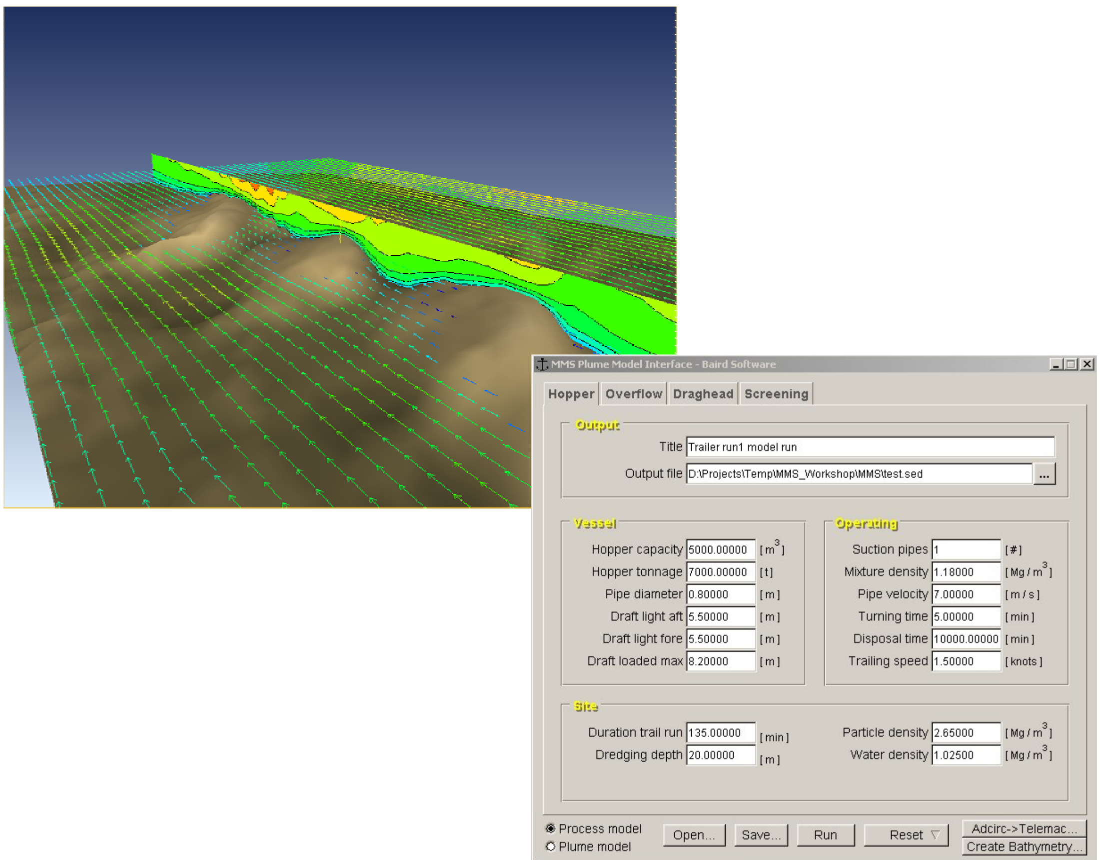
MMS DREDGE PLUME MODEL USER MANUAL



PREPARED FOR:
U.S. DEPARTMENT OF THE INTERIOR
MINERALS MANAGEMENT SERVICE
Under Contract Number No. 01-01-CT-31127

U.S. Department of the Interior
Minerals Management Service

May 2004

DEVELOPMENT OF THE MMS DREDGE PLUME MODEL

Prepared for:

Leasing Division, Sand and Gravel Unit
Minerals Management Service
U.S. Department of Interior
Herndon, Virginia

Prepared by:

W.F. Baird & Associates Ltd.
Madison, Wisconsin

May 2004

DISCLAIMER

This report has been reviewed by the Minerals Management Service and approved for publication. Approval does not signify that the contents necessarily reflect the views and policies of the Service, nor does mention of trade names or commercial products constitute endorsement or recommendation for use.

TABLE OF CONTENTS

LIST OF FIGURES

ACKNOWLEDGEMENTS

This report was prepared under contract to the Sand and Gravel Unit, Leasing Division, Minerals Management Service, Contract No. 01-02-CT-85139. Barry Drucker was the MMS Project Manager and contributed significantly to the project objectives and the overall success of the project. Dr. Rob Nairn of Baird and Associates was the Project Manager. The senior authors were Douglas Scott and Michael Fullarton of Baird & Associates, Jeremy Spearman of HR Wallingford and Nick Bray of Dredging Research Ltd.

1.0 INTRODUCTION

This document describes the setup and use of the MMS Plume Model. The objective of the model development was to implement and test a state-of-the-art numerical model that can be used to predict the plume and sedimentation characteristics associated with both the overspill and draghead disturbance created in dredging operations. The target vessel is a Trailing Suction Hopper Dredge (TSHD). The overall goal of the development program is to make available a software tool that can be used to assess the sedimentation footprint from dredging operations, which may then be utilized to evaluate the physical and biological impacts of dredging operations.

Unique features of the numerical model include:

- A Process model that reproduces the complete sedimentation processes occurring within a TSHD from sediment uptake to overspill. This model can simulate oversize or undersize screening, as required, and the effects of either Constant Volume or Constant Tonnage systems.

- The ability to simulate plumes created by draghead disturbance.

- A Plume model that can simulate three distinct phases of the overspill sedimentation phases:

    - The Dynamic Phase in which the overspill descends through the water column as a density plume.

    - The Passive Phase in which the plume is advected and dispersed through the ambient waters. The model uses a Lagrangian random walk approach to trace the movement of sediment. This approach involves the release of large numbers of individual particles representing sediment mass which move in three dimensions under the action of turbulent diffusion, settling, tidal currents, deposition and re-suspension.

    - The Bed Re-suspension Phase in which sediment already deposited on the bed may become re-suspended by the ambient currents.

- Dredgers are simulated along a user-defined pathways which may be linear or of variable in speed and direction.

- The model allows for the simulation of multiple releases from multiple dredgers (as well as incorporating the facility for multiple additional point source releases).

- Linkages to the widely-used ADCIRC hydrodynamic model supported by the U.S. Army Corps of Engineers. The model interfaces have been prepared in such a way that the model can be readily adapted to many other types of hydrodynamic models in the future.

- Development of a Graphical User Interface which facilitates use of the model and binds together the various model modules.

- Creation of a tool for three-dimensional, animated displays of the plume model, to aid in use and interpretation of the model results.

The ability to simulate all of the plume sedimentation phases is a unique characteristic of the MMS Plume Model not found in any of the currently available models in use. A detailed description of the model physics may be found in the Technical Manual (Baird, 2004).

In the following report sections, the usage, inputs, and outputs for the MMS Plume Model are provided.

2.0 OVERVIEW OF THE MMS PLUME MODEL

The Plume Model is designed in a series of separate modules and software tools, that need to be applied in a specific sequence. In order to undertake a dredging sedimentation analysis at a specific site, the following steps should be followed:

1. Conduct hydrodynamic modelling of the project site for the time period of interest using the *ADCIRC-2D* model.

2. Convert the ADCIRC model result files into a file format suitable for input to the MMS Plume model by means of the *ad2tele3D* utility. There are two hydrodynamic files that need to be input to the Plume Model:

    a. The three-dimensional current field (*.tel file)

    b. The hydrodynamic model boundaries (*.bnd file)

3. Run the *Dredger Process Model*, which simulates the dredger overspill and draghead disturbance processes. This will require the prior creation of a process model steering file (*.prj) that provides the details of the dredger and its operations. The Process Model produces as output two basic log files (*.drg and *.scn) as well as an input file (*.sed) for the Plume Model.

4. Run the *Plume Model*, which simulates the dynamic, passive and re-suspension phases of the dredging process. This will require the prior creation of a trailer path file that provides the time and position of the trailer in space, and a plume model steering file that gives data about the soil characteristics and dredger to the model.

Figure 1 provides a flow chart for the overall model use.

A Graphical User Interface (see Section 8.0) has been developed to facilitate creation of the various model steering files. A complete list of the available steering file commands is provided in Appendices I and II.

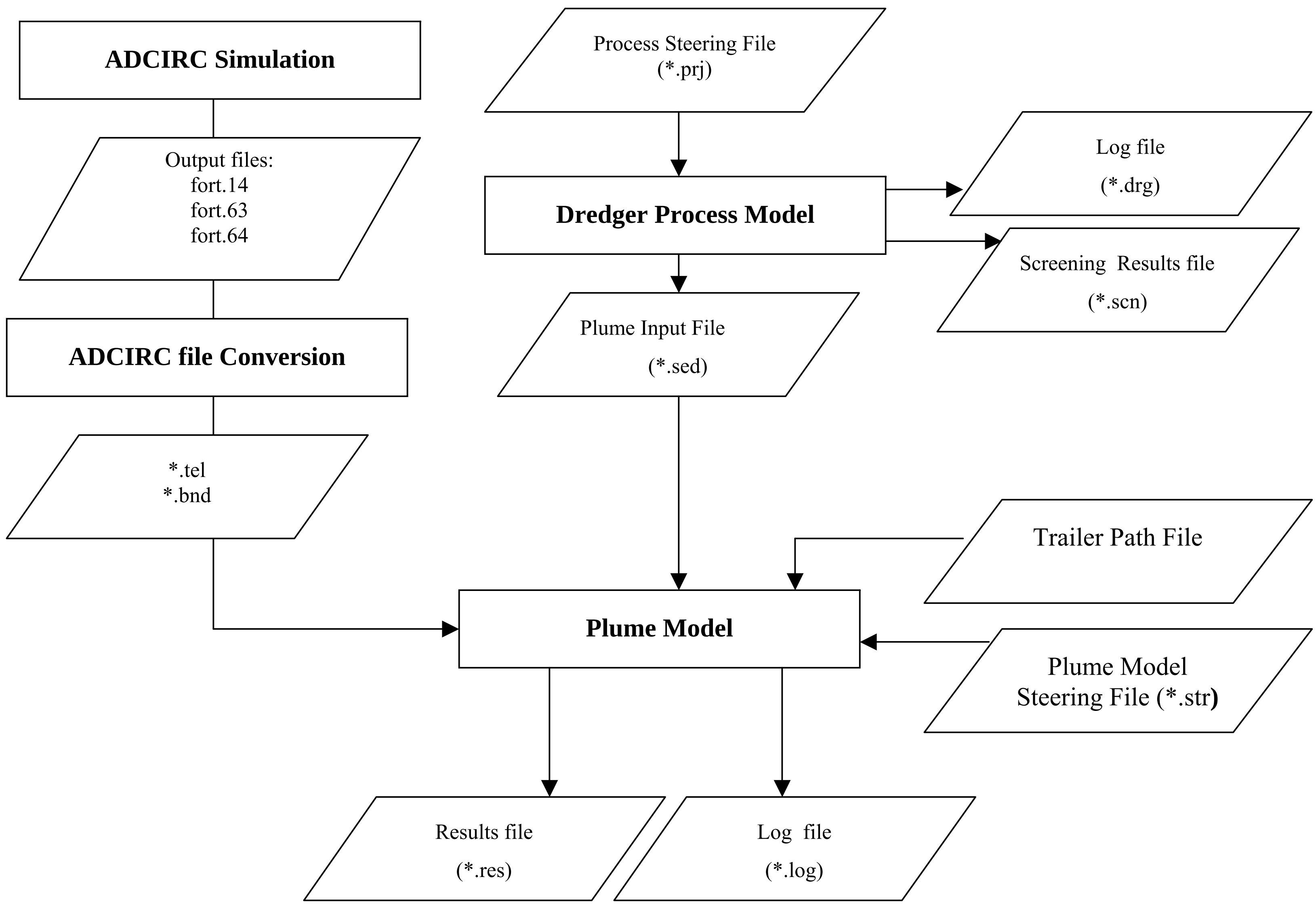


FIGURE 1. Flow Diagram for the Plume Modeling Process

3.0 BASIC DATA REQUIREMENTS FOR MODEL USE

The following is an outline of the basic types of information needed in order to simulate the TSHD plume processes:

TSHD Dimensions

- Hopper capacity
- Hopper tonnage
- Hopper length
- Hopper width
- Hopper depth
- Hopper draught (light and loaded)
- Number of suction pipes
- Suction pipe diameter

Site Characteristics

- Dredging depth
- Water density
- In-situ soils grain size curve

Operating Parameters

- TSHD path
- Duration of the trailing run
- Vessel turning time between runs
- Vessel disposal time when full
- Pre-load volume
- Use of Constant Tonnage or Constant Volume system
- Incoming mixture density
- Settled mixture density in hopper

Draghead Properties (if needed)

- Draghead length
- Draghead width
- Draghead depth of cut
- AV in-situ density

Screening Properties (if needed)

- Screeen aperture size
- Screen void ratio
- Screen angle
- Screen discharge point (hull or shipside)

Soil Characteristics

- Minimum settling velocity or settling factor
- Critical stress for erosion
- Critical stress for deposition
- Erosion rate

Current Field

- 3D current field encompassing area of interest developed in the ADCIRC model
- Bottom roughness length

4.0 ADCIRC MODEL USAGE

The MMS Plume Model has been designed to utilize output from the finite element hydrodynamic model ADCIRC as input to the plume dispersion processes. In this study, the two-dimensional version of ADCIRC has been utilized.

Usage of the ADCIRC model will is not covered in this User's Guide. Full details on utilization of ADCIRC may be found in either Leuttich and Westerink (2000) or an on-line users manual available at:

http://www.marine.unc.edu/C_CATS/adcirc/document/ADCIRC_main_frame.html

Leuttich and Westerink (2003) provides a complete description of the theory and equations behind the ADCIRC model.

The original H.R. Wallingford version of the plume model links only to the Telemac hydrodynamic model. A software utility, called *ad2tele3d.exe,* was developed in this project that converts the ADCIRC output files into Plume Model input file format and also transforms the 2D ADCIRC currents into a three-dimensional flow field based on a user-supplied bottom roughness value. This utility has been incorporated into the model Graphic User Interface (see Section 8.0) but may also be run manually as follows:

Ad2tele3d fort.14 fort.64 fort.63 file.tel file.bnd nlayers rough

where:

fort.14 is the ADCIRC grid file

fort.63 is the ADCIRC water surface elevation file

fort.64 is the ADCIRC currents file

file.tel is the flow field file input to the Plume Model

file.bnd is the flow field boundary file input to the Plume Model.

nlayers is the number of vertical layers that the users wishes to use in the plume model analysis. A typical value might in the order of 10.

rough is the specified bottom roughness value.

5.0 MMS MODEL USAGE AND STRUCTURE

5.1 Introduction

The MMS model is a sophisticated plume dispersion model designed to reproduce the release of fine sediment from trailer suction hopper dredging and the resulting dispersion of this sediment under the action tidal currents. The model reproduces the release of sediment into the water column that occurs as a result of the dredging process and the subsequent (three-dimensional) dynamic and passive phases of dispersion under the action of tidal currents.

This section of the document describes the model installation, interactions and input file structure.

5.2 Model Installation

The modeling system is installed from CD by double-clicking on the Install program. The programs are installed by default in the C:\Program Files\MMS Plume directory, and icons are created on the desktop.

5.3 Model Structure and Modes of Use

As noted in Section 2.0, the model is composed of the following modules:

- The dredger process model – this reproduces the removal of in situ sediment, the settling of material within the hopper and the loss of the finer fractions to the water column as a plume. This model has been developed by Dredging Research Ltd. of the U.K.

- The dynamic plume model (see Chapter 5 of the Technical Report) – this reproduces the initial (dynamic) phase of dispersion whereby the dispersion of the fine sediment plume is dominated by the nature and conditions of release. The dynamic plume model has been developed by HR Wallingford and incorporated into the HR Wallingford SEDPLUME dispersion model system.

- The passive plume model (see Chapter 6 of the Technical Report) – this reproduces the second (passive) phase of dispersion whereby the dispersion of the plume is a function of the ambient tidal conditions. The passive plume model is based on the HR Wallingford SEDPLUME-RW(3D) model.

- The visualization system – this is a state-of-the-art 3D graphics system for presenting the output from the model. This system has been developed by Baird based on an existing Unix data animation tool.

In terms of the overall Plume Model structure there are really three main parts as the dynamic and passive plume models are coded as a single model, which for we shall henceforth refer to as the *MMS Plume Model*. The main computational activity takes place within this model, and in particular within the passive plume routine. The Dredger Process Model and the Dynamic Plume Model can be thought of as providers of source terms to the passive plume model while the visualization system is essentially a means to display the dispersion of the passive plume.

The model can be run in basically two modes:

- By means of the Graphical User Interface, which provides a strict sub-set of the available keywords and abilities. Key restrictions include the inability to simulate stationary point source discharges and a dredging path file must be provided as input.

- By means of the manual editing of the steering files. In this manner, the complete list of steering file commands (Appendices I and II) are available to the user.

5.4 Steering File Structure

The steering files for both the Process and Plume Models are simple ASCII files containing keywords and associated values with the keywords. A small section of a typical steering file is shown below:

```
/Hopper
TITLE = Test screen model run
SEDPLUME FILE = plumefile.sed
DREDGER RESULTS FILE = plumefile.drg
HOPPER CAPACITY = 11750.00000
HOPPER TONNAGE = 16774.00000
SUCTION PIPE DIAMETER = 0.60000
```

The following points are important as regards the general format of the steering file:

- Keywords must be in upper case and the values are assigned using either '=' or ':'.

- Where two or more releases/trailers are to simulated, parameters must be separated by a semi colon, e.g. TRAILER TURNING POINT X1 = 827697.0 ; 826697.0

- Keywords can be entered in any order.

- Any line starting with a forward slash '/' is considered a comment line and not considered in the computations.

- File names embedded in the steering file cannot have spaces.

- At the end of the steering file the following line should appear (otherwise the PC program may crash),

END

5.5 Interaction between the Hydrodynamic Model and the Plume Model

The MMS Plume model requires flow model input to drive the dispersion of the plume. The original HR Wallingford model was designed to read output from the TELEMAC hydrodynamic model; however, the MMS model supports the use of both the ADCIRC 2D/3D flow models and the TELEMAC 2D flow model. By the use of the standard assumption regarding the logarithmic variation of velocity through the water column, ADCIRC 2D results are converted to 3D for use in the plume model.

The MMS model tracks the 3D movement of released sediment particles using the flow model output and with relation to the flow model grid. However, the output of suspended sediment concentrations and bed deposition is based on a user-defined grid which can be a subset of the flow model area. This enables efficient storage of output information since only relevant information about the area impacted by the plume is stored. Figure 2 illustrates the difference between the flow model and output grids.

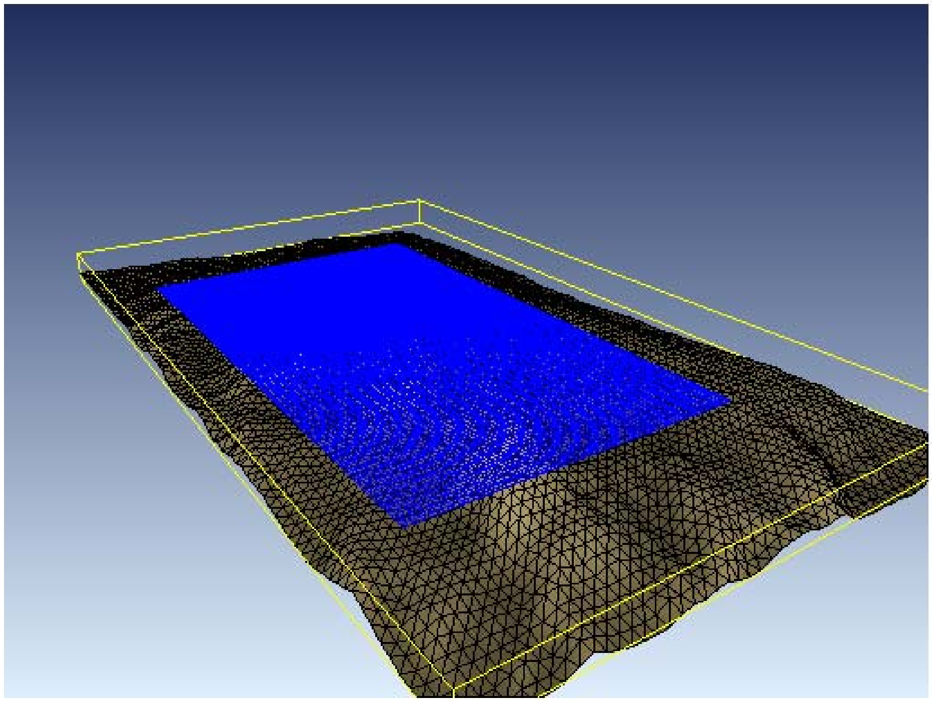


FIGURE 2. The Plume Model Rectangular Grid (in blue) Shown Embedded in the Larger ADCIRC Grid (in black)

6.0 DREDGING PROCESS MODEL USAGE

6.1 Overall Model Structure

The following is a brief overview of the key processes involved in running the Dredger Process model. All key information is defined in a single project file an example of which is shown below in Section 6.2.

6.2 Description of Steering File

A sample steering file is shown below:

```
/Hopper
TITLE = Test screen model run
SEDPLUME FILE = plumefile.sed
DREDGER RESULTS FILE = plumefile.drg
HOPPER CAPACITY = 11750.00000
HOPPER TONNAGE = 16774.00000
SUCTION PIPE DIAMETER = 0.60000
DRAFT LIGHT AFT = 4.45000
DRAFT LIGHT FORE = 3.45000
DRAFT LOADED MAX = 9.40000
TRAILING SPEED = 1.00000
SUCTION PIPE NUMBER =  1
MIXTURE DENSITY = 1.20000
SUCTION PIPE VELOCITY = 5.45000
TURNING TIME = 5.00000
DISPOSAL TIME = 120.00000
DURATION TRAIL RUN = 16.00000
TIME INCREMENT = 1.00000
DREDGING DEPTH = 11.00000
PARTICLE DENSITY = 2.65000
WATER DENSITY = 1.02500

/Overflow
OVERFLOW ON = 1
HOPPER LENGTH = 57.17000
HOPPER WIDTH = 18.00000
HOPPER DEPTH = 11.42000
CONSTANT TONNAGE = 1
/CONSTANT VOLUME = 1
WATER KINEMATIC VISCOSITY = 1.36e-006
SOIL RATIO LOST = 1.00000
PRELOAD VOLUME = 1.00000
SETTLED DENSITY = 1.90000
SOILS = 5
10.000       0.000
30.000       10.000
100.000      20.000
1000.000     40.000
75000.000    100.000
FINE SOILS SUMMED = yes
```

```
/Screening
SCREENING RESULTS FILE = plumefile.scn
SCREEN SIZE INDEX = 2.00000
SCREEN FRACTION INDEX = 1.00000
SCREEN FLOW INDEX = 2.00000
SCREEN DENSITY INDEX = 1.50000
SCREEN DISCHARGE OUTPUT = HULL
OVERFLOW OUTPUT = HULL
/UNDERSIZE SCREEN ON = 1
/UNDERSIZE APERTURE SIZE = 2000.00000
/UNDERSIZE VOID RATIO = 0.50000
/UNDERSIZE SCREEN ANGLE = 30.00000
/UNDERSIZE WATER LOSS = 1.20000
OVERSIZE SCREEN ON = 1
OVERSIZE APERTURE SIZE = 10000.00000
OVERSIZE VOID RATIO = 0.90000
OVERSIZE SCREEN ANGLE = 30.00000
OVERSIZE WATER LOSS = 0.10000
```

The project file shown above is setup to simulate a constant tonnage dredging operation for a dredger with hopper dimensions (L x W x D) of 57.17 m x 18.0 m x 11.42 m and a hopper capacity of 11750 m^3. One trailing arm with a pipe size of 0.6 m and a pipe velocity of 5.45 m/s was defined and the density of the mixture in the pipe is specified as 1.2 tonnes/m^3. Other vessel characteristics include the trailer speed, duration trail run, turning time and disposal time which were defined in the project file above as 1.0 knots, 16.0 minutes, 5.0 minutes and 120 minutes respectively.

The *in situ* particle size distribution is defined in the project file by the percent passing of 5 different soil sizes (in µm). OVERSIZE SCREENING is activated therefore particles greater than the oversize aperature size, which in this case is 1000 µm, are rejected. Although undersize screening is shown it has been commented out, therefore it is not activated.

Output for this particular simulation includes a dredger result file, Sedplume file and if screening is activated then a screening results file. Each output file is discussed in more detail in the following sections.

Dredger Results File (.drg)*

The dredger file is the main output file for the Process Model. It contains a summary of the information in the project file as well as a detailed description of various aspects of the dredging process including total weight in hopper (Mg), Overflow Load Rate (Mg/min), Overflow Weight Saved (Mg/min), Total Weight of Overflow Saved (Mg), Settling Efficiency per particle and Depth Averaged Velocity per Particle to name a few. The units here are Megagrams (10^6 grams), equivalent to a Metric Tonne.

Sedplume File (.sed)*

The Sedplume File is generated by the Process model and is then used as input into the MMS Plume model. It contains information pertaining to the dredger overflow including the flow rate (m^3/s) as well as the mass rate in kg/s for each particle size discharged back into the coastal region. An example of the Sedplume file is presented below.

```
DRAGHEAD = NO
SCREENING OUTPUT = HULL
OVERFLOW OUTPUT = SHIP SIDE
FINE SOILS SUMMED = YES
 4
     10.000
     40.000
     70.000
     90.000
      1.000      0.622      5.319      0.000      0.000      0.000
      2.000      0.622      5.319      0.000      0.000      0.000
      3.000      0.622      5.319      0.000      0.000      0.000
      4.000      0.622      5.319      0.000      0.000      0.000
      5.000      0.622      5.319      0.000      0.000      0.000
      6.000      0.622      5.319      0.000      0.000      0.000
      7.000      0.622      5.319      0.000      0.000      0.000
      8.000      0.622      5.319      0.000      0.000      0.000
      9.000      0.622      5.319      0.000      0.000      0.000
     10.000      0.622      5.319      0.000      0.000      0.000
     11.000      0.622      5.319      0.000      0.000      0.000
     12.000      0.622      5.319      0.000      0.000      0.000
     13.000      0.622      5.319      0.000      0.000      0.000
     14.000      0.622      5.319      0.000      0.000      0.000
     15.000      0.622      5.319      0.000      0.000      0.000
     16.000      0.622      5.319      0.000      0.000      0.000
     17.000      0.622      5.319      0.000      0.000      0.000
     18.000      0.622      5.319      0.000      0.000      0.000
     19.000      0.622      5.319      0.000      0.000      0.000
     20.000      0.622      5.319      0.000      0.000      0.000
     21.000      0.622      5.319      0.000      0.000      0.000
     22.000      0.622      5.319      0.000      0.000      0.000
     23.000      0.622      5.319      0.000      0.000      0.000
     24.000      0.622      5.319      0.000      0.000      0.000
     25.000      0.622      5.319      0.000      0.000      0.000
     26.000      0.622      5.319      0.000      0.000      0.000
     27.000      0.622      5.319      0.000      0.000      0.000
     28.000      0.622      5.319      0.000      0.000      0.000
     29.000      0.622      5.319      0.000      0.000      0.000
     30.000      0.622      5.319      0.000      0.000      0.000
     31.000      0.622      5.319      0.000      0.000      0.000
     32.000      0.622      5.319      0.000      0.000      0.000
     33.000      0.622      5.319      0.000      0.000      0.000
     34.000      0.622      5.319      0.000      0.000      0.000
     35.000      0.622      5.319      0.000      0.000      0.000
     36.000      0.622      5.319      0.000      0.000      0.000
     37.000      0.622      5.319      0.000      0.000      0.000
     38.000      0.622      5.319      0.000      0.000      0.000
     39.000      0.622      5.319      0.000      0.000      0.000
     40.000      0.622      5.319      0.000      0.000      0.000
     41.000      0.622      5.319      0.000      0.000      0.000
```

There is four grain sizes defined in the Sedplume file (i.e. 10, 40, 70, 90 µm). The first column below the grain size information is the time increment in minutes followed by the overflow rate (m^3/s) and finally the mass rate in kg/s for each particle size. For example the particle size of 10 µm has a mass rate equal to 5.319 kg/s, however particle sizes 40, 70, and 90 µm have mass rates of 0.0 kg/s.

Screening Results File (.scn)*

This file summarizes the efficiency of the screening process by specifying the % rejected and/or the % into hopper for each particle size. An example of an undersize screening file is presented below.

```
Screen Process Model
=============================
Title :  Trailer run1 model run

Model includes the following sub-process types:-
   Undersize screening

Discharge from suction pipe
===========================

Mixture flow (cu.m/sec) =  3.52
Diameter of suctionpipe (m) = 0.80
Density of mixture (Mg/cum) = 1.25

Screen characteristics
======================

Undersize screening
===================

Screen size (mu) = 2000.00
Ratio of void to total = 0.50
Screen angle degrees =  0.00
Screen length (m) =  2.00
Screen width (m) =  1.50
Water loss constant =  1.20

Screening index
===============

Index W = 2.00
Index X = 1.00
Index Y = 2.00
Index Z = 1.50

Data Computed for screening processes
=====================================

Proportion greater than undersize screen (fraction) = 0.34
Water loss during undersize screen (cum/s) = 2.52
Percentage water loss during undersize screen (% of inflow) = 71.69
Water flow after undersize screen (cum/s) =  1.00
Density of mixture after undersize screen (Mg/cum) = 1.48

  Single Size  % in situ   Undersize   % in situ   % in situ   % in stream
  (mu)                     to hopper   to hopper   to reject   to hopper

       10.00        0.00       28.31        0.00        0.00        0.00
       40.00        1.00       28.33        0.28        0.72        0.49
       70.00        3.00       28.38        0.57        1.43        0.98
       90.00        4.00       28.43        0.28        0.72        0.49
      125.00        6.30       28.53        0.66        1.64        1.13
      175.00       12.00       28.75        1.64        4.06        2.83
      250.00       23.10       29.22        3.24        7.86        5.61
      350.00       31.60       30.09        2.56        5.94        4.42
      500.00       40.90       31.94        2.97        6.33        5.14
      800.00       52.60       37.62        4.40        7.30        7.61
     1000.00       57.30       42.86        2.01        2.69        3.48
     1500.00       66.20       61.04        5.43        3.47        9.39
     3000.00       77.40      100.00       11.20        0.00       19.36
     5000.00       85.50      100.00        8.10        0.00       14.00
     8000.00       90.40      100.00        4.90        0.00        8.47
    15000.00       95.30      100.00        4.90        0.00        8.47
    25000.00       98.00      100.00        2.70        0.00        4.67
    50000.00       99.50      100.00        1.50        0.00        2.59
    75000.00      100.00      100.00        0.50        0.00        0.86
```

7.0 PLUME MODEL USAGE

7.1 Description of Steering File

A sample steering file is shown below:

```
/------------------- FILES ---------------------------------------------------
FLOW RESULTS FILE = flow.tel
OUTPUT FILE = out.res
BOUNDARY FILE = flow.bnd
LOG FILE =  out.log
RELEASE FILE = dredgerfile

/------------------ PROGRAM PARAMETERS -------------------------
TIMESTEP = 60.0
NUMBER OF TIMESTEPS = 1500
STORAGE INTERVAL = 30
START TIME = 61200.0            / start of file
MINIMUM SETTLING VELOCITY = 0.001
SETTLING VELOCITY FACTOR = 0.01
SETTLING VELOCITY POWER = 1.0
DIFFUSION = 0.2
CRITICAL STRESS FOR DEPOSITION = 0.10
CRITICAL STRESS FOR EROSION = 0.11
EROSION RATE = 0.0002
ROUGHNESS LENGTH = 0.01

/Keyword below gives expansion coefficients for 1st & 2nd stored active
tracers
/Set values to zero when T and/or S not stored
/Values are not used if density values are stored
EXPANSION COEFFICIENTS = 0.0; 0.0
REFERENCE DENSITY = 1000.0

TIME OF HIGH WATER = 61200.0
TIDAL PERIOD = 86400.0

/------------------ SQUARE GRID PARAMETERS --------------------
SQUARE GRID = YES
SQUARE GRID ORIGIN X = 822800.0
SQUARE GRID ORIGIN Y = 817100.0
SQUARE GRID X POINTS = 70
SQUARE GRID Y POINTS = 80
SQUARE GRID SIZE = 100
SQUARE GRID ORIENTATION = 0.

/------------------ OUTPUT --------------------
OUTPUT LAYERS = 1;4
OUTPUT DEPOSITS = yes
OUTPUT TIDE AVERAGE SSC = yes
VERTICAL INFO = yes
LOG ADDITIONAL PLUME INFORMATION = yes

/------------------- POINT SOURCE DATA ---------------------------------------
NUMBER OF POINT SOURCES = 1
```

```
POINT SOURCE LOCATION X = 821068.0
POINT SOURCE LOCATION Y = 819654.0

/------------------ DYNAMIC PLUME PARAMETERS --------------------
NUMBER OF TRAILERS = 2
TRAILER FRACTION = 1
INCLUDE DYNAMIC PLUME = yes
CONSTANT TRAILER RELEASE = no
TRAILER SCREEN AREA = 1.0 ; 1.0
TRAILER OVERFLOW AREA = 8.0 ; 8.0
TRAILER HULL AREA = 2.0 ; 2.0
TRAILER DRAUGHT = 7.0 ; 7.0
TRAILER SPEED = 0.5 ; 1.0
TRAILER TURNING POINT X1 = 827697.0 ; 826697.0
TRAILER TURNING POINT Y1 = 819654.0 ; 819654.0
TRAILER TURNING POINT X2 = 826283.0 ; 825283.0
TRAILER TURNING POINT Y2 = 821068.0 ; 821068.0
/TRAILER PATH FILE 1 = wiggly_path_1
/TRAILER PATH FILE 2 = wiggly_path_2
TRAILER UPDATE TIMESTEP = 900.0
TRAILER PROCESS RESULTS FILE 1 = mssSedplm1.uni
TRAILER PROCESS RESULTS FILE 2 = mssSedplm2.uni
/-----------------------------------------------------------------------------------
END
/--------------------------------------------------------------------------------
```

The suspended sediment concentrations will be computed on a 100m square grid, with 70 columns and 80 rows. This output grid will be aligned with the geographical grid on which the model is based, and the southwest corner will be at (822800.0, 817100).

The flow results are read from file 'flow.tel'. The output will be written to 'out.res'. A time-step of 60 seconds has been chosen, and the horizontal turbulent diffusivity has been set to $0.2m^2.s^{-1}$. The timing of releases is read from file 'dredgerfile'. The movement of the dredger is read from file 'dredgepath' and the dredger process files for each of the trailer dredgers are 'mssSedplm1.uni' and 'mssSedplm2.uni'.

The total number of time steps is set to 1500, thus specifying the overall length of the program run at 1500x60 seconds = 25 hours (2 semi-diurnal tides). Results will be stored every 30 time steps, that is, every half an hour.

It is important to note that if the length of the program run is specified to be greater than the period covered by the flow results, the program will go back to the start of the flow file and continue. This makes it easy to use a single tide of flow results several times for a long SEDPLUME-RW simulation. If the flow results do not constitute a repeating tide, however, then the random walk simulation length should be limited to the length of the flow results.

The sediment properties specified in the example steering file define a constant settling velocity of $1mm.s^{-1}$, because this is the value set under the MINIMUM SETTLING VELOCITY keyword, and the specification of a value of 0.0 under the SETTLING

VELOCITY FACTOR keyword causes settling velocities calculated using equation (2) to be zero. Deposition can occur when the computed bed shear stress falls below $0.1 N.m^{-2}$, and re-suspension can begin when the stress exceeds $0.11 N.m^{-2}$. The erosion constant is here set to $0.0002 kg.N^{-1}.s^{-1}$.

Note the format of the example steering file. Keywords must be in upper case and their values are assigned using '=' or ':' interchangeably. Any line beginning with '/' or text contained between two '/' is treated as a comment. The keywords can be in any order; the sections in the file above are just one example of how to organise the entries. Where multiple data entries can be used with a keyword, for example TRAILER SPEED, under which a release can be specified for up to 10 sources, the entries should be separated by ';'.

<u>Point source releases only</u>

It is possible to switch point sources (only) on and off during a run in the steering file using the keywords NUMBER OF RELEASE PERIODS, START RELEASE, STOP RELEASE and RELEASE EVERY TIDE. The maximum permitted value for NUMBER OF RELEASE PERIODS is 10, and the array keywords can thus have up to 10 values. The values of START RELEASE and STOP RELEASE are times given in hours after the start of the run. Example the keyword settings are as follows:

NUMBER OF RELEASE PERIODS = 2

START RELEASE = 2.0;6.0

STOP RELEASE = 3.0;8.0

RELEASE EVERY TIDE = no

These settings define a release which is switched off until 2 hours into the run, then switched on for an hour, and switched off again until 6 hours into the run, at which time another two hours of discharging commences. The source would then remain switched off until the end of the simulation.

To model a periodic release, for instance a one-hour release beginning each High Water, the RELEASE EVERY TIDE keyword can be set to 'yes' (this assumes that the flow model results files contain a single tide of data). In this case the flow results will be re-used if the MMS PLUME MODEL run passes the end of the flow file as defined by the first time in the file plus the time given under TIDAL PERIOD. Consequently, if the reference High Water is at the start of the flow run, the following keyword settings should be used:

NUMBER OF RELEASE PERIODS = 1

START RELEASE = 0.0

STOP RELEASE = 1.0

RELEASE EVERY TIDE = yes

Note that, if there are several sources, they will all be switched on and off in unison by these keywords. If sources are to be simulated independently of each other, a load file would be required). During the periods of suspended sediment release, the particles are emitted at a constant rate specified by the usual keywords.

However, if trailer dredgers are specified the release file must always be used.

Dynamic plume parameters

The steering file input relates to two trailer dredgers each with a draft of 7m and release areas relating to screening and overflow of $2m^2$ and $8m^2$ respectively. The speed of the dredgers over the ground is different – dredger 1 will move a 0.5m/s while dredger 2 will move at the faster speed of 1.0m/s. The ends of the dredging run for each dredger are specified by the TURNING POINT values of X1, X2, Y1 and Y2. Alternatively the movement of the dredgers could be read from the files WIGGLY_PATH_1 and WIGGLY_PATH_2 but in this case these commands have been commented out.

The only unser-defined run-time parameter for the dynamic plume relates to the TRAILER UPDATE TIMESTEP which controls how frequently the dyanamic plume routine is called. It is called at the beginning of a dredge run (as specified in the dredger file), it is called if the dredger changes direction and called at the end of every period specified by TRAILER UPDATE TIMESTEP (in seconds). For normal conditions 900 seconds is a suggested upper limit for this period, although circumstances will vary from location to location. Calling the routine at every time step will cause the model to slow down signficiantly.

Note that at the end of the steering file two lines must be present with the word ‘END’.

A full list of the keywords associated with the dispersion model steering file is shown in Appendix II.

7.2 Description of Release file

This file which must have the following format:

Number of records, n, for the first source
Time(1) Loading rate Particles per hour

Time(2) Loading rate Particles per hour
   .   .   .
   .   .   .
Time(n) Loading rate Particles per hour

Number of records for the second source
Time(1) Loading rate Particles per hour
   .   .   .
etc.

Note that the times are in hours relative to the reference high water. If TIME OF HIGH WATER is specified then this relates to the specified time, otherwise it relates to time zero. **Note also that TIME OF HIGH WATER can be any useful time** and the title of this parameter only refers to high water as this is a common reference time. The source data are treated as a step function, so that, for example, the data at time(1) are used constantly from time(1) to time(2). An example of a load file is given below. The maximum number of records (or break points) for each source is 200.

```
  2
  0.00        0.0        0
  1.00       50.0     1000
  5
  0.00        0.0        0
  1.00        1.0     1000
  5.00        0.0        0
 13.00        1.0     1000
  5.00        0.0        0
  5
  0.00        0.0        0
  2.00        1.0     1000
  4.00        0.0        0
  6.00        1.0     1000
  8.00        0.0        0
```

Three releases will be simulated (as specified in the steering file, one outfall release and 2 trailer dredgers). The first is a point source release which will occur at a rate of 50kg/s and at 1000 particles per hour, (which is equivalent to 16 particles per timestep since, in this case, there are 60 time-steps per hour). Note that the number of particles released per timestep is always an integer, rounded down from the number of particles per hour divided by the number of timesteps per hour.

The second and third releases are trailer dredgers and thus the rates of release are governed by the Dredger Process Results files which are specified for each trailer in the steering file. To indicate that release occurs, however, a non-zero discharge rate is still required. For each dredger a release rate of 1000 particles per hour (16 per time step is specified).

7.3 Description of the Trailer Path File

The format of the trailer path file is:

Time(1) x1 y1
Time(2) x2 y2
. . .
. . .
Time(n) xn yn

One dredgepath file is required for each trailer dredger simulated.

7.4 Description of Output File (*.res)

An output file is created during the model which contains the prediction of suspended sediment concentrations and deposition over time and space. This file is a binary files in the TELEMAC 'LEONARD' format. The description of this format is as follows (EDF-1997):

- 1 record containing the title of the study (80 characters),
- 1 record containing the two integers `NBV(1)` et `NBV(2)` (number of variables and the number 0),
- `NBV(1)+NBV(2)` records containing the names and units of each variable (on 32 characters),
- 1 record containing the integers `IM,JM,IM,JM,1` (number of columns and lines of the mesh),
- 1 record containing the integers 0,1,0,0,0,0,0,0,0,0 (10 integers, of which only the second is currently used),
- 1 record containing table `X` (real array of dimension `NPOIN` containing the abscissae of the points),
- 1 record containing table `Y` (real array of dimension `NPOIN` containing the ordinates of the points),
- 1 record containing table `INDIC` (integer array of dimension `NPOIN`) ; `INDIC` has the value 0 for an external point of the domain, it is negative for an internal point of the domain and positive for a boundary point,

Next, for each time step, there are the following:

- 1 record containing time `T` (real),
- `NBV(1)+NBV(2)` records containing the results tables for each variable in time `T`.

This file is written during the progress of the simulation so that if the model terminates prematurely a partial results file is still available for inspection.

If the keyword OUTPUT TIDE AVERAGE SSC is set to “yes” a time-averaged output file is produced in addition to the results file.

7.5 Description of Log file

An output file is created which gives information on the progress of the run and lists any error messages occurring of the model simulation is prematurely terminated. This is known as the Log file as in ASCII in format. The basic form of this file records the information given in the steering file and records the completion of each time step and the number of particles released. If required additional information can be recorded. Keyword VERTICAL INFO allows the saving of information about the distribution of sediment particles through the different layers of the model. Keyword LOG ADDITIONAL PLUME INFORMATION allows the recording of information from the dynamic plume phase.

7.6 Good practice in applying the model

7.6.1 Number of particles

The choice of how many particles to use is a compromise between run time and the accuracy/resolution (Section 2.6) of the results. The maximum number of particles allowed is 1,000,000. Ideally, in any MMS PLUME MODEL application, sensitivity tests would be carried out to ensure that the particle release rate is sufficiently high for the model results to be insensitive to further increases in release rate.

7.6.2 Choice of timestep

In general, smaller timesteps are preferable to larger ones in terms of the quality of the results, but this leads to longer run times. Ideally, in any new MMS PLUME MODEL application, sensitivity tests would be carried out to ensure that the timestep is sufficiently small for the model results to be insensitive to further timestep reductions. In practice, this will generally be the case if the criteria discussed below are satisfied.

One factor limiting the timestep is the size of the random steps in the vertical when compared to the depth of water. For example a current speed of $0.5 m.s^{-1}$ in 10m of water would lead to a maximum (undamped) diffusivity of approximately $0.02 m^2.s^{-1}$ in the centre of the water column, and so a timestep of 60 seconds gives random steps of approximately 1.5 m, which is about the largest acceptable in this case. (Closer to the surface and bed these random steps would be smaller). Deeper water allows longer timesteps, and higher current speeds require shorter timesteps. As a rule of thumb, it is recommended that the following condition is approximately satisfied:

$$\Delta t \leq 4d/U \qquad (1)$$

where U is the maximum depth-averaged velocity, Δt is the timestep and d is the water depth. Small bed roughness values tend to relax the timestep limitation and large bed roughness values make it more severe, but equation (7) is adequate for general guidance.

In addition to equation (7), the following criterion should also be satisfied, in order for horizontal particle steps to be less than the output grid dimension, Δ, thus preventing excessive plume fragmentation:

$$\Delta t \leq \Delta/U \qquad (2)$$

7.6.3 Size of Model Grid

The minimum concentration, c_{min} (kg.m^{-3}), that can be resolved by SEDPLUME-RW is that equivalent to one model particle in a cell of the output grid. This can be calculated from:

$$c_{min} = P/\Delta^2 dM \qquad (3)$$

where d is the total water depth (m)
P is the total amount of suspended sediment released (kg)
Δ is the output grid cell dimension (m)

There is therefore a trade-off between the number of particles that can be represented in the model (which effectively controls the mass represented by any one particular particle), the size of the model grid and the resolution of changes in suspended sediment concentration.

7.6.4 Parameters and Coefficients Not Under User Control

A number of parameters have been hard-coded into MMS PLUME MODEL to define array sizes for particular variables. These parameters have been set to values which do not limit general use of the model, yet do not result in excessive program size and memory requirements. These array size parameters are described below:

Max. number of elements (per layer) in TELEMAC-3D grid = 50,000
Max. number of model points (per layer) in TELEMAC-3D grid = 25,000
Max. number of storage times in flow results file = 250
Max. number of particles released = 1,000,000
Max. number of sources = 10
Max. number of records per source in load file = 200
Max. number of cells in square grid = 100,000
Max. number of model layers = 15
Max. number of sediment fractions = 20
Max. number of times (per dredger) in dredgerfile/dredgepath files = 2000

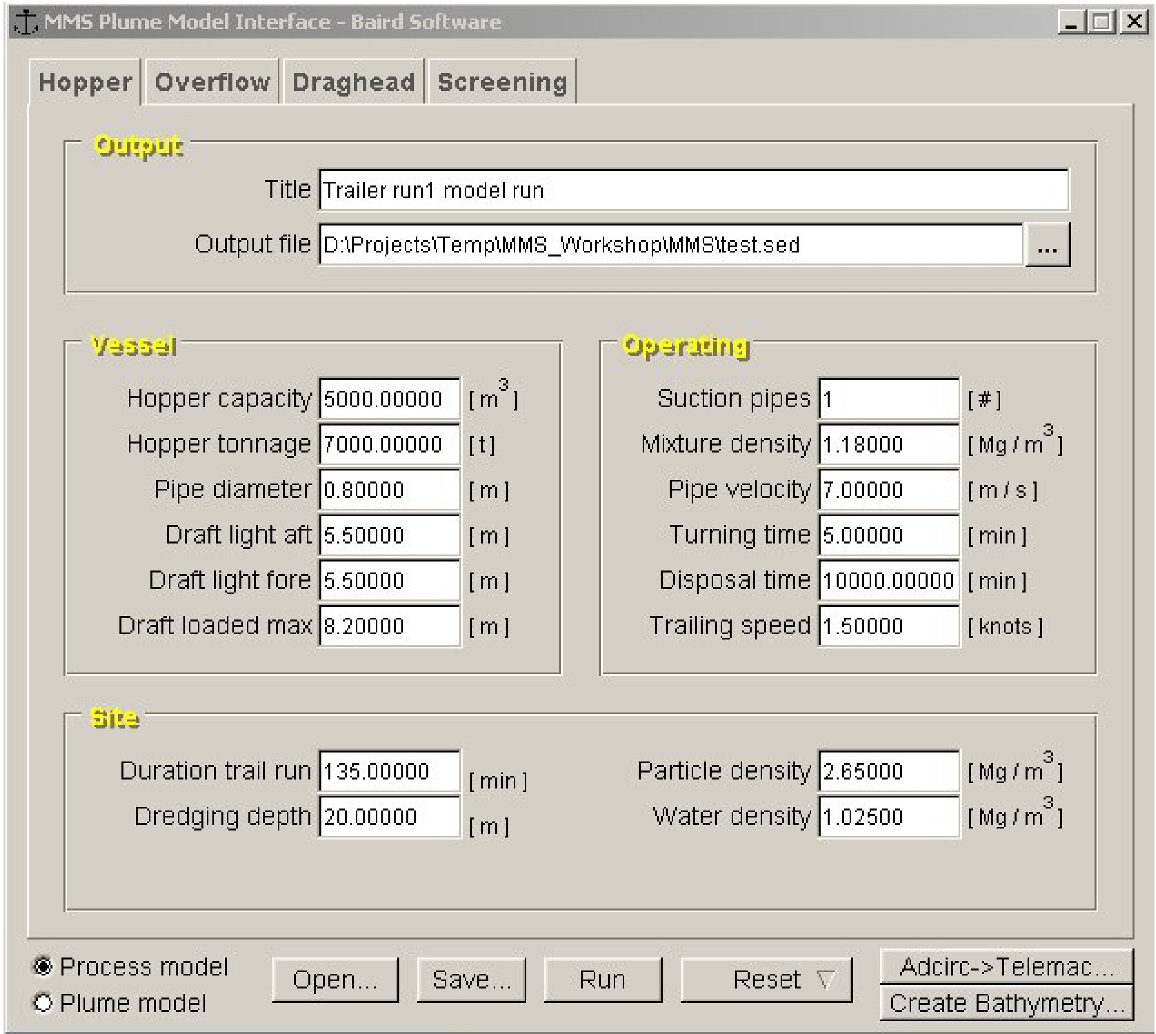
8.0 MMS PLUME MODEL GRAPHICAL USER INTERFACE

8.1 Introduction

A Graphical User Interface (GUI) was developed to facilitate the input data entry necessary for the Plume Modeling System. It is important to note that the GUI produces a restricted sub-set of steering file keywords, and is designed for use in a typical dredging scenario.

The GUI is divided into two parts, for the Process and Plume models, triggered by a radio button.

8.2 The Process Model



On this menu, all of the parameters related to the TSHD dimensions, its operating characteristics and the dredging site are entered.

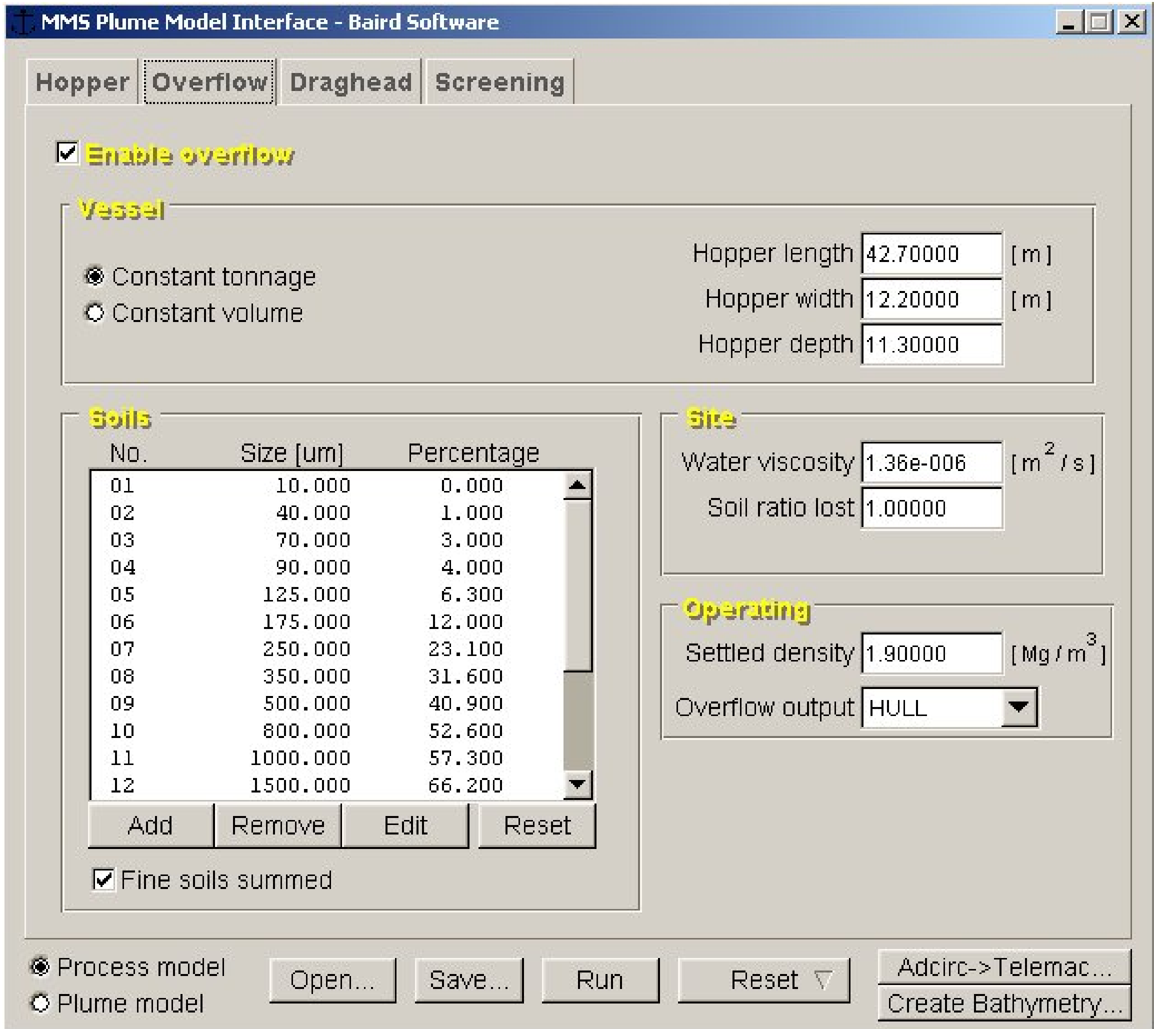
Notes:

- The output file is the file created by the process model that than is fed into the plume model.
- The *Open* button permits the opening of a previously saved process model steering file. The *Save* button allows the steering file to be save without running the model.
- The *Run* button runs the process model with the current inputs.

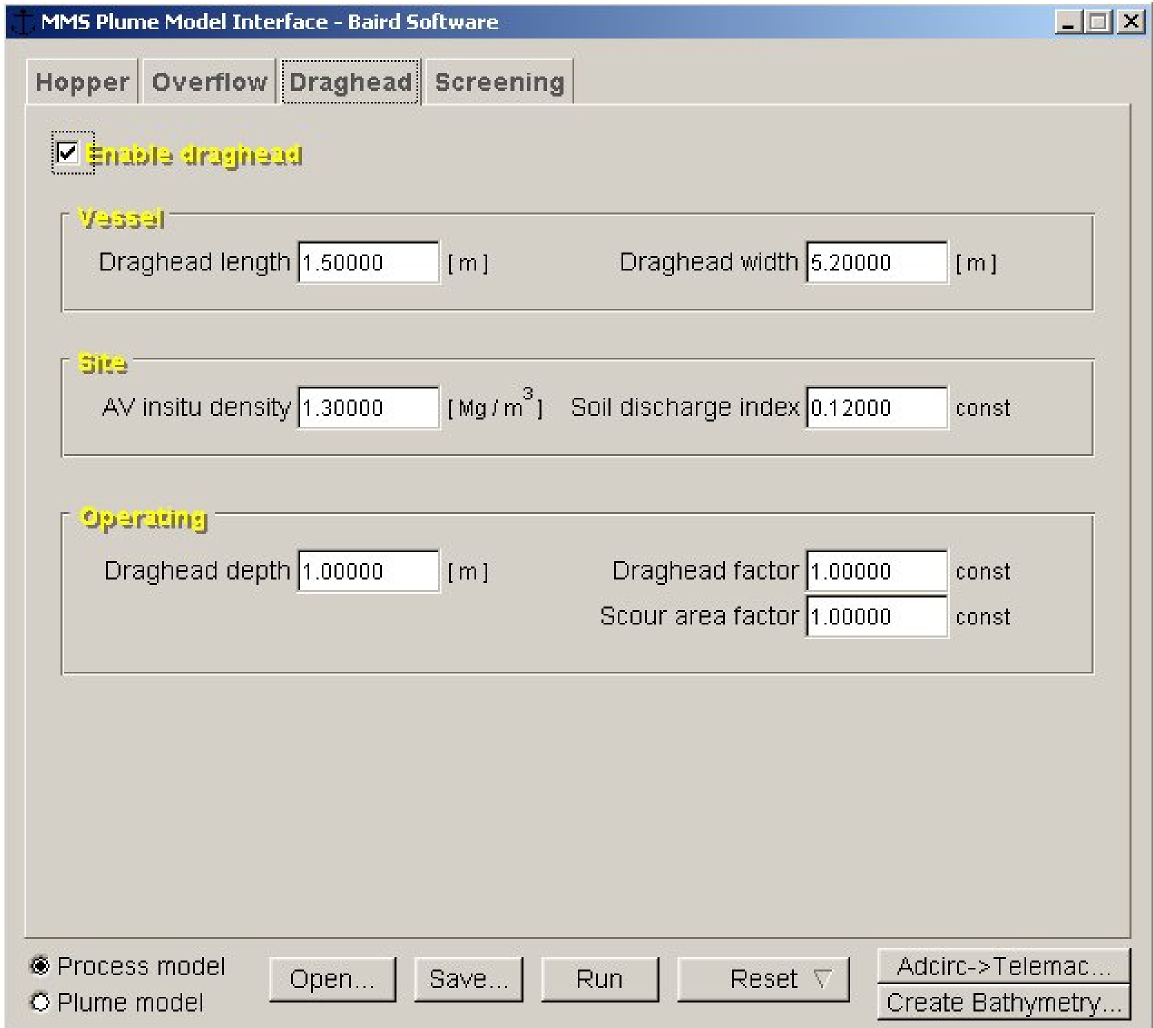
- The *Reset* button resets everything to their default values.

- The *ADCIRC→ Telemac* button converts ADCIRC output into the Telemac file format suitable for input to the Plume Model.

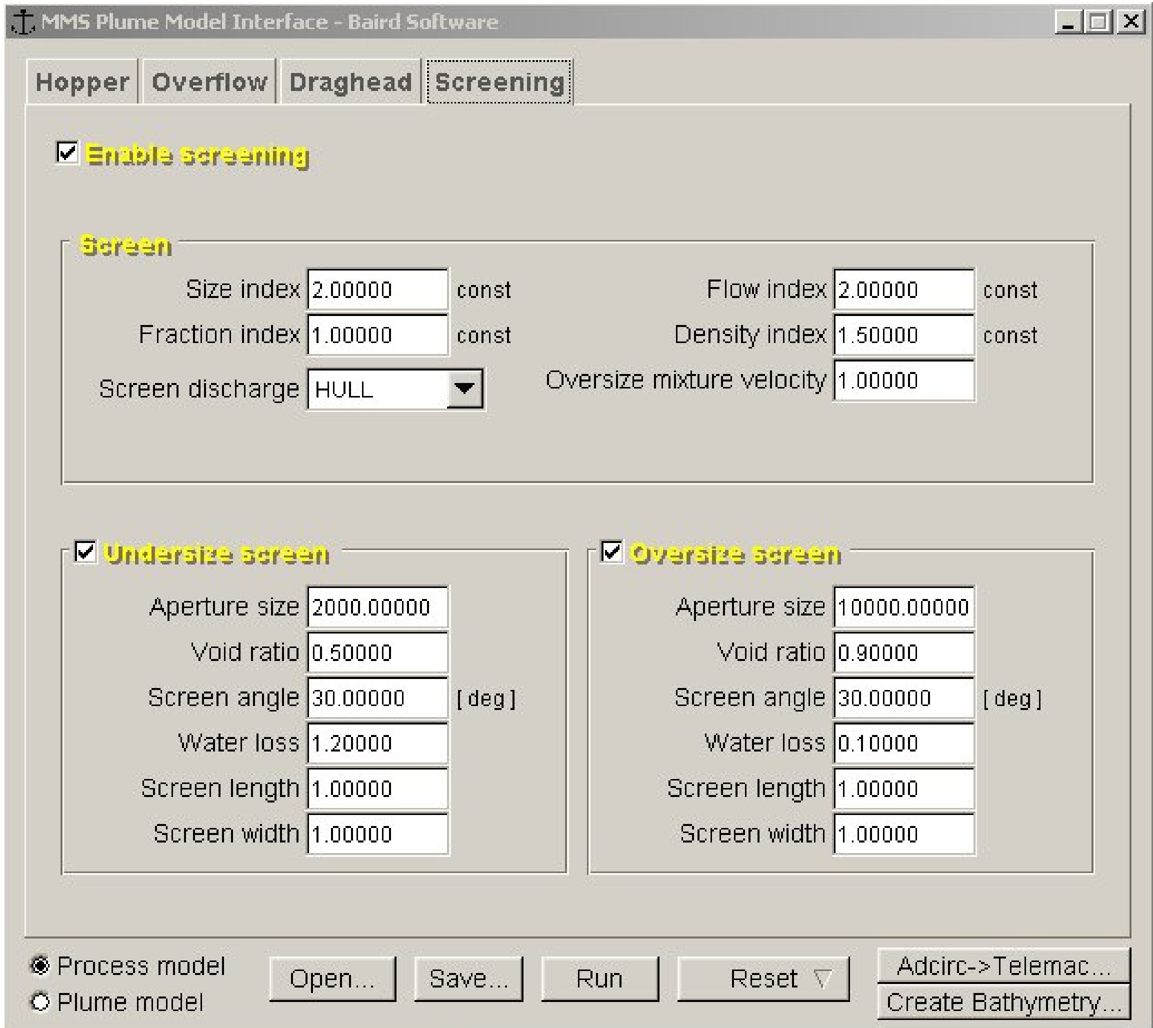
- The Plume model does not retain real world coordinates in the computational process. The *Create Bathymetry* button writes out a special file (*.idi) that permits the Baird Animator software to display the plume model results in the original real world coordinate system.



In this menu, specifics related to the hopper dimensions, the settled density within the hopper and the in-situ grain size curve are entered. The user can select either a *constant tonnage* system or a *constant volume* system.

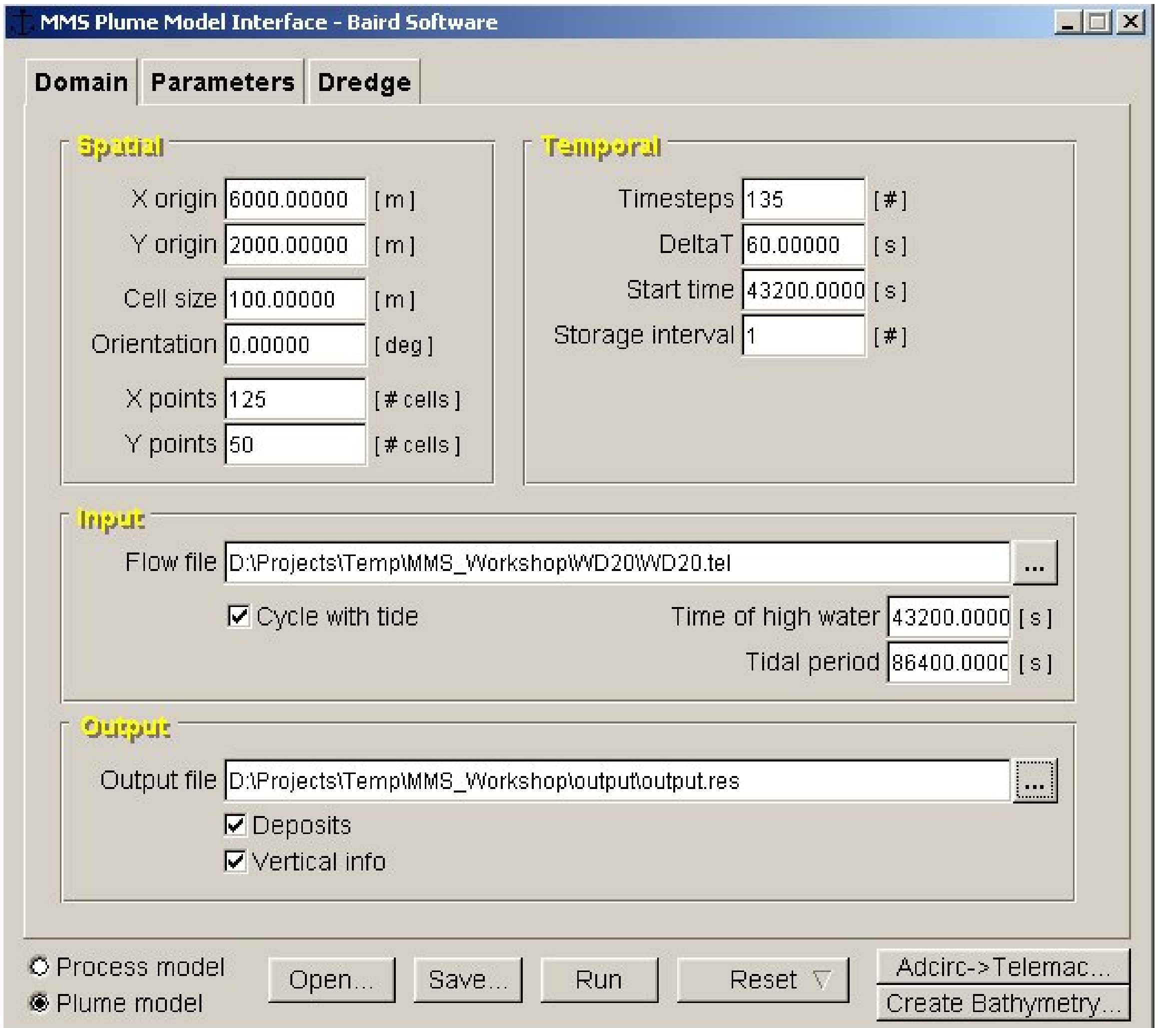


Draghead effects can be enabled by means of the checkbox. If it is enabled, the dimensions of the draghead and various other draghead properties must be entered.

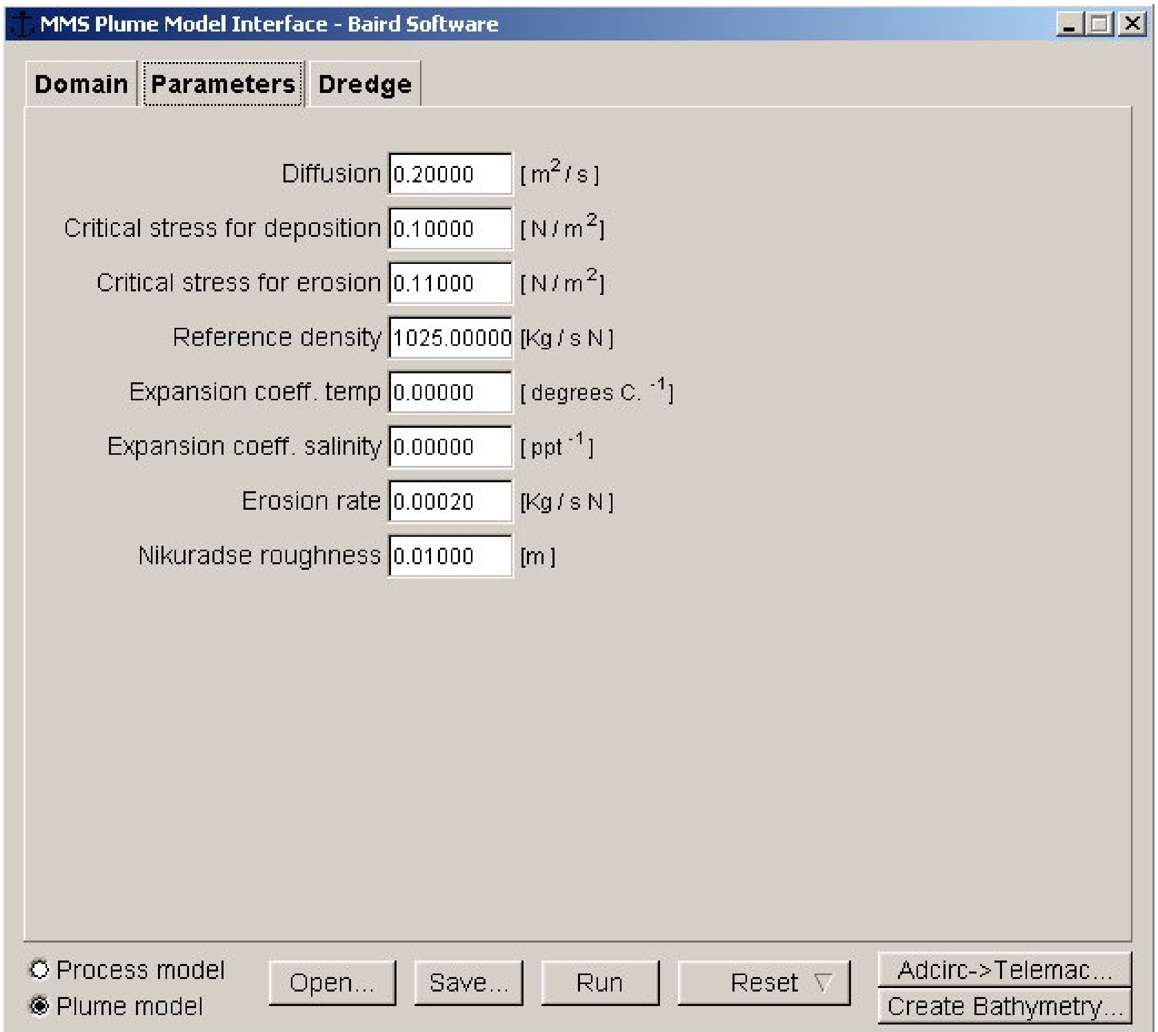


Undersize and oversize screening of the aggregate can be enabled by means of the various checkboxes. Screen dimensions and other screening-related properties are entered in this menu.

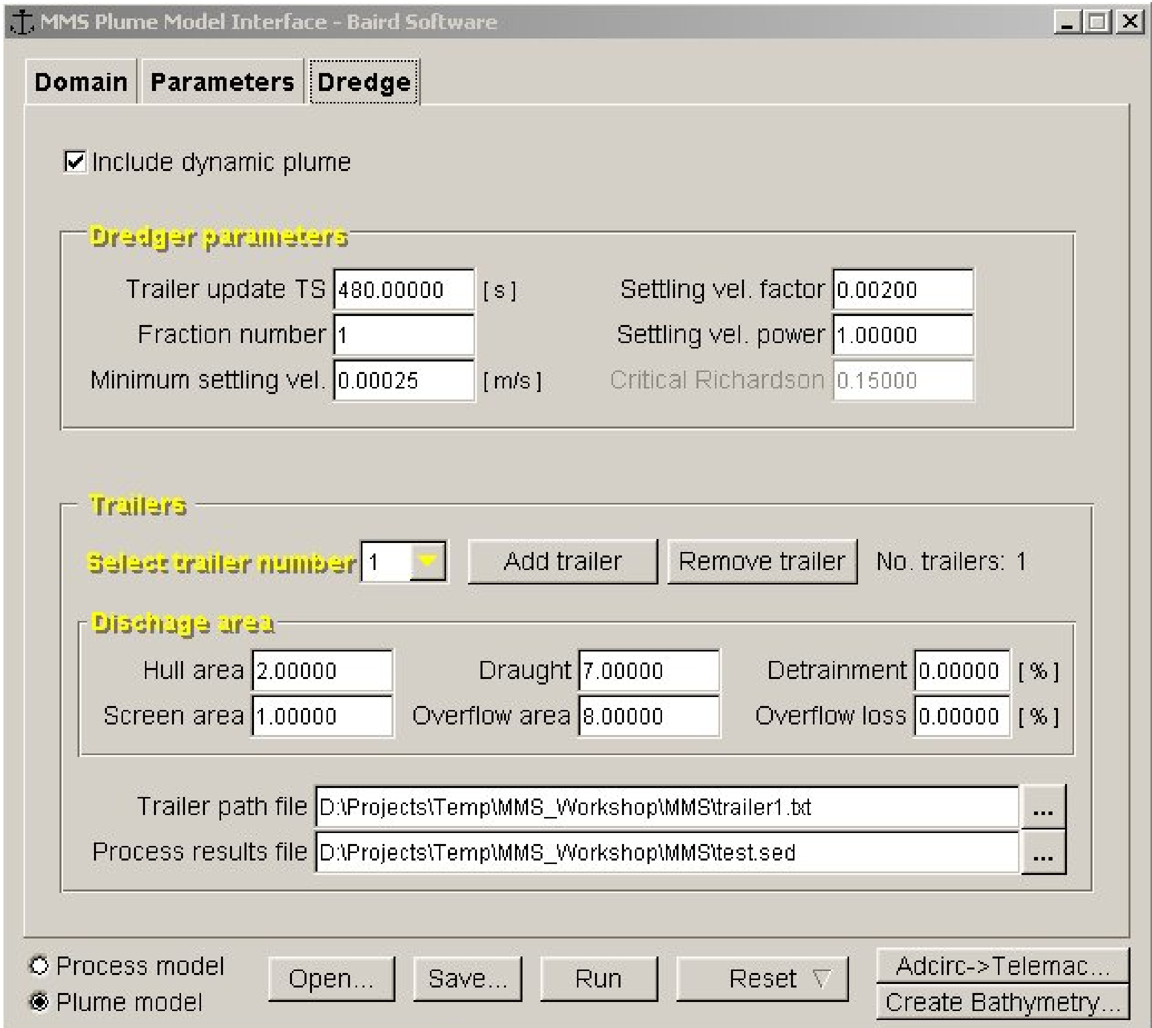
8.3 The Plume Model



The domain tab allows the user to define the spatial extent of the plume model grid, the simulation time as well as the hydrodynamic input and plume model output. The hydrodynamic model input files consist of a flow file containing a 3D current field (*.tel) and a corresponding boundary file (*.bnd) which does not need to be specified but is accessed by the model in the background. These two files are created when the Adcirc->Telemac utility is launched. Typically the flow and boundary file have the same name. The plume model results are written (in binary format) to a user defined output file with the extension *.res (for example output.res).



Various values related to the soil erodibility properties and reference density are entered on this page.



On this menu, the number and characteristics of each dredger (up to 10 in total) are entered including the grain (Fraction Number) size to be simulated, the dredger path, and overspill data such as discharge and mass rates, which are determined from the process model.

The following plume model steering file was written out:

```
/---------------FILES----------------------------
FLOW RESULTS FILE = flow.tel
OUTPUT FILE = result.res
BOUNDARY FILE = bound.bnd
LOG FILE = result.log
RELEASE FILE = C:\PROGRA~1\MMSPLU~1\outfall.const

/-----------PROGRAM PARAMETERS-------------------
TIMESTEP = 50.00000
NUMBER OF TIMESTEPS = 100
STORAGE INTERVAL = 10
START TIME = 0.00000
DIFFUSION = 0.20000
CRITICAL STRESS FOR DEPOSITION = 0.10000
CRITICAL STRESS FOR EROSION = 0.11000
EROSION RATE = 0.02000
ROUGHNESS LENGTH = 0.01000
MINIMUM SETTLING VELOCITY = 0.00025
SETTLING VELOCITY FACTOR = 0.00500
SETTLING VELOCITY POWER = 1.00000

/Keyword below gives expansion coefficients for 1st & 2nd stored active
tracers
/Set values to zero when T and/or S not stored
/Values are not used if density values are stored
EXPANSION COEFFICIENTS = 0.00000 ; 0.00000
REFERENCE DENSITY = 1025.00000


/---------------SQUARE GRID PARAMETERS-----------
SQUARE GRID = yes
SQUARE GRID ORIGIN X = 0.00000
SQUARE GRID ORIGIN Y = 0.00000
SQUARE GRID X POINTS = 100
SQUARE GRID Y POINTS = 100
SQUARE GRID SIZE = 100.00000
SQUARE GRID ORIENTATION = 0.00000

/-------------POINT SOURCE DATA------------------
NUMBER OF POINT SOURCES = 0

/------------------------OUTPUT------------------
OUTPUT DEPOSITS = no
OUTPUT STRESS = no
VERTICAL INFO = no
DISSOLVED = no

/-----------DYNAMIC PLUME PARAMETERS--------------
CONSTANT TRAILER RELEASE = no
NUMBER OF TRAILERS = 1
TRAILER FRACTION = 1
TRAILER SCREEN AREA = 3.00000
TRAILER OVERFLOW AREA = 1.00000
TRAILER HULL AREA = 5.00000
TRAILER DRAUGHT = 6.00000
TRAILER SPEED = 0.1
```

```
INCLUDE DYNAMIC PLUME = yes
TRAILER UPDATE TIMESTEP = 900.00000
TRAILER PROCESS RESULTS FILE 1 = C:\temp\Run12_Oversize.in
TRAILER PATH FILE 1 = C:\temp\trailerpath.txt

LOG ADDITIONAL PLUME INFORMATION = yes
DETRAINMENT PER CENT = 0
END
```

9.0 MODEL OUTPUT VISUALIZATION: PLUME ANIMATOR

The MMS Plume Model generates an extensive amount of data at output. In order to aid in the use and interpretation of the model, a specific data visualization tool was developed. Built upon the framework of an existing Unix visualization tool developed by Baird Software, the Plume Animator provides a comprehensive three-dimensional visualization environment for exploring the Plume Model output. It contains a variety of features including:

- The ability to generate time-varying 3D views of the sediment plumes and seabed bathymetry.
- Show time-varying sediment accumulations by means of color-contoured maps draped over the existing seabed bathymetry.
- Vertical and horizontal slicing through the plume concentration data.
- Display of the ADCIRC hydrodynamic data (water levels, current vectors, etc.).
- Generate animated flights through the datasets.
- Fully interactive graphical exploration of the data by means of mouse control.
- Ability to view two or more datasets simultaneously in different viewports for model run comparisons.
- A wide range of contour colouring options and display types.

Use of the Plume Animator software is described in a separate document, the Plume Animator Users Guide (Baird, 2003).

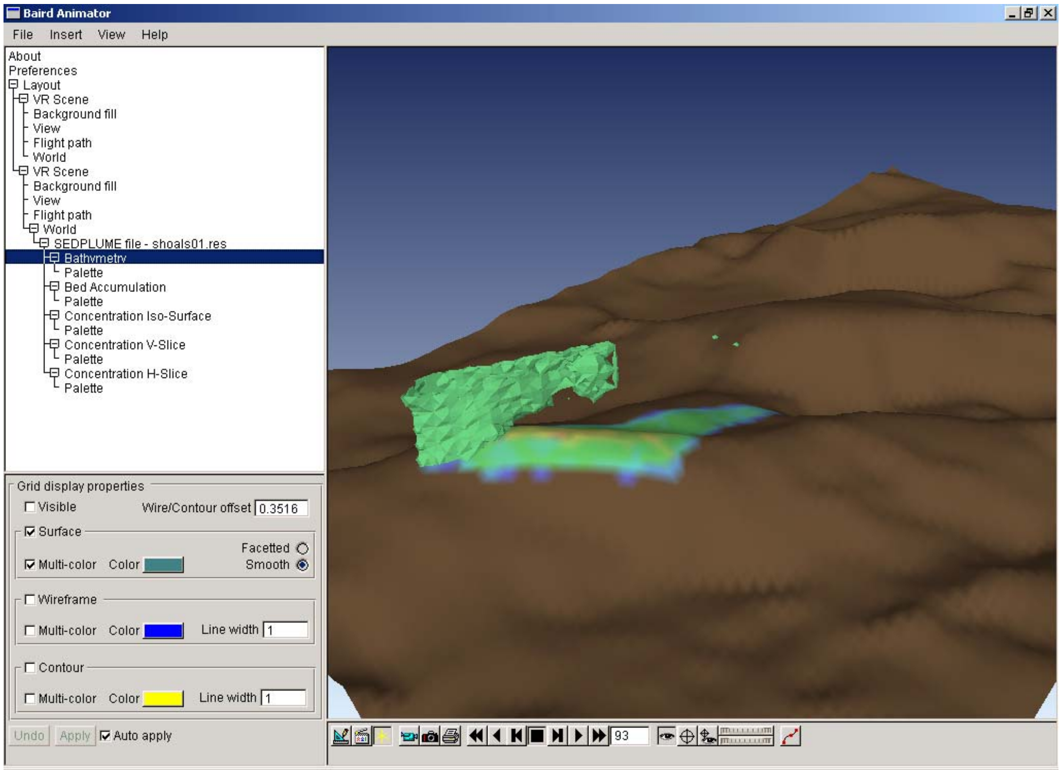


FIGURE 3. The Plume Animator Interface

10.0 REFERENCES

Baird & Associates. (2003). Baird Plume Animator User's Guide. Prepared for the U.S. Minerals Management Service.

Baird & Associates. (2004). Development of the MMS Dredge Plume Model. Prepared for the U.S. Minerals Management Service. March.

Leuttich, R.A. and J.J. Westerink, 2000. ADCIRC: An ADvanced CIRCulation Model for Oceanic, Coastal and Estuarine Waters. Online Document: http://www.marine.unc.edu/C_CATS/adcirc/adcirc.htm.

APPENDIX I
SUMMARY OF DREDGER PROCESS MODEL KEYWORDS

Existing keywords

//compulsory//

TITLE = Test screen model run

DREDGER RESULTS FILE = .\msssed.drg

/

/Vessel

/

HOPPER CAPACITY = 11750.0 m^3

HOPPER TONNAGE = 16774.0 tonnes

SUCTION PIPE DIAMETER = 0.6 m

DRAFT LIGHT AFT = 4.45 m

DRAFT LIGHT FORE = 3.45 m

DRAFT LOADED MAX = 9.4 m

/

/Site

/

DURATION TRAIL RUN = 16.0 mins

TIME INCREMENT = 1.0 mins

PARTICLE DENSITY = 2.65 Mg/m^3

WATER DENSITY = 1.025 Mg/m^3

DREDGING DEPTH = 11.00 m

/

/Operating

/

TRAILING SPEED = 2.0 Knots

SUCTION PIPE NUMBER = 1 Number of suction pipes

MIXTURE DENSITY = 1.2 Mg/m^3

SUCTION PIPE VELOCITY = 5.45 m/s

TURNING TIME = 5.0 min

DISPOSAL TIME = 120.0 min

/

/////////////////////////////Additional compulsory keywords for Overflow////////////////////////////////////

/

OVERFLOW ON = 1 allows hopper to overflow

/

/Vessel

/

HOPPER LENGTH = 57.17 m

HOPPER WIDTH = 18.0 m

HOPPER DEPTH = 11.42 m

CONSTANT TONNAGE = 1 **either constant tonnage or**

CONSTANT VOLUME = 1 **constant volume**

/

/Site

/

WATER KINEMATIC VISCOSITY = 1.36E-06 m^2/s

SOIL RATIO LOST = 1.0 All small particle size are lost when equal to 1.0

/

/Soils

/	
SOILS = 16	Number of particle sizes
10.0 2.0	For each particle:
40.0 3.0	size(mu) cum %distribution
70.0 4.0	
90.0 5.0	
125.0 6.0	
175.0 8.0	
250.0 20.0	
350.0 30.0	
500.0 42.0	
800.0 48.0	
1500.0 54.0	
3000.0 60.0	
5000.0 66.0	
8000.0 74.0	
15000.0 92.0	
25000.0 100.0	
/	
/Operating	
PRELOAD VOLUME = 1.0	m^3
SETTLED DENSITY = 1.9	Mg/m^3
/	
///////////////////////////////Additional compulsory keywords for Draghead///////////////////////////////	
/	
DRAGHEAD ON = 1	allows Draghead loss to be calculated
/	
/Vessel	
/	
DRAGHEAD LENGTH = 1.5	m
DRAGHEAD WIDTH = 5.2	m
/	
/Site	
/	
AV INSITU DENSITY = 1.30	Mg/m^3
SOIL DISAGGREGATION INDEX = 0.12	constant
/	
/Operating	
/	
DRAGHEAD DEPTH = 1.0	m
DRAGHEAD FACTOR = 1.0	constant
SCOUR AREA FACTOR = 1.0	constant
/	

////////////////////////////Additional keywords for screening///

SCREENING RESULTS FILE =	\sed.scn output file for screening results
SEDPLUME FILE = .\Sedplm.in	output file for input to sedplume
SCREEN DISCHARGE OUTPUT =	HULL include keyword if screen discharge to Hull
OVERFLOW OUTPUT = HULL	include keyword if overflow to Hull
SCREEN SIZE INDEX = 2.0	constant - set to this value at present
SCREEN FRACTION INDEX = 1.0	constant - set to this value at present
SCREEN FLOW INDEX = 2.0	constant - set to this value at present
SCREEN DENSITY INDEX = 1.5	constant - set to this value at present

Include following if oversize screening required:

OVERSIZE SCREEN ON = 1	flag to indicate oversize required
OVERSIZE APERTURE SIZE =	10000.0 particle size equal to or above rejected
OVERSIZE VOID RATIO = 0.9	ratio of void to total area
OVERSIZE SCREEN ANGLE = 30.0	angle from horizontal in degrees
OVERSIZE WATER LOSS = 0.1	

Include following if undersize screening required:

UNDERSIZE SCREEN ON = 1	flag to indicate undersize required
UNDERSIZE APERTURE SIZE = 2000.0	particle size equal or smaller rejected
UNDERSIZE VOID RATIO = 0.5	ratio of void to total area
UNDERSIZE SCREEN ANGLE = 30.0	angle from horizontal in degrees
UNDERSIZE WATER LOSS = 1.2	

Include following if both screening reject and overflow to Hull:

SCREEN DISCHARGE OUTPUT =	HULL screen discharge to Hull
OVERFLOW OUTPUT =	HULL overflow to Hull

Note: **Formatted read of steering file. REALS are F10.3, Integers are I10. In general this causes no problems, if comments are not put on same line as data. Make sure list of soil sizes and percentages have correct number of spaces between.**

<u>Output File for SedPlume</u>

NumberParticleSizes (I3)	Number of particle sizes in distribution
Particle Size(1X,F10.3)	repeated for each particle size (mu)

IF (Compute Draghead) then:

Weight Drag Loss per particle size(1X,F10.3)	loss of sediment due to draghead repeated for each particle size (kg/s)

IF (Screening activated) and both not through HULL then:

Screening reject flow(1X,F10.3)	total flow of rejected material (m^3/s)
Total rejected material per particle size (1X,F10.3)	for each particle size total material rejected through screening process (kg/s)

In all cases hopper overflow:

Time Flow Particle(1) …. Particle (n) overflow (20*(1X,F10.3))*	for each time step until end of run: time (mins), flow (m^3/s), followed by loss per particle size (kg/s) for time step. The flow and loss can either be overflow of hopper or if both through Hull then total of screening reject + hopper overflow.

APPENDIX II
SUMMARY OF PLUME MODEL KEYWORDS

The keywords have been classified under seven headings: Files, Grid parameters, Physical parameters, Run parameters, Point source releases, Trailer Suction Hopper Dredger parameters, and deactivated parameters.

Within each of these groupings the keywords are in alphabetical order.

1 Files

Keyword	**BOUNDARY FILE**
Type	character
Default value	conlim
Comments	Maximum length is 80 characters. This file is used by the TELEMAC-3D flow model.

Keyword	**FLOW RESULTS FILE**
Type	character
Default value	resu.in
Comments	Maximum length is 80 characters. This file is produced by TELEMAC-3D, and must be in SERAPHIN format.

Keyword	**LOG FILE**
Type	character
Default value	rw.log
Comments	Maximum length is 80 characters. Contains information on time and date of run, values of keywords used, progress reports on the run and any error messages.

Keyword	**OUTFALL FILE**
Type	character
Default value	"none" (outfall file not used)
Comments	Maximum length is 80 characters. Used for specifying time-varying releases. See Section 3.3.3. When the OUTFALL FILE keyword is used, the DISCHARGE RATE, NUMBER OF RELEASE PERIODS, PARTICLES PER HOUR, RELEASE EVERY TIDE, START RELEASE and STOP RELEASE keywords should not be present in the steering file. **This file must be used when trailers are being simulated.**

Keyword	**OUTPUT FILE**
Type	character
Default value	resu.out

Comments	Maximum length is 80 characters. This file contains the model results in RUBENS LEONARD format.

2 Grid parameters

The SEDPLUME output grid should be chosen to cover the area of interest, as model results can only be plotted for the area of the output grid. It should be noted that it does not affect the results if particles “fall off” the edge of the square grid, provided they are still within the area of the flow model. They will be tracked as usual, and can return to the square grid area at a later time. If, however, a particle leaves the flow model, through an open boundary, it is lost permanently. The maximum number of cells in the square grid is 90,000. If the maximum number of cells in the square grid exceeds 90,000, an error message will be generated (refer Chapter 6).

Keyword	**SQUARE GRID**
Type	logical
Default value	yes
Comments	This parameter is redundant.

Keyword	**SQUARE GRID ORIENTATION**
Type	real
Default value	0.0
Comments	This defines the angle (in degrees) between the x-axis of the flow model and the x-axis of the output square grid, such that the angle increases as the square grid is rotated anti-clockwise.

Keyword	**SQUARE GRID ORIGIN X**
Type	real
Default value	0.0
Comments	x-coordinate of the bottom left corner of the square grid. This origin need not necessarily lie within the TELEMAC-3D model area, although some part of the square grid must intersect with the flow model area. Output grids should be chosen to cover the full area of interest. Although model particles are tracked outside the output grid, SEDPLUME results cannot be plotted for such areas.

Keyword	**SQUARE GRID ORIGIN Y**
Type	real
Default value	0.0
Comments	y-coordinate of the square grid origin.

Keyword	**SQUARE GRID SIZE**
Type	real

Default value	0.0
Comments	The size in metres of each cell in the square grid. This cell dimension should be sufficiently small to resolve the main features of simulated plumes.

Keyword	**SQUARE GRID X POINTS**
Type	integer
Default value	10
Comments	Number of cells along the x-axis of the grid. The value of this parameter multipled by the value of SQUARE GRID Y POINTS should be less than or equal to 300,000 **[NB to be confirmed].**

Keyword	**SQUARE GRID Y POINTS**
Type	integer
Default value	10
Comments	Number of cells along the y-axis of the grid.

3 Physical parameters

Keyword	**CRITICAL STRESS FOR DEPOSITION**
Type	real
Default value	0.0
Comments	When suspended model particles impinge on the sea bed, deposition occurs if the bed shear stress exerted by the tidal currents is less than the value specified here ($N.m^{-2}$). A typical value is $0.1N.m^{-2}$. This keyword is ignored if simulated substances are dissolved (see DISSOLVED).

Keyword	**CRITICAL STRESS FOR EROSION**
Type	real
Default value	0.0
Comments	Model particles deposited on the sea bed are eroded whenever the bed shear stress exerted by the tidal currents exceeds the value specified here ($N.m^{-2}$). This keyword is ignored if simulated substances are dissolved (see DISSOLVED).

Keyword	**DIFFUSION**
Type	real
Default value	0.2
Comments	This is the value of the horizontal eddy diffusivity in units of $m^2.s^{-1}$.

Keyword	**DISSOLVED**
Type	logical
Default value	yes

Comments

If the value of this keyword is “yes”, tracer substances are treated as dissolved; that is, settling, deposition, etc are not included. If the value is “no”, then substances are treated as suspended sediment, and settling velocities and bed stress parameters (see SETTLING VELOCITY FACTOR, SETTLING VELOCITY POWER, MINIMUM SETTLING VELOCITY, CRITICAL STRESS FOR DEPOSITION, CRITICAL STRESS FOR EROSION and EROSION RATE) must be specified. In this latter case, sediment deposit distributions will be included in the output file.

Note that this keyword should not be used in conjunction with the trailer releases.

Keyword	**EROSION RATE**
Type	real
Default value	0.0

Comments

This is the rate constant used to calculate the number of model particles re-suspended from the sea bed during each time-step of a model simulation. Units are $kg.s^{-1}.N^{-1}$. Use of a very large value here will result in re-suspension of particles being essentially instantaneous when the bed shear stress exceeds the erosional shear strength of the deposit. This keyword is ignored if dissolved substances are being simulated (see DISSOLVED).

Keyword	**EXPANSION COEFFICIENTS**
Type	two real numbers
Default value	both 0.0

Comments

This keyword is used to specify the expansion coefficients to be used to calculate water densities from the first and second stored active tracers in a TELEMAC-3D results file. The keyword can be omitted if densities are actually stored in the results file. If temperature and/or salinity are stored in the results file, then the model derives densities using $\rho/\rho_o=\alpha.T_1+\beta.T_2$, where T_n are active tracers, in the order stored. Typical values of the expansion coefficients are -0.0003°C^{-1} and 0.0008ppt^{-1} (ppt = parts-per-thousand) for temperature and salinity respectively.

Keyword	**MINIMUM SETTLING VELOCITY**
Type	real
Default value	**0.00025**

Comments

This is the minimum settling velocity (w_{min}, $m.s^{-1}$) to be applied and must be positive. It can be used to set a constant, concentration-independent settling velocity by settling the value under SETTLING VELOCITY

FACTOR to 0.0. This keyword is ignored if dissolved substances are being simulated (see DISSOLVED).

Keyword	**REFERENCE DENSITY**
Type	real
Default value	**1025.**
Comments	Gives the value of the reference density of ambient water. This may represent the seawater density in homogeneous situations or a "reference density" the variation from which is derived from the computed flow model densities or the temperature/salt tracer concentrations.

Keyword	**ROUGHNESS LENGTH**
Type	real
Default value	0.01
Comments	Gives the value in metres of the Nikuradse roughness length for the type of bed material typical of the area being modelled. The value specified here is typically the same as that used in the flow model.

Keyword	**SETTLING VELOCITY FACTOR**
Type	real
Default value	**0.005**
Comments	This parameter allows particles to be assigned a negative buoyancy. It is a proportionality constant by which suspended sediment concentrations (kg.m^{-3}) are multiplied to derive settling velocities in m.s^{-1} (P in Equation (3)). The value specified here must be positive. This keyword is ignored if dissolved pollutants are being simulated (see DISSOLVED).

As an alternative to the use of a functional settling velocity, a constant value can be specified – see the MINIMUM SETTLING VELOCITY keyword.

Keyword	**SETTLING VELOCITY POWER**
Type	real
Default value	1.0
Comments	This is the power to which pollutant concentrations (kg.m^{-3}) are raised before being used to derive a settling velocity. The value specified here must be positive. This keyword is ignored if dissolved substances are being simulated (see DISSOLVED).

Keyword	**TIDAL PERIOD**
Type	real
Default value	44400.0
Comments	Length of the flow model results file (s). The value specified here is used by SEDPLUME to determine whether the input flow model results

will be used more than once; when the SEDPLUME simulation reaches a time greater than the flow model start time plus the tidal period, flows are re-read from the start of the flow model results file. In the case of flow model runs consisting of multiple tides, or where tidal effects are negligible, the value specified here should be the duration of the flow files.

Keyword	**TIME OF HIGH WATER**
Type	real
Default value	0.0
Comments	Gives time of High Water in terms of flow data timing (units: seconds). The value of this parameter is used for time varying discharges. Where the flow model results file covers more than one tidal cycle, or where there are no significant tidal effects, the time specified here should be that of a 'reference high water', relative to which the release time series data is defined.

Keyword	**TIME OF DAY OF HIGH WATER**
Type	character string in HH:MM format
Default value	not set
Comments	This keyword is redundant.

4 Run parameters

Keyword	**OUTPUT LAYERS**
Type	real array of size N-1, where N is the number of layers in the flow model.
Default value	output all flow model layers
Comments	If any of the layers is set then calculated sediment concentrations are written to the output file for the defined layer (Layer 1 is nearest the sea bed, and Layer N-1 is nearest the sea surface). Any combination of layers can be chosen. Note that the SEDPLUME results always consist of one layer less than the associated TELEMAC-3D results

Keyword	**OUTPUT DEPOSITS**
Type	logical
Default value	no
Comments	If this keyword is set to "yes", calculated deposits of particulate substances are written to the output file at the same intervals as the concentration results. This keyword can only be used for suspended substances (see DISSOLVED).

Keyword	**OUTPUT STRESS**
Type	logical

Default value	no
Comments	If this keyword is set to “yes”, calculated bed shear stresses are written to the output file at the same intervals as the concentration and deposits results.

Keyword	**MINIMUM DEPTH**
Type	real
Default value	0.3
Comments	Used to specify which areas the program regards as effectively dry. Areas are treated as dry wherever the water depth is less than the value specified here. Units are metres.

Keyword	**NUMBER OF TIMESTEPS**
Type	integer
Default value	100
Comments	This parameter specifies the length of the run. If the number of time-steps multiplied by the duration of each time-step is greater than the time covered by the flow results, then the flow results are reused in a periodic way.

Keyword	**OUTPUT TIDE AVERAGE SSC**
Type	logical
Default value	no
Comments	This keyword is redundant.

Keyword	**START TIME**
Type	real
Default value	0.0
Comments	Specifies the point in the flow results file where the SEDPLUME run begins on the reference timescale of the flow model. For example, if a run is to begin at the start of a set of flow model results, in which the first time is 129600s, then START TIME should be set to 129600.0. The units are seconds.

Keyword	**STORAGE INTERVAL**
Type	integer
Default value	10
Comments	Gives the number of time-steps between writing to the results file.

Keyword	**TIDE AVERAGE SSC FILE**
Type	string
Default value	avssc.out

Comments This keyword is redundant.

Keyword **TIMESTEP**
Type real
Default value 50.0

Comments Units are seconds.

Keyword **VERTICAL INFO**
Type logical
Default value no

Comments If set to "yes", this prints information on the vertical distribution of particles to the log file while the program is running. The water depth is split into 10 equal parts, and the number of particles in each tenth is summed over the model area, giving an indication of whether particles are evenly spread through the depth or predominantly at the surface, etc.

5 Point Source Releases

Keyword **DECAY**
Type real
Default value -99.9 (conservative substance)

Comments This keyword is redundant as water quality aspects of programme have been deactivated.

Keyword **DISCHARGE RATE**
Type real array of size 10
Default value all 100.0

Comments SEDPLUME can simulate discharges for up to 10 sediment sources in a single run. Here, the sediment discharge (the product of the source discharge and the source concentration) is specified for each release point. The sources must be specified in the same order as for other keywords used for multiple sources.

Keyword **INITIAL SPREADING RADIUS**
Type real array of size 10
Default value all 0.0

Comments The radii of circles (m), centred on the sediment release point coordinates, in which particles can be released at random. Model results are **(over the medium/long term)** generally insensitive to values specified here, and the keyword is often omitted.

Keyword **RELEASE POINT DEPTH RANGE**
Type real array of size 10

Default value	all 0.0
Comments	Optional depth range (m), downward from the sea surface over which sediment release at each sediment source can be evenly spread.

Keyword	**NIGHT TIME DECAY**
Type	real
Default value	-99.9
Comments	This keyword is redundant as water quality aspects of programme have been deactivated.

Keyword	**NUMBER OF OUTFALLS**
Type	integer
Default value	1
Comments	The value of this parameter specifies the number of **stationary** sediment sources to be simulated. The maximum permitted value of this parameter is **(10-N_{tr}) where N_{tr} is the number of trailers being simulated.**

Keyword	**NUMBER OF RELEASE PERIODS**
Type	integer
Default value	0
Comments	This keyword can be used for non-intermittent releases, or can be used together with START RELEASE, STOP RELEASE and RELEASE EVERY TIDE to model intermittent releases. It specifies the number of distinct release periods to be used, and should be set to zero, or omitted for releases which are constant through the simulation or which vary as prescribed in an file. The maximum permitted value for this keyword is 10. If the simulation uses the flow model results more than once (ie, for a repeating tide), and the keyword RELEASE EVERY TIDE is set to 'yes', then the specified discharge periods will be repeated each time the flows are used. In this instance, the START RELEASE and STOP RELEASE keywords relate to the first tide, only. The name of the keyword arises because flow model results are often stored for a repeating tidal cycle; however, it is not strictly appropriate for longer sets of results, such as spring-neap cycles.

Keyword	**OUTFALL LOCATION X**
Type	real array of size 10
Default value	all 0.0
Comments	Specifies the x-coordinate (in metres) of the location of up to 10 sediment sources in terms of the coordinate system on which the model is based.

Keyword	**OUTFALL LOCATION Y**
Type	real array of size 10
Default value	all 0.0
Comments	Specifies the y-coordinate (in metres) of the location of up to 10 sediment sources in terms of the coordinate system on which the model is based. The coordinates must be specified in the same release point order as OUTFALL LOCATION X.

Keyword	**T90 CUTOFF DEPTH**
Type	real
Default value	-1000.0
Comments	This keyword is redundant as water quality aspects of programme have been deactivated.

Keyword	**PARTICLES PER HOUR**
Type	integer array of size 10
Default value	all 100
Comments	This keyword specifies the number of particles released each hour at each sediment release point - the number of particles released each time-step at each source is calculated from these numbers and the value of TIMESTEP. The rates must be specified in the same order as under OUTFALL LOCATION X, etc. Release rates of 1000 particles/hour are adequate for most modelling purposes. The maximum number of particles which can be used in a SEDPLUME run is **1,000,000**. SEDPLUME carries out a check upon execution to determine if the maximum number of particles will exceed **1,000,000**. If this is the case, the user will be informed.

Keyword	**RELEASE EVERY TIDE**
Type	logical
Default value	no
Comments	See also NUMBER OF RELEASE PERIODS. If this keyword is set to ‘yes’, the release periods will be reused each time the flow file is repeated. Consequently, a release of, for example, 1 hour following each high water can be specified as a single repeating release in a simulation lasting for several cycles of a single-tide flow model results file.

Keyword	**START RELEASE**
Type	real array of size 10
Default value	all 0.0
Comments	See also NUMBER OF RELEASE PERIODS. The value(s) assigned to START RELEASE give the times in hours after the start of the run when each release period begins.

Keyword	**STOP RELEASE**
Type	real array of size 10
Default value	all 0.0
Comments	See also NUMBER OF RELEASE PERIODS. The value(s) assigned to STOP RELEASE give the times in hours after the start of the run when each release period ends. Note that the release periods should not overlap.

6 Trailer Suction Hopper Dredger parameters

Keyword	**CONSTANT TRAILER RELEASE**
Type	logical
Default value	no
Comments	If this flag is set neither the dynamic plume routine or the process file are used. The release is given by the outfall file and is distributed uniformly through the water column depth and in a gaussian distribution with (standard deviation 10m) about the dredger centre line.

Keyword	**INCLUDE DYNAMIC PLUME**
Type	logical
Default value	yes
Comments	If this flag is set to “no” the dynamic plume routine is not used and the sediment release given by the process file is distributed uniformly through the water column depth and in a gaussian distribution with (standard deviation 10m) about the dredger centre line.

Keyword	**NUMBER OF TRAILERS**
Type	integer
Default value	0
Comments	Specifies the number of trailer suction hopper dredgers to be simulated. The number specified here cannot exceed that specified under the NUMBER OF OUTFALLS keyword.

Keyword	**NUMBER OF TRAILER SPILLWAYS**
Type	integer
Default value	6
Comments	Specifies the number of ship-side spillways on each dredger. It is used to work out the total release area in the case of ship-side overflow.

Keyword **TRAILER DRAUGHT**
Type real array of size 10
Default value All 0.0

Comments Specifies the draught of each dredger. Only used in conjunction with discharge through the hull and resets the point of release to the bottom of the dredger hull.

Keyword **TRAILER FRACTION**
Type integer
Default value 1

Comments Specifies which sediment fraction is of interest in the dispersion simulation.

Keyword **TRAILER HULL AREA**
Type real array of size 10
Default value All 1.0

Comments Specifies the area of the hull mounted overflow and/or screening discharge.

Keyword **TRAILER OVERFLOW AREA**
Type real
Default value All (number of sills x $1.0m^2$)

Comments Specifies the area of the ship-side overflow discharge – more specifically the area of the interface of an individual descending weir flow with the water surface.

Keyword **TRAILER PATH FILE i (i = 1,2, …number of trailers)**
Type character array of size 10
Default value none

Comments Specifies the name(s) of the file(s) containing the path(s) of each of the trailers being simulated.

Keyword **TRAILER PROCESS RESULTS FILE**
Type character array of size 10
Default value none

Comments Specifies the name(s) of the dredger process model results file that will be used to define the sediment releases in the simulation. The maximum length of each file name is 80 characters.

Keyword **TRAILER SCREEN AREA**
Type real array of size 10
Default value All 1.0

Comments	Specifies the area of the screening chute discharge – more specifically the area of the interface of the jet with the water surface.
Keyword	**TRAILER SPEED**
Type	real array of size 10
Default value	All 1.0
Comments	x-coordinate of the most westerly point of the dredger path.
Keyword	**TRAILER TURNING POINT X1**
Type	real array of size 10
Default value	All 0.0
Comments	x-coordinate of the most westerly point of the dredger path.
Keyword	**TRAILER TURNING POINT Y1**
Type	real array of size 10
Default value	All 0.0
Comments	y-coordinate of the most westerly point of the dredger path.
Keyword	**TRAILER TURNING POINT X2**
Type	real array of size 10
Default value	All 0.0
Comments	x-coordinate of the most easterly point of the dredger path.
Keyword	**TRAILER TURNING POINT Y2**
Type	real array of size 10
Default value	All 0.0
Comments	y-coordinate of the most easterly point of the dredger path.
Keyword	**TRAILER UPDATE TIMESTEP**
Type	real
Default value	900.0
Comments	This keyword specifies the time step after which the dynamic plume simulation will be repeated, updating the size and distribution of the releases for the dredger.

7 Dynamic plume parameters

Keyword	**CRITICAL RICHARDSON NUMBER**
Type	real
Default value	0.15
Comments	This governs the point at which sediment in the density current on the sea bed can mix with the ambient overlying water

Keyword	**DETRAINMENT PER CENT**
Type	real array of size 10
Default value	All 0.
Comments	Specifies the amount lost from the dynamic plume through detrainment. This does not include the effect settling from the plume which is calculated separately.

Keyword	**LOG ADDITIONAL PLUME INFORMATION**
Type	logical
Default value	no
Comments	If this flag is set the results of the dynamic plume model are written to the log file every time the dynamic plume routine is called.

Keyword	**MAXIMUM TIME FOR PLUME COLLAPSE**
Type	real
Default value	1800
Comments	Specifies the maximum allowable duration for the dynamic plume phase.

Keyword	**OVERFLOW LOSS PER CENT**
Type	real array of size 10
Default value	All 0.
Comments	Specifies the amount lost from the overflow plume as a surface plume due to aeration.

8 General format of steering file

The following points are important as regards the general format of the steering file:

- Where two or more releases/trailers are to simulated, parameters must be separated by a semi colon, e.g. `TRAILER TURNING POINT X1 = 827697.0 ; 826697.0`

- Keywords can be entered in any order

- At the end of the steering file the following line should appear (otherwise the PC program may crash),
  `END`

APPENDIX III
ERROR MESSAGES

A list of error messages which may be produced by MMS PLUME MODEL, and suggested corrective action, are given in this section.

Providing the user follows the instructions in this manual, and all the input is sensible, problems should not arise during the running of the program.

Message:	**FATAL ERROR: boundary condition file does not exist**
Cause:	The specified flow model boundary conditions file is not in the current directory.
Action:	Copy the boundary conditions file into the current directory, or set up a suitable link.

Message:	**FATAL ERROR: no dictionary file in current directory**
Cause:	There is no 'rw.dico' file in the current directory.
Action:	Copy or link this file into the current directory.

Message:	**FATAL ERROR: flow results file does not exist**
Cause:	The specified flow model results file is not in the current directory.
Action:	Copy the results file into the current directory, or set up a suitable link.

Message:	**Source *n* Need more particles per hour**
Cause:	The particle release rate specified for the *n*th source is too low for at least one particle to be released at the source per timestep.
Action:	Increase the particle release rate at the appropriate source.

Message:	**FATAL ERROR: release file does not exist**
Cause:	The specified load file is not in the current directory.
Action:	Copy the load file into the current directory, or set up a suitable link.

Message:	**Warning:** **Data for source *n* starts too late** **Time of first record altered**
Cause:	The time of the first record for at least one of the sources in the specified load file is after the start of the run, due to an error in the load file or incorrect data under the TIME OF HIGH WATER keyword. MMS PLUME MODEL has changed the time of the appropriate record to the start time of the model run.
Action:	If necessary, add another record to the load file, or change the specified time of high water.

Message:	**FATAL ERROR** **Read error while reading *filename***
Cause:	The header of the input results file contains data in an unexpected form.
Action:	Change the specified flow file name, as appropriate.

Message:	**FATAL ERROR** **Premature end of file while reading *filename***
Cause:	The header of the input results file is incomplete.
Action:	Check the results file for errors, and re-run MMS PLUME MODEL as necessary.

Message:	**FATAL ERROR** **Too many nodes in input mesh**
Cause:	The mesh of the input flow file has more than 25,000 nodes.
Action:	Re-generate the flow model result file with a reduced number of nodes.

Message:	**FATAL ERROR** **Too many elements in input mesh**
Cause:	The mesh of the input flow file has more than 50,000 elements.
Action:	Re-generate the TELEMAC-format files with a reduced number of elements.

Message:	**WARNING: START TIME reset to *t***
Cause:	The start time of the MMS PLUME MODEL run is before the time of the first stored data in the flow model results file.
Action:	MMS PLUME MODEL will reset the start time to the start of the flow model file. If this is not required, terminate the MMS PLUME MODEL run and re-run with a revised start time.

Message:	**Warning: period of run is greater than that covered by flow file. Flow will be reused.** **If flow is not repeating, errors may result.**
Cause:	A MMS PLUME MODEL run length longer than the flow model results files has been specified.
Action:	This may be required by the user, but will lead to errors if the flow model has not been run for repeating conditions. If re-use of the flow model results is not required, terminate the MMS PLUME MODEL run and re-run after altering the total number of timesteps.

Message:	**whichelem failed for *x y*** **Location of source *n* incorrect**
Cause:	MMS PLUME MODEL has failed to locate a specified source within an element of the flow model mesh, possibly because the source is outside the wet area of the flow model.
Action:	Re-position the source and re-run MMS PLUME MODEL.
Message:	**FATAL ERROR: lost in release file**
Cause:	MMS PLUME MODEL has been unable to identify the current model time with respect to the timing of the records in the load file.
Action:	Check that the times of the records in the load file are correct, remembering that they are relative to the reference HW, change the file as necessary, and re-run MMS PLUME MODEL.
Message:	**Release period *n* has zero duration**
Cause:	For one of the release periods specified for an intermittent source, the start and stop times have been set equal, so no release can occur.
Action:	Re-set the details of the intermittent release, and re-run MMS PLUME MODEL.
Message:	**FATAL ERROR: Too many particles**
Cause:	The number of particles specified for the MMS PLUME MODEL run exceeds 1,000,000.
Action:	Reduce the number of particles required, possibly by dividing the run into separate runs with fewer sources in each.
Message:	**ERROR: Number of output grid cells (*n*) exceeds allowed maximum of 100,000** **PROGRAM TERMINATED.**
Cause:	The specified MMS PLUME MODEL output grid would contain more than 100,000 square cells.
Action:	Reduce the number of square cells, either by reducing the area of the output grid, or by increasing the output grid cell dimension.
Message:	**FATAL ERROR; the maximum allowed number of particles (1,000,000) would be exceeded in this run.** **PROGRAM TERMINATED**
Cause:	The specified MMS PLUME MODEL run would use more than 1,00,000 model particles.
Action:	Reduce the number of particles, either by decreasing the particle release rates or by shortening the run.

Message:	**FATAL ERROR**
	Flow file *filename* must contain at least u,v and at least two "depth" variables
Cause:	An attempt has been made to run MMS PLUME MODEL using a flow model results file which does not contain results for at least four variables (two velocity components, and at least two of total water depth, water surface elevation and sea bed depth).
Action:	Re-run flow model or conversion routine to generate a results file in the required format.

Message:	**FATAL ERROR**
	Flow file variables must include U
Cause:	An attempt has been made to run MMS PLUME MODEL using a results file which does not contain eastward velocities.
Action:	Re-run flow model or conversion routine to generate a results file in the required format.

Message:	**FATAL ERROR**
	Flow file variables must include V
Cause:	An attempt has been made to run MMS PLUME MODEL using a flow model results file which does not contain northward velocities.
Action:	Re-run flow model or conversion routine to generate a results file in the required format.

Message:	**FATAL ERROR**
	Flow file variables must include at least two "depth" variables
Cause:	An attempt has been made to run MMS PLUME MODEL using a flow model results file which does not contain at least two of total water depth, water surface elevation and sea bed depth.
Action:	Re-run flow model or conversion routine to generate a results file in the required format.

Message:	**FATAL ERROR:**
	Dredger speed cannot be zero'
Cause:	An attempt has been made to run MMS PLUME MODEL using a dredger speed of zero.
Action:	Re-run with a positive dredger speed.

Message:	**FATAL ERROR:**
	Outfall" file is compulsory if any of the sources are trailers
Cause:	An attempt has been made to run MMS PLUME MODEL using trailer and point sources, without using the dredgerfile to set release times/rates.
Action:	Re-run with using the dredgerfile
Message:	**FATAL ERROR:**
	Specified turning points for trailer ',i,' are identical.
Cause:	The specified turning points X1, Y1 and X2,Y2 for the trailer are identical
Action:	Correct turning points and re-run.
Message:	**FATAL ERROR:**
	Number of trailer path files differs from number of trailers.
Cause:	There should be one trailer path file for each dredger
Action:	Correct dredger path files and re-run.
Message:	**FATAL ERROR:**
	Trailer path file does not exist
Cause:	There should be one trailer path file for each dredger
Action:	Write trailer path file and re-run.
Message:	**FATAL ERROR:**
	Number of records in pathfile exceeds 200
Cause:	The number of recorded positions in the path file cannot exceed 200.
Action:	Re-write trailer path file and re-run.
Message:	**FATAL ERROR:**
	Dredger locations not in correct time sequence in file
Cause:	The times specified in the path file are not sequential
Action:	Re-write trailer path file and re-run.
Message:	**FATAL ERROR:**
	Process file does not exist
Cause:	There should be a Process file (which may be the same) for each dredger
Action:	Write trailer path file and ensure it is in the correct directory and re-run.

Message:	**FATAL ERROR:** **path data in pathfile(i) starts too late**
Cause:	There is no dredger location data for the start of the dredger release
Action:	Correct trailer path file and re-run.

Message:	**FATAL ERROR:** **Draft of dredger is greater than the depth of water**
Cause:	The dredger cannot operate if its draught isgreater than the water depth.
Action:	Correct dredger draft and re-run.

Message:	**FATAL ERROR:** **Dredger speed cannot be zero, Problem with trailer 'i'**
Cause:	The model is not appropriate for a stationary drdger
Action:	Set dredger speed to positive value and re-run.