User manual: AIM (Ash Impact Model)

A procedure for modelling volcanic ash hazards

GEOSCIENCE AUSTRALIA
RECORD 2009/XX

CVGHM logo PHIVOLCS logo

Australian Government
Geoscience Australia

Department of Resources, Energy and Tourism
Minister for Resources and Energy: The Hon. Martin Ferguson, AM MP
Secretary: Mr John Pierce

Geoscience Australia
Chief Executive Officer: Dr Neil Williams PSM

ISSN 1448-2177
ISBN 0 642 XXXXX X

GeoCat # XXXXX

Bibliographic reference: xxxx

Table of Contents

Introduction

PURPOSE

Fall3D-6 models the dispersal of volcanic ash during explosive eruptions in order to determine how ash is concentrated in the atmosphere and where it is deposited at ground level. The purpose of this manual is to introduce a new user with no modelling background to Fall3d (Version 6). This will be achieved through step-by-step instructions for installing and running simulations of volcanic ash fallout, both for validation (historical eruptions) and predictive (ongoing or future) purposes. Fall3D-6 has been incorporated into a larger framework called the Ash Impact Model (AIM) for the purposes of eventually extending the capability of Fall3D to model the impact of volcanic ash loading at ground level. AIM currently models the hazard component only.

SCOPE

This manual will provide instructions for installing and running AIM in a Unix/Linux environment. It will incorporate step-by step instructions for creating volcanological, meteorological and topographic input files, running an eruptive scenario and viewing the results.

AUDIENCE

This resource is intended for geoscientists and natural hazard modellers with a volcanological background but no computer programming background.

Background

The distribution and thickness of volcanic ash deposited during mildly to highly explosive volcanic eruptions has important life safety, livelihood, economic and political implications for densely populated areas that are affected. A number of computational modelling tools have been developed in recent decades for forecasting the transport and deposition of volcanic ash. Fall3d-6 is one of these tools and was developed by Osservatorio Vesuviano (http://datasim.ov.ingv.it/Fall3d.html). Fall3d-6 is able to model the distribution and thickness of volcanic ash on the ground and in the atmosphere during explosive volcanic eruptions. It has the ability to volcanic ash dispersion in a 3-dimensional wind field that experiences changing wind speed, direction and air temperature over time. Fall3d-6 also considers the interaction between topography and the 3D wind field and the impact this may have on dispersal of ash at ground level.

The hazard models produce isopach maps. An isopach is a contour line connecting points of equal volcanic ash thicknesses (or ash load or ash concentration etc). Therefore each map may contain isopachs of ash thicknesses that vary in appearance according to the volcanological and wind conditions of the eruption that the user is modelling.

UNIX setup instructions

AIM is designed to run in a UNIX/Linux environment such as Ubuntu Linux. Although directories and output files can be viewed and manipulated through the windows manager the user is still required to run the model from a UNIX command line using a terminal window. The user is therefore required to know a number of basic UNIX commands. These are usually combinations of typed characters. There are four main commands to remember for navigating through a UNIX environment:

- **cd** <*directory name*> **- CHANGE DIRECTORY**
  (Open <u>this</u> directory)

- **cd ..** *(type a space after cd)* **- GO UP ONE DIRECTORY**
  (Close this directory and open the parent directory)

- **cd ../..** *(type a space after cd)* - **GO UP TWO DIRECTORIES**
  (Keep adding "**/..**" to go up more than two directories)

- **ls -l** *(type a space after ls)* **- LIST**
  (Show me a list of what is inside this directory)

- **pwd** **- PRINT WORKING DIRECTORY**
  (Show me where I am located)

Other commands that the user may require to use AIM include:

- **svn co** **- CHECKOUT**
  (Refers to 'checking out' a repository, scripts etc)

- **mkdir – MAKE DIRECTORY**
  (Make a new directory (folder) at this location – this is followed by a space and the name of the new directory)

- **python – RUN PYTHON SCRIPT**
  (This is then followed by a space and the name of the python script that the user would like to run)

- **ln –s – LINK**
  (Allows you to create a shortcut to a specified directory from the current directory)

Initial Installation

The steps to install AIM and Fall3d on the AIFDR cluster are outlined below. Apart from specific pathnames, the instructions apply more generally to other Linux systems.

STEP 1 – INSTALLING AND TESTING AIM

Instructions for installing AIM onto your PC for use in a linux/UNIX environment are detailed below. Green text highlights the UNIX commands that are used. Blue text indicates the path to be taken and red text indicates other variables such as file names, websites and usernames. A number of test scenarios are also included for testing the installation. You will only need to follow this step once for initial setup purposes.

The “<” and “>” symbols indicate that the user must insert information in this space and not include the symbols themselves (see example below).

Eg. **mkdir /model_area/sandpits/<username>**
mkdir /model_area/sandpits/bearad

1) Open a new terminal (double click on the display icon on the desktop)

2) To create a work area directory (known as a sandpit) in AIFDR’s modelling area type:

   mkdir /model_area/sandpits/<username> (<username> e.g. bearad)

3) To create a shortcut to your sandpit type:

   ln -s /model_area/sandpits/<username> ~/sandpit

4) To change directory into your sandpit type:

   cd ~/sandpit

5) To get AIM from the AIFDR repository type:

   svn co --username <userame> http://www.aifdr.org/svn/aim/branches/fall3d_v6 aim

6) To change to the AIM source code directory type:

   cd ~/sandpit/aim/source/aim

7) To install Fall3D-5.1.1 type:

   python install_fall3d.py

8) When prompted with 'update .bashrc file (Y or N) type:

   Y

9) You will be asked to give a location for FALL3D to be installed. You need to type:

~/FALL3D

10) You will then be asked to give a location for TEPHRADATA to be stored. You need to type:

/model_area/tephra

11) You will then need to type:

source ~/.bashrc

(Important - include the space between 'source' and '~/.bashrc')

12) You then need to run the python script again to complete the installation. Type:

python install_fall3d.py

13) To change directories and begin testing type:

cd ~/sandpit/aim/testing

14) To run the test suite type:

python test_aim.py

We have supplied five test eruptive scenarios in the testing area and it is highly recommended that you run each one in turn and confirm that results are produced.

15) To run each test scenario (one at a time) type:

python merapi.py
python tambora.py
python galunggung.py
python mayon.py
python pinatubo.py

The time required to output results varies according to eruption duration, magnitude etc (allow 5 – 20 mins per scenario). To view the output files for each scenario:

16) On the desktop select the **Places** tab on the toolbar and select **home folder** to open the file browser window
17) Select **File System** from the list of options on the left
18) Double click on the directory **model_area**
19) Double click on the directory **tephra**
20) Double click on the directory for the volcano that was tested (i.e. merapi, tambora etc)
21) Double click on the directory that contains the simulation you ran (time stamped)

The name of this directory will reflect the volcano name, your user name and a time stamp (i.e **volcano_username_time** for when the simulation was run in local Indonesian time. The output files can be viewed here as well as a copy of the input data used the run the scenario.

STEP 2 – INSTALLING VOLCANIC ASH MODELLING REPOSITORY

Production runs can now be run from anywhere on your account. AIFDR suggests AIM models be run from the repository **volcanic_ash_modelling**. To checkout the repository:

1. In the terminal window return to your sandpit
   i.e **home/<username>/sandpit** – this directory already contains a folder called **aim** and you can confirm this by typing:

   ls –l

2. Once in the directory sandpit type:

   svn co http://www.aifdr.org/svn/volcanic_ash_modelling --username <username>

   (i.e. svn co http://www.aifdr.org/svn/volcanic_ash_modelling --username bearad)

3. To confirm that your sandpit contains the directory **volcanic_ash_modelling** type:

   ls –l

4. Change to the **volcanic_ash_modelling** directory

   cd volcanic_ash_modelling

5. To run a new scenario, change the indonesian_scenarios directory

   cd indonesian_scenarios

6. Change into the directory for the volcano you would like to model and see below for preparation of input data.

   cd volcano *(i.e. merapi)*

Preparation of input data

AIM requires the user to provide three kinds of input data:

- a volcanological input file which details the physical properties of the eruption being modelled (eruption column height, mass flow rate, grainsize distribution) created using the python script <**scenarioname**>**.py**

- a topographic grid file which contains spatial data (x,y,z) for the region (the terrain onto which ash is deposited) - <**scenarioname**>**_topography.txt**

- a projection file which accompanies the spatial data and contains geo-referencing information required to create Google Earth KML output files - <**scenarioname**>**_topography.prj**

- a meteorological file which contains wind speed, direction and air temperature information for the eruption. There are 3 options:

  Vertical wind profile (simplest) - user created - <**scenarioname**>**_wind.txt**
  Calmet62 (complex) – not used
  NCEP re-analysis 1 (complex) – processed data (optional)

STEP 3 – TOPOGRAPHY INPUT FILE

AIM users have access to a repository of DEM's. This repository contains DEM's of varying spatial scale for every volcano in Indonesia as well as a small number of Philippines and Papua New Guinea volcanoes. Each DEM has the file extension '*.txt' and has an accompanying projection file used by AIM to generate Google Earth layers. The projection file has the same filename as the DEM but the extension '*.prj'. This repository is located in:

/model_area/tephra/dems

1. Copy the DEM and the projection file for the volcano you are modelling

   Eg. **srtm_**<**scenarioname**>**_**<**km**>**_**<**cellsize**>**.txt** and;
   srtm_<**scenarioname**>**_**<**km**>**_**<**cellsize**>**.prj**

2. Open the subdirectory for the scenario you are running in volcanic_ash_modelling

   (i.e. **~/sandpit/volcanic_ash_modelling/indonesian_scenarios/merapi**)

3. Paste the two files into this subdirectory

4. Rename the files <**scenarioname**>**_topography.txt and** <**scenarioname**>**_topography.prj** respectively

STEP 4 – METEOROLOGICAL INPUT FILE

A. 3D wind field (multiple points)

To model in a 3-dimensional wind field the user will input NCEP (re-analysis 1) meteorological data pre-processed for the time period of interest. This data captures the monthly mean conditions for a user specified domain and is particularly useful for recent eruptions (last 40 years) and predictive scenarios. Fall3d-6 will create a 3D wind field based on this information. One ncep1 input file is needed per scenario. A number of files are provided for AIM users in the directory:

5. Copy the ncep1 file for the volcano you are modelling
6. Open the subdirectory for the scenario you are running in volcanic_ash_modelling

   (i.e. **~/sandpit/volcanic_ash_modelling/indonesian_scenarios/merapi**)

7. Paste the ncep1 file into this subdirectory
8. Rename the file <**scenarioname**>**.ncep1.nc**

   (i.e **~/sandpit/volcanic_ash_modelling/merapi/merapi.ncep1.nc**)

B. Vertical wind profile (single point)

The user will not always be able to obtain 3D wind data particularly in the case of historical eruptions where only the prevailing wind direction can be estimated from the preserved ash deposits. In this instance the user can create a simplified vertical wind profile using the template in Appendix 2. To construct a vertical wind profile the user must have information on typical prevailing wind directions, speeds and air temperatures for the region being modelled. Fall3d-6 will create a 2D wind field based on this information (typically wind conditions at a single point directly above the vent).

1. Create a profile using guidelines in Appendix 2
2. Save as <**scenarioname**>**_wind.txt** in the subdirectory for that scenario.

   (i.e **~/sandpit/volcanic_ash_modelling/merapi/merapi_wind.txt**)

STEP 5 – VOLCANOLOGICAL INPUT FILE

The python script <**scenarioname**>.**py** is deigned to create the volcanological input file (the forth and final input file required by fall3d-6). This is where the user adds all the remaining information needed to run a scenario. A worksheet of all input parameters required by <**scenarioname**>.**py** is detailed in Appendix 3.

Preparation:

1. For each new scenario use the worksheet to record the details of the eruption you are running (location, date, eruption observations etc) and to ensure you have all the input variables required to run the scenario. A range table (Appendix 4) and a glossary of terms (Appendix 5) are also available to assist with completing the worksheet. The worksheet can be kept for reference and also referred to when using <**scenarioname**>.**py**.

Using the model script <scenarioname>.py:

1. In the Ubuntu environment (orange screen) select the **places** tab from the tool bar at the top of he screen and select **home folder** to open the file browser as a new window

2. A copy of the script can be found at **/sandpit/aim/testing/** *(The same scripts used during STEP 1 – Installing and testing AIM)*

   i.e **merapi.py**

3. Copy the script and paste it into the subdirectory for scenario you are modelling in volcanic_ash_modelling

   Note: It is important that each model script has the following code at the bottom:

```
# Run model using specified parameters
if __name__ == '__main__':
 from aim import run_scenario
    run_scenario(__file__)
```

4. Modify the script input variables using the worksheet you created.
5. Rename the script <**scenarioname**>.**py** where applicable

This completes the data gathering process. You should now have four files in the subdirectory for the volcano you are modelling:

<scenarioname>.py
<scenarioname>_topography.txt
<scenarioname>_topography.prj
<scenarioname>_wind.txt

Modelling procedure

STEP 6 – RUNNING A SCENARIO USING AIM

1. Open a new terminal
2. Navigate to the subdirectory where the four input files are located by typing:

   cd ~/sandpit/volcanic_ash_modelling/indonesian_scenarios/<scenarioname>

3. To run the scenario type:

   python <**scenarioname**>**.py**

   (i.e. **python merapi.py**)

4. Whilst the model is running select the **places** tab from the toolbar and select **home folder** to open the file browser as a new window
5. Enter the directory:

   /model_area/tephra/scenarios/<scenarioname>

6. The output files for each simulation are located in time stamped directories. If you added a comment on the eruption in the script <scenarioname>.py this will appear at the end of the directory name.

   e.g. <**username**>_<**year**>-<**month**>-<**day**><**time**>_<**eruption_comment**>

7. Double click on the simulation you are running to open it and you will be able to view the output files as they are being generated.

This completes the modelling procedure. Refer to the Appendix 8 for a summary on viewing model results.

Appendix 1 – Template for topography input file

1. Surfer GRD file (used by Fall3d-6 in native mode – will be automatically generated by AIM using the DEM srtm_<scenarioname>_<km>_<cellsize>.txt (Refer to Step 3)

```
DSAA
59 64
412432.3 471432.3
9106708.6 9170708.6
0.0 0.0
0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
5.200000E+01   1.250000E+02   1.030000E+02   1.710000E+02   1.920000E+02   1.830000E+02   2.150000E+02
```

2. ESRII ASCII file (format of srtm_<scenarioname>_<km>_<cellsize>.txt: required by AIM)

```
ncols         59
nrows         64
xllcorner     412432.33601038
yllcorner     9106708.627275
cellsize      1000
NODATA_value  -9999
358 347 335 325 358 376 404 442 469 488 548 575 578 664 690 779 814 850 964 951 975 1091 1164 1234 1366
1426 1562 1586 1571 1420 1370 1296 1095 986 924 836 789 751 659 615 545 517 484 432 405 393 351 352 287
```

Appendix 2 – Template for vertical wind profile (AIM only)

Each block of data represents one hour. Each one hour block contains the number of z-layers defined by the user in the input variable worksheet (Appendix 3). The number of z layers in each block must remain constant. For each z layer, the user must input three variables: wind speed (m/s), wind direction (degrees from North/ radians) and air temperature beginning with the z layer of lowest altitude. Wind direction (second variable) can be reported as a number value between 0 and 359 or as a direction N, NE, E, SE, S, SW, W or NW. The example below is for a 3 hour eruption. This eruption involves nine z-layers. The wind speed, direction and air temperature for each z layer for each hour is listed: wind directions are reported were possible in degrees from azimuth (north) or as a direction (NE, E).

Eg.

```
Hour 1
10 10 14
10 12 4
10 12 1
10 11 -2
10 10 -12
10 13 -30
10 10 -42
10 11 -50
10 11 -50

Hour 2
10 NE 14
10 NE 4
10 NE 1
10 10 -2
10 10 -12
10 NE -30
10 10 -42
10 10 -50
10 NE -50

Hour 3
10 89 14
10 E 4
10 E 1
10 94 -2
10 90 -12
10 87 -30
10 88 -42
10 E -50
```

10 E -50

Appendix 3 – Volcanological input worksheet

 # Run Date: ____________________________________
 # Scenario Name: ____________________________________
 # Run number: ____________________________________

 # Eruption observation details:

____

#Time Eruption_Year = ____# YYYY
Eruption_Month = ____# MM
Eruption_Day = ____# DD
Start_time_of_meteo_data = ____# Hours after 00
Meteo_time_step =____# Hours
End_time_of_meteo_data =____# Hours after 00
Start_time_of_eruption =____# Hours after 00
End_time_of_eruption = ____# Hours after 00
End_time_of_run =____# Hours after 00

 # Grid
Coordinates = ____# LON-LAT/UTM
Longitude_minimum =____# LON-LAT only
Longitude_maximum =____# LON-LAT only
Latitude_minimum = ____# LON-LAT only
Latitude_maximum =____# LON-LAT only
Longitude_of_vent =____# LON-LAT only

Latitude_of_vent =____# LON-LAT only
X_coordinate_minimum =____# UTM only
X_coordinate_maximum =____# UTM only
Y_coordinate_minimum = ____# UTM only
Y_coordinate_maximum =____# UTM only
X_coordinate_of_vent = ____# UTM only
Y_coordinate_of_vent = ____# UTM only
Number_of_cells_X_direction = _______________________________________
Number_of_cells_Y_direction = _______________________________________
Z_layers = ____# List Z layers in increasing height order

 # Granulometry
Grainsize_distribution = ____# Possibilites are GAUSSIAN/BIGAUSSIAN
Number_of_grainsize_classes = _______________________________________
Mean_grainsize = ____# phi
Sorting = ____
Minimum_grainsize = ____# phi
Maximum_grainsize = ____# phi
Density_minimum = ____# kg/m3
Density_maximum = ____# kg/m3
Sphericity_minimum = ____
Sphericity_maximum =____

 # Source
Source_type = ____# Possibilities are 'plume', 'suzuki', 'point'
Mass_eruption_rate = ____# kg/s (if point, if suzuki, if plume where Height_or_MFR = MFR)

Height_above_vent = ______________________________ # m (if point, if suzuki or if plume where Height_pr_MFR = q Height)
A = ______________________________ # (suzuki only)
L = ______________________________ # (suzuki only)
Height_or_MFR = ______________________________ # plume only
MFR_minimum = ______________________________ # kg/s (plume only)
MFR_maximum =______________________________ # kg/s (plume only)
Exit_velocity = ______________________________ # m/s (plume only)
Exit_temperature = ______________________________ # K (plume only)
Exit_volatile_fraction = ______________________________ # % (plume only)

 # Fall3D
Terminal_velocity_model = ______________________________ # Possibilities are ARASTOOPOR/GANSER/WILSON/DELLINO
Vertical_turbulence_model = ______________________________ # Possibilities are CONSTANT/SIMILARITY
Horizontal_turbulence_model = ______________________________ # Possibilities are CONSTANT/RAMS
Vertical_diffusion_coefficient = ______________________________ # m2/s
Horizontal_diffusion_coefficient = ______________________________ # m2/s
Value_of_CS = ______________________________ # RAMS only

 # Output
Postprocess_time_interval = ______________________________ # Hours
Postprocess_3D_variables = ______________________________ # Yes/No
Postprocess_classes = ______________________________ # Yes/No
Track_points = ______________________________ # Yes/No

Appendix 4 – Range table

Eruption scenarios that are modelled using Fall3d-5.1.1 are subject to a range of acceptable input limits regarding eruption column height, mass eruption rate and eruption duration. If the user attempts to model a scenario not possible in nature (i.e. a VEI 2 eruption with an 80000m eruption column for 1 hour) then the model is likely to fail. The range table below details the acceptable range of eruption column heights, eruption column increments, mass eruption rates and eruption durations that should be adhered to when considering a new scenario. The table is based on the volcano explosivity index (VEI).

Ranges	**VEI 2***	**VEI 3**	**VEI 4**	**VEI 5**	**VEI 6**	**VEI 7**	**VEI 8**
Eruption column height (m)	2000-5000	3000-15000	10000-25000	25000-30000	30000-50000	30000-50000	50000+
Eruption column height increments (m)[#]	1000	1000-10000	10000	10000	10000	10000	10000
Mass eruption rates (kg/s)	$1x10^{4}-1x10^{6}$	$1x10^{4}-1x10^{6}$	$1x10^{5}-1x10^{8}$	$1x10^{5}-1x10^{8}$	$1x10^{9}-1x10^{12}$	$1x10^{9}-1x10^{12}$	$1x10^{9}-1x10^{15}$
Eruption duration (hours)	1-6	1-6	1-6	6-12	>12	>12	>12

** Fall3d-5.1.1 will not model effusive eruptions (VEI 0) or eruptions that produce eruption columns <2000m in height (VEI 1).*

[#] Increments must always be the same magnitude of order as the eruption column height (i.e. 3000m (1000 increments; 40000m (10000)

Appendix 5 – Glossary of volcanological and meteorological terms

Point Mass of an eruption column is released at a single source point (Refer to Pg.13 of the Technical manual located in FALL3D/Fall3d-Pub/Documents directory)	**ARASTOOPUR** Mathematical formula for estimating the settling velocity of particles. (Refer to Pg. 4 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.
Suzuki Mass of an eruption column released according to an empirically derived formula (Refer to Pg.13 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.	**GANSER** Mathematical formula for estimating the settling velocity of particles.(Refer to Pg.4 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.
Plume Mass of an eruption column released according to the buoyant plume theory. (Refer to Pg.13 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.	**WILSON** Mathematical formula for estimating the settling velocity of particles. (Refer to Pg.4 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.
Rams/ Constant (horizontal) Equations for solving the horizontal diffusion co-efficient of settling particles. (Refer to Pg. 3 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.	**DELLINO** Mathematical formula for estimating the settling velocity of particles. (Refer to Pg. 5 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.
Similarity/ Constant (vertical) Equations for solving the vertical diffusion co-efficient of settling particles. (Refer to Pg.3 of the Technical manual located in the FALL3D/Fall3d-Pub/Documents.	

Appendix 6 – Using the Fall3D-6 natively (without AIM).

A. CREATING A VOLCANOLOGICAL INPUT FILE NATIVELY:

If you plan to run Fall3d-6 natively (without the assistance of AIM) then you need to manually create the volcanological input file. The simplest way to do this is to copy an example input file and edit it.

In the Ubuntu environment (orange screen) select the **places** tab from the toolbar and select **home folder** to open the file browser as a new window:

1. Double click on the directory **FALL3D**
2. Double click on the directory **Fall3d-PUB**
3. Double click on the directory **Runs**
4. Double click on the directory **Vesuvio1944**
5. Copy the input file **Vesuvio1944.inp**
6. Return to Runs
7. Create a new directory for the scenario

   i.e. **merapi**

8. Paste the input file into the new directory
9. Rename the input file <**scenarioname**>**.inp**

   i.e. **merapi.inp**

10. Make changes to <**scenarioname**>**.inp** using the worksheet as a guide.
11. Save and close

B. CREATING A TOPOGRAPHY INPUT FILE NATIVELY:

1. Create a topography file using the template in Appendix 1 (Surfer GRD format)
2. Alternatively edit an example file located in:

   FALL3D/Fall3d-PUB/Runs/Vesuvio1944

   i.e. **Vesuvio1944.top**

3. Rename the file <**scenarioname**>**.top** and save it in the subdirectory for the scenario being modelled:

   i.e. **/FALL3D/Fall3d-PUB/Runs/merapi/merapi.top**

C. CREATING A METEOROLOGICAL INPUT FILE NATIVELY:

4. The **ncep1** file used by AIM can be used natively by fall3d-6 (Refer to Step 4).
5. Save the file in the subdirectory of the scenario being modelled:

   i.e. **/FALL3D/Fall3d-PUB/Runs/merapi/merapi.ncep1**

   Alternatively if you choose the vertical profile option you must create your own profile. The profile template in Appendix 2 is for AIM only.

6. To create a vertical profile for use by Fall3d-6 in native mode, edit an example file located in:

   FALL3D/Fall3d-PUB/Runs/Vesuvio1944

   i.e. **Vesuvio1944.profile**

7. Rename the file <**scenarioname**>**.profile** and save it in the subdirectory for the scenario being modelled:

   i.e. **/FALL3D/Fall3d-PUB/Runs/merapi/merapi.profile**

D. RUNNING A SCENARIO USING THE FALL3D-6 NATIVE SCRIPTS (WITHOUT AIM).

1. Open a new terminal
2. Change to the directory **FALL3D**
3. Change to the directory **Fall3d-PUB**
4. Change to the directory **Scripts**

There are a number of scripts responsible for executing each stage of the modelling process. The Script-manager coordinates the process. The user needs to create an alias for the Script-manager. This means that when the user types the alias and presses enter the Script-manager will be executed. The alias used in the example below is 'fall3d'. Example runs for Vesuvio, Mt Etna and Mt. Spurr are provided with the initial download of Fall3d-6 for the user to test. Each scenario uses a different meteorological input file format (profile, ncep1 or calmet62) in order to give a more complete understanding of the capabilities of the model. For the purpose of this demonstration we will continue to use the merapi example. Please note this is a fictitious example and will not appear in the directory Runs.

5. To create an alias type:

   alias fall3d='./Script-manager'

6. To test the alias is in affect type:

   fall3d *and press enter*

The Script-manager should have run and the user should be able to see a print-out of the contents of the Script-manager (see below)

```
*****************************************************
Fall3d-6.2  usage: Fall3d option Problemname [args]

 Options:

 -h      This help

 SetDbs Problemname [profile/calmet62/ncep1]
       Runs the SetDbs utility program for Problemname
       Input  files:
            Problemname.inp                 (input file)
            Problemname.type.nc             (meteo file)
       Output files:
            Problemname.dbs.nc              (netCDF database)
            Problemname.SetDbs.log          (SetDbs log file)

 SetGrn Problemname
       Runs the SetGrn utility program for Problemname
       Input  files: Problemname.inp        (input file)
       Output files: Problemname.grn        (granulometry file)
             Problemname.SetGrn.log         (SetGrn log file)

 SetSrc Problemname
       Runs the SetSrc utility program for Problemname
       Input  files: Problemname.inp        (input file)
            Problemname.grn                 (granulometry file)
            Problemname.dbs.nc              (netCDF database)
Output files:
            Problemname.src                 (source file)
            Problemname.SetSrc.log          (SetSrc log file)
            Problemname.Plume.res           (characteristics of the plume. Only for PLUME option)
            Problemname.Plume.height        (height of the plume. Only for PLUME option)
            Problemname.Plume.mass          (mass of the plume. Only for PLUME option)

 Pub Problemname
     Runs Fall3d Public version for Problemname
     Input  files:
            Problemname.inp                             (input file)
            Problemname.grn                             (granulometry file)
            Problemname.src                             (source file)
            Problemname.dbs.nc                          (netCDF database)
            Problemname.pts                             (tracking points list file)
     Output files: Problemname.Fall3d_ser.log           (Fall3d log file)
            Problemname.res.nc                          (netCDF results file)
```

Problemname.rst.nc *(netCDF restart file)*
Problemname.res.tps *(tracking points file)*

****
**

If this is the case then the test is complete and the user is ready to model. If not, repeat the previous two steps.

To run Fall3d-6 natively the user must execute each script individually and in the **correct** order. This is important as output files from one script form the input files needed by the next script. You should also open the subdirectory for the scenario you are running in a separate window

(Places>Home Folder>FALL3D>Fall3d-PUB>Runs>Vesuvio1944).

From here you can:

- view the input files you are using and confirm they are all named correctly
- view the output files of each stage of the modelling process as they are generated
- view the log files created by each utility to confirm successful completion of each stage

7. To run the utility SetGrn and generate a grainsize profile for the scenario type:

   fall3d SetGrn <**scenarioname**>

   Eg. **fall3d SetGrn Vesuvio1944**

8. Double click on the log file '<scenarioname>.SetGrn.log' (eg. Vesuvio1944.SetGrn.log) and confirm the utility was completed successfully.

9. To run the utility SetDbs and generate the meteorological database files type:

   fall3d SetDbs <**scenarioname**> **profile** (or **ncep1**)

   Eg. **fall3d SetDbs Vesuvio1944 profile**

10. Double click on the log file '<scenarioname>.SetDbs.log' (eg. Vesuvio1944.SetDbs.log) and confirm the utility was completed successfully.

11. To run the utility SetSrc and generate the eruptive source profile for the scenario type:

    fall3d SetSrc <**scenarioname**>

    Eg. **fall3d SetSrc Vesuvio1944**

12. Double click on the log file '<scenarioname>SetSrc.log' (eg. Vesuvio1944.Src.log) and confirm the utility was completed successfully.

13. To run the utility Pub and run the fallout model type:

    fall3d Pub <**scenarioname**>

    Eg. **fall3d Pub Vesuvio1944**

    Double click on the log file '<scenarioname>.Fall3d_ser.log' (eg. Vesuvio1944.Fall3d_ser.log) and confirm the utility was completed successfully.

This completes the modelling process. The results files are now available in the same directory as the input files.

Appendix 7 – Viewing results

Fall3d-6 generates output files in netCDF format with the file extension '*.res.nc'. AIM converts the netCDF format outputs into a number of file formats including ESRI ASCII and Google Earth KML. However, there are a number of open-source freely available graphic browsers which can be used to view the netCDF filed as well as commercial GIS software. We detail the procedure for two options below:

A. OPEN SOURCE – NCBROWSE

ncBrowse is a freely available java application which can be used to view model results in netCDF format. To download ncBrowse:

1. Open an Internet browser and go to the website

   ftp://ftp.epic.noaa.gov/java/ncBrowse/install.htm

2. Click on '**Download Installer for Windows**' at the top of the screen.
3. After downloading double-click on '**install_rell_6.3.exe'** to install ncBrowse.
4. Open ncBrowse.
5. Select **File>OpenFile** and select **merapi.res.nc**

To view the topography file (the underling DEM):

6. Double-click on "**TOPOGRAPHY**" in the 'Select Variable For Display' window.
7. The 'Domain Selector' window will open
8. Check the 'Dependant Variable' boxes are checking correctly - 'x' for x' and 'y' for 'y' (the default is the opposite).
9. Confirm that the UTM coordinates for the DEM are correct.
10. Click '**Graphic Variable**'

To view volcanic ash load or volcanic ash thickness for a particular time step (i.e. end of the scenario):

11. Double-click on "**LOAD**" OR "**THICKNESS**" in the 'Select Variable For Display' window.
12. The 'Domain Selector' window will open
13. Check the '**Dependant Variable**' boxes are checking correctly - '**x**' for **x**' and '**y'** for **'y**' (time is not a dependant variable in this case).
14. Choose a start time step to view by clicking on the square symbol with three dots. This will open a new window that will have a time slider. Select the starting time (usually in hours). Recommendation: View the final time step first and work back from there.
15. Confirm that the UTM coordinates for the DEM are correct.
16. Click '**Graphic Variable**'

You can create maps for each time step for volcanic ash load or volcanic ash thickness. Other outputs also include volcanic ash concentration in the atmosphere.

To view the change in volcanic ash load or volcanic ash thickness over time as an animation:

17. In the '**File Browser**' window, click on '**New Map**'
18. The '**Variable Map Designer**' window will appear.
19. Drag '**x'** from the 'Source' field to the '**X-axis**' box in the 'Target' field.
20. Drag '**y**' from the 'Source' field to the '**Y-axis**' box in the 'Target' field.
21. Drag '**LOAD**' or '**THICKNESS**' from the 'Source' field to the 'Contour' box in the 'Target' field.
22. Give the new map a name in the **Variable Map Name** box at the top of the window.
23. Click '**Done**'
24. You will return the the '**File Browser**' window.
25. Scroll down the '**Select Variable for Display**' field and double-click on the new map created (it will be displayed towards the bottom of the list).
26. The '**Parameter Editor'** window will open.
27. Confirm the UTM coordinates are correct.
28. Set the start time (end of eruption)
29. Click '**Plot**' to create a map that will be used to run the animation.
30. In the '**Parameter Editor**' window click on the animation icon (film strip icon next to the start time). This will open the '**Animation Controller for time (h)**' window.
31. Set the speed for the animation and press '**Play**' to run the simulation using the map created in step 29.

The animation will loop continuously at the speed set by the user.

B. COMMERCIAL SOFTWARE – ARCGIS 9.3

IN PROGRESS