entered here. If no value is entered it defaults to 1/50<sup>th</sup> of the maximum initial pile height.



**Test if flow reaches height [m] ...: and ... at test point (x and y location):** These options set the criteria to determine if the flow reaches a particular height or location, namely did flow of this depth reach this location at any time during the calculation.

## Output Tab



**Override Output:** This option allows the user to overwrite the output data of a previous simulation when the 'Restart Output' option is enabled.

**Restart Output Enabled:** This option allow the user to restart saving the output files of a previous simulation that crashed. If 'True' is selected, be sure that you also selected 'True' in the 'Restart' option of the 'Run Parameters' Tab.

**Iterations/Time Between Restart File Output:** The user can specify how often an output file is to be created after a previous simulation crashed.

**Keep All:** This option allow the user to keep all the output data of a previous simulation when the 'Restart Output' option is enabled.

**Keep Redundant Data:** This option allow the user to keep the redundant data of a previous simulation when the 'Restart Output' option is enabled.

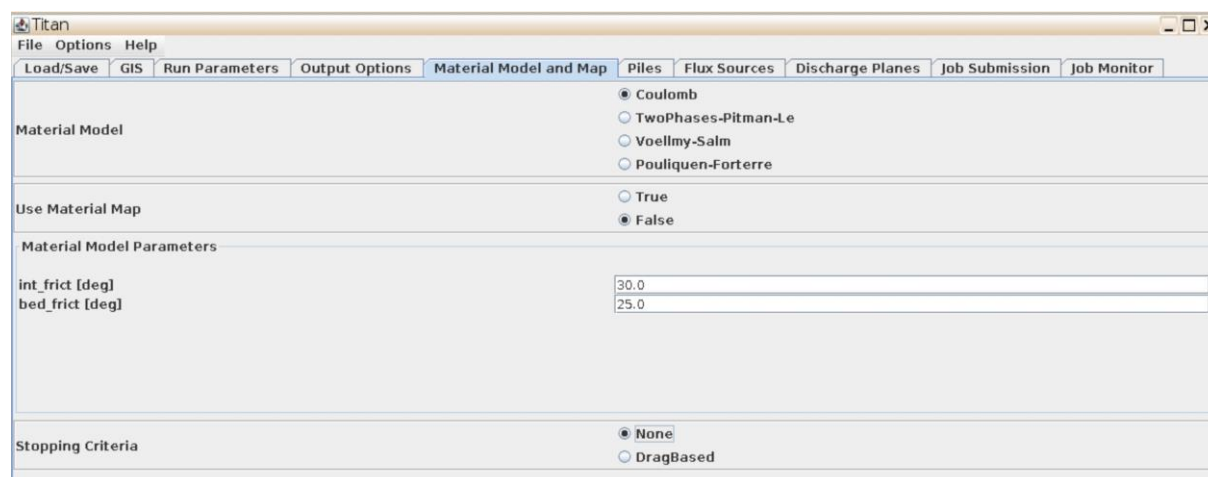
**Restart File Storage Directory:** The user can specify the directory and name of the folder where the restart files will be saved when the 'Restart Output' option is enabled.

**Visualization Output Type(s):** This option allows the user to choose the formats of the output files (tecplotxxxx.plt, mshplotxxxx.plt, GMFG Viz, XDMF/Paraview, Web Viz, or grass sites). Activate the buttons corresponding to the visualization outputs that are desired. tecplotxxxx.plt and mshplotxxxx.plt are tecplot files (<http://www.tecplot.com/>). XDMF/Paraview generates xdmfxxxx.h5 and xdmfxxxx.xml files used with the freely available Paraview visualization application (<http://www.paraview.org/HTML/Index.html>). The user can have multiple visualization output formats with each simulation run.

**Iterations/Time Between Snapshot Files Output:** The user can specify how often an output file is to be created. For example, the user may wish to generate an output file every 5 seconds of simulated time or every let's say 5,000 timesteps. Remember that the lower these values, the greater the storage space that is used in your account.

**Snapshot Files Storage Directory:** The user can specify the directory and name of the folder where the snapshot files will be saved.

## Material Model and Map Tab



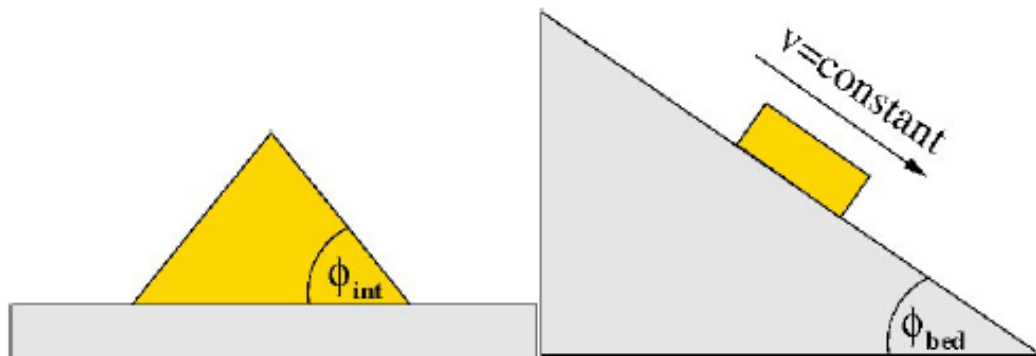
**Material Model:** The user can specify the rheological law used by the model during the simulation. For a description of the different laws, please refer to the Table below and the paper of Kelfoun (2011) published in Journal of Geophysical Research entitled ‘Suitability of simple rheological laws for the numerical simulation of dense pyroclastic flows and long-runout volcanic avalanches’. The ‘TwoPhases-Pitman-Le’ rheology is only suitable to model water-saturated flows (i.e., lahars). Please note that the code is very sensitive to the rheological law and associated values chosen by the user. Therefore, a proper understanding of the different laws and the range of realistic values of the parameters proposed here is crucial before starting a simulation.

Name of the Law	Equation
Coulomb	$\mathbf{T} = \rho h \left( g \cos \alpha + \frac{u^2}{r} \right) \tan \varphi_{\text{bed}} \frac{\mathbf{u}}{\ \mathbf{u}\ }$
One angle	$k_{\text{act/pass}} = 1$
Two angles	$k_{\text{act/pass}} = 2 \frac{1 \pm [1 - \cos^2 \varphi_{\text{int}} (1 + \tan^2 \varphi_{\text{bed}})]^{1/2}}{\cos^2 \varphi_{\text{int}}} - 1$
Viscous	$\mathbf{T} = 3\mu \frac{\mathbf{u}}{h}$
Voellmy (Coulomb + $u^2$ term)	$\mathbf{T} = \rho h \left( g \cos \alpha + \frac{u^2}{r} \right) \tan \varphi_{\text{bed}} \frac{\mathbf{u}}{\ \mathbf{u}\ } + \xi \rho \ \mathbf{u}\  \times \mathbf{u}$
Plastic (constant retarding stress)	$\mathbf{T} = T_0 \frac{\mathbf{u}}{\ \mathbf{u}\ }$
Plastic + $u^2$ term	$\mathbf{T} = T_0 \frac{\mathbf{u}}{\ \mathbf{u}\ } + \xi \rho \ \mathbf{u}\  \times \mathbf{u}$
Bingham (plastic + viscous)	$\mathbf{T} = T_0 \frac{\mathbf{u}}{\ \mathbf{u}\ } + 3\mu \frac{\mathbf{u}}{h}$

<sup>a</sup>Terms are  $\rho$ , density;  $h$ , thickness;  $g$ , gravity;  $\alpha$ , slope;  $u$ , depth-averaged velocity;  $r$ , slope curvature;  $\varphi_{\text{bed}}$ , basal friction angle;  $\varphi_{\text{int}}$ , internal friction angle;  $k_{\text{act/pass}}$ , earth pressure coefficient;  $\mu$ , viscosity;  $T_0$ , yield strength;  $\xi$ , Voellmy coefficient.



**Use Material Map:** The next feature is available only to users who have a specific material map (file ends in `_Mat` in the GIS subdirectory folder). The ‘True’ check box enables the input of a GIS-based surficial material map that matches the area covered by the DEM. This map is used to define the zones in the region where changes in the surface morphology results in a change in the basal friction angle. When this function is enabled, the subsequent boxes will ask for the internal and basal friction angles for each material represented on the material map. If leave ‘False’, only one internal friction angle and one basal friction angle are necessary.



The internal friction angle  $\phi_{int}$ , is the steepest angle that the upper surface of a conical pile of dry sand can make with respect to the horizontal plane it is resting on. The bed (also known as basal) friction angle,  $\phi_{bed}$ , is the angle that a plane needs to be inclined so that a block of material will slide downslope at a constant speed.

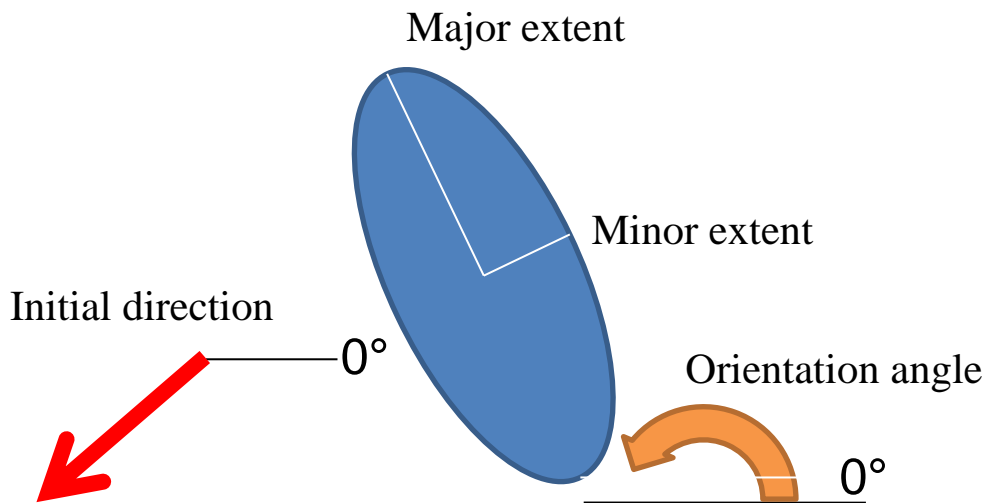
**Stopping criteria:** This option allow the user to use the drag-based criteria developed in the Yu et al. (2009) paper published in ‘Natural Hazards’ entitled ‘Numerical issues in computing inundation areas over natural terrains using Savage-Hutter theory’.

## Piles Tab

Titan									
File Options Help									
Load/Save GIS Run Parameters Output Options Material Model and Map Piles Flux Sources Discharge Planes Job Submission Job Monitor									
Type (1:P 2:C)	Maximum Initial Thi...	Center of Initial Vo...	Center of Initial Vol...	Major Extent	Minor Extent	Orientation Angle	Initial Speed	Initial Direction	
1	30	439000	9166150	100	75	-90	0	0	

This user-input information is used to characterize the nature of the material in the simulation.

Titan2D Pile geometry in map view:



The data to be entered by the user are: Type (1: Paraboloid, 2: Circular), Maximum Initial Thickness (in meters), Center of Initial Volume (in UTM coordinates), the Major and Minor Extent of the initial pile (in meters), the Orientation angle (degrees from X axis to major axis), its Initial speed (m/s) and direction (degrees from X axis). The orientation angle value must range from  $-180^\circ$  (clockwise rotation) to  $+180^\circ$  (counter-clockwise rotation) from east =  $0^\circ$ .

NOTES: The user must fill EACH box with numbers, even if the corresponding value is 0 (and press Return after typing the values in each field). Initial volumes in Titan2D are in  $\text{m}^3$  and are calculated from the pile parameters like this:

Volume =  $1/2 * \pi * \text{radius\_min} * \text{radius\_max} * h$ , which for a  $r_{\text{max}}=100$ ,  $r_{\text{min}}=75$  and  $h=30$  gives you a volume of  $353429.2 \text{ m}^3$ .

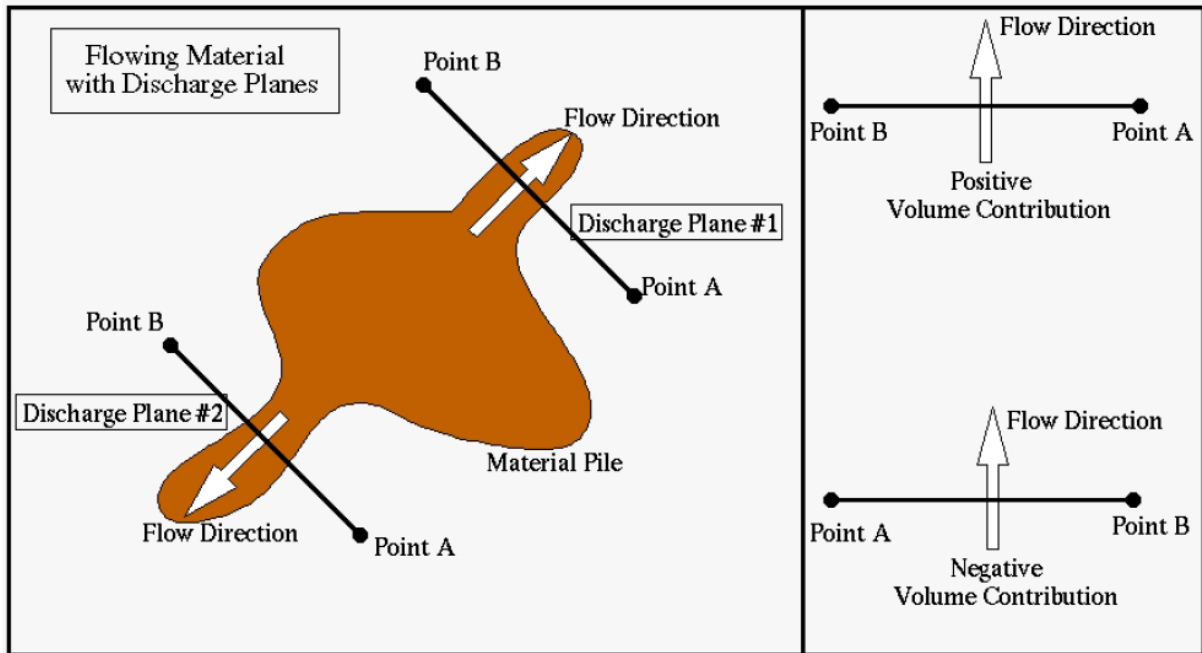
## Flux Sources Tab

The flux source shares the same paraboloid profile as a pile source, with the vertical dimension being the rate of mass per unit area (units of meters per second). As with a pile source, the major axis of the paraboloid can be aligned at an arbitrary angle in the East-North plane, and an arbitrary initial East-North velocity (tangential to the terrain) can be assigned to the mass effusing-from or raining-down-on the ground. The flux source starts at the specified value and linearly decreases to zero at the end of the fluxes duration.

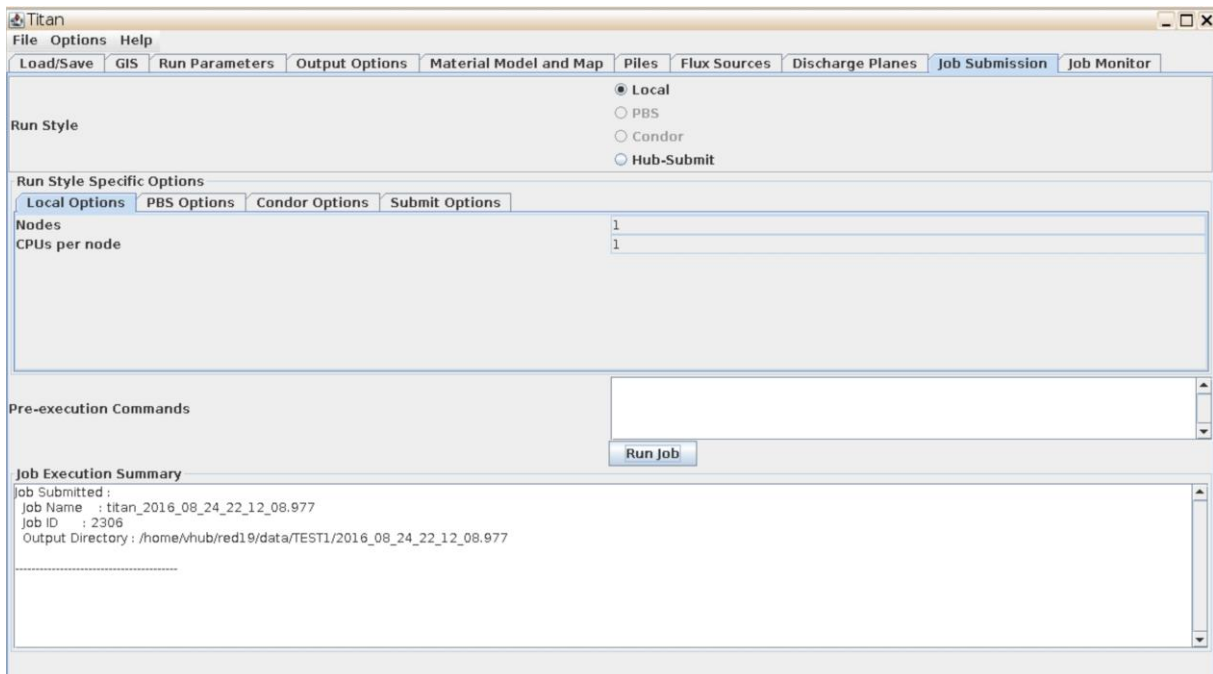
## Discharge Planes Tab

This window lets the user enter the coordinates of the two endpoints of a vertically-oriented discharge-plane. Each point is identified by a UTM E and UTM N coordinate. During the simulation Titan will calculate the amount (cubic meters) of material passing between these points and write the data in 'discharge.out' files. Discharge planes have an orientation - meaning that flow passing through them in one direction may be recorded as having a positive volume, whereas flow passing through in the opposite direction will have a negative volume contribution. The orientation of discharge planes is such that if each point of 4 planes are

successively laid out in a counter-clockwise manner (where the result is that the planes form a closed box), then any flow leaving the box will have a positive volume contribution. (That is, the material flux through a discharge plane obeys a right-hand rule sign convention.) See Figure below for further explanation.



## Job Submission Tab

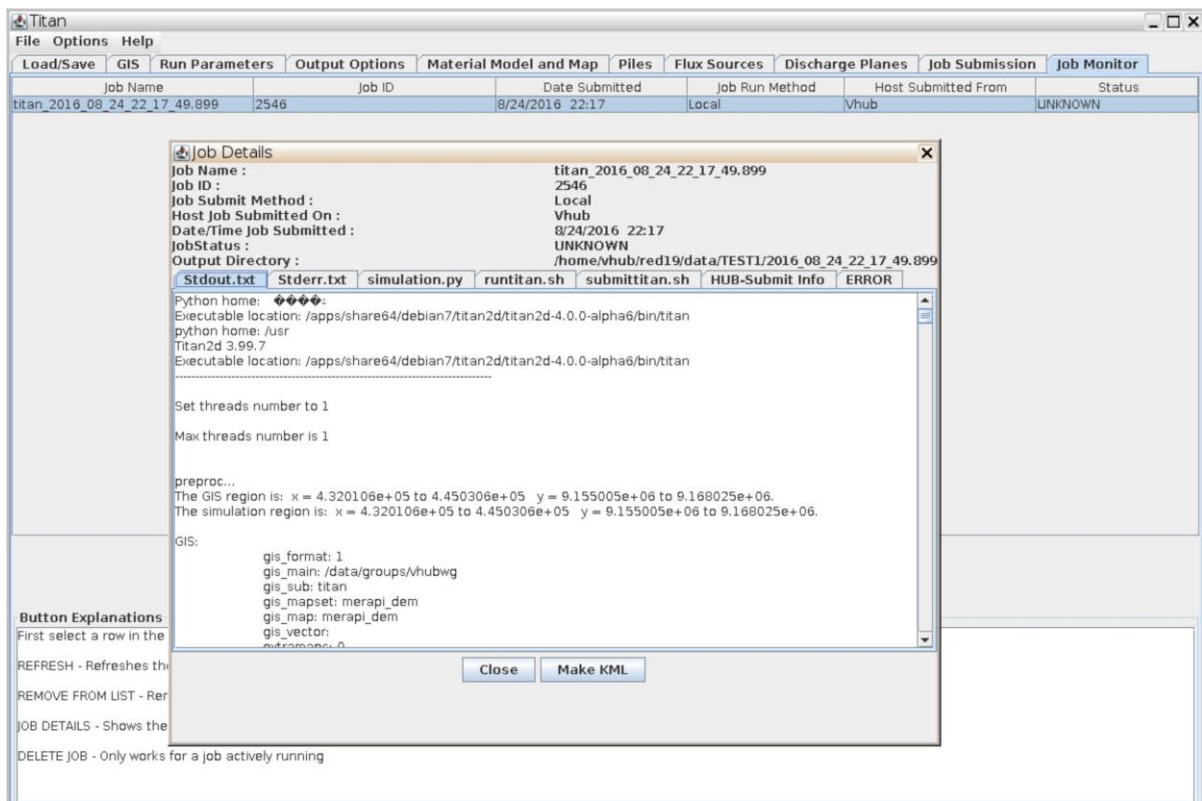


Once the GUI tabs have been filled out for the GIS/DEM and mass flow parameters, you reach the Job Submission tab. For execution in the VHub environment you should choose the Local execution (the Hub-submit option can be used to run on a remote cluster, useful for larger jobs, but it takes longer to queue up and wait for the jobs to run) and click on the 'Run Job' button. Remember to save your parameters in the Load/Save tab before you submit the

job. The stock example parameters should take less than 1 minute to run on a single processing core, and once completed successfully you can use your preferred method for accessing the VHub filesystem to browse the results. Note that you can find some help in using [WebDAV](https://vhub.org/kb/tips/webdav) at <https://vhub.org/kb/tips/webdav>.

Please only use the ‘Local’ and ‘Hub-Submit’ options, the ‘PBS’ and ‘Condor’ options will not work in the VHub environment.

## Job Monitor Tab



From the ‘Job Monitor’ tab, click on a job and then click on the ‘Job Details’ button. A new window will appear that will show the output from the job. Note that you can read here the Output Directory with the full path to your results; you will need to retrieve this directory when downloading the output files from your Vhub storage to your computer.

## Visualizing the output files using Paraview

Download and install on your computer the last version of Paraview software at <http://www.paraview.org/paraview/resources/software.php>.

Using the Vhub ‘Workspace’ online tool or any other ssh file transfer protocol tool, download the entire Titan2d simulation folder from your Vhub storage into your computer. See more details at <https://vhub.org/kb/tips/webdav>.

Open Paraview and open the ‘vizout\_xdmf\_p0000.xmf’ file from your Titan2d simulation folder and play with the software options to display the various simulation results...