Users Guide To

RMA4 WES Version 4.5

US Army, Engineer Research and Development Center
Waterways Experiment Station
Coastal and Hydraulics Laboratory

This manual and on-line documentation were produced using *Doc-To-Help*®, by WexTech Systems, Inc.

WexTech Systems, Inc.
Valhalla, NY 10595
1-800-WEXTECH

FastTABS (1990-1994) , SMS4-SMS6 (1994-1998),

SMS7 (1999-2002), SMS 8.0 SMS8- (2002-2005), and

SMS9- (2005-2008) SMS 10.0 SMS10- (2008-present) are products of the Environmental Modeling Research Laboratory (EMRL), Brigham Young University.

Users Guide To RMA4 Version 4.5

Last document modification date: Aug 2008.

Effective RMA4 source code modification date: Nov 26, 2007

Original Author: *Dr. Ian King*

Revised and Updated by: *Barbara P. Donnell*

World Wide Web Editor*:* *Clay LaHatte*

Contributing Authors: *Joseph V. Letter, Jr., Gary L. Brown, and Barbara P. Donnell*

Source Code Authors: *Joseph V. Letter, Jr., Gary L. Brown, and Barbara P. Donnell*

Technical Assistance: *Joseph V. Letter, Jr., Allen M. Teeter, Gary L. Brown, and William L. Boyt*

Technical Support

World Wide Web: *http:/chl.wes.army.mil*

E-Mail: *SMS@erdc.usace.army.mil*

Cataloging Information

Letter, Joseph V., Donnell, Barbara P., and others, “Users Guide for RMA4 Version 4.5”, [14 Aug] 2008, [http://chl.wes.army.mil/software/tabs/docs.htp].

Contents

Overview

What Is RMA4

RMA4 is a finite element water quality *transport* numerical model in which the depth concentration distribution is assumed uniform. It computes concentrations for up to 6 constituents, either conservative or non-conservative, within the one-and/or two-dimensional computational mesh domain.

Origin Of The Program

The original RMA4 was developed by Norton, King and Orlob (1973), of Water Resources Engineers, for the Walla Walla District, Corps of Engineers, and delivered in 1973. Subsequent enhancements have been made by King and Rachiele, of Resource Management Associates (RMA), and by the US ERDC WES Coastal and Hydraulics Laboratory, culminating in the current version of the code supported in TABS-MD. Personnel in the Coastal Hydraulics Laboratory at ERDC-WES developed the data input module and gradual boundary condition reinstatement after a flow reversal.

Applications For RMA4

The water quality model, RMA4, is designed to simulate the depth-average advection-diffusion process in an aquatic environment. The model can be used for the evaluation of any conservative substance that is either dissolved in the water or that may be assumed to be neutrally buoyant within the water column. The model is also used for investigating the physical processes of migration and mixing of a non-conservative substance in reservoirs, rivers, bays, estuaries and coastal zones. The model is useful for evaluation of the basic processes or for defining the effectiveness of remedial measures. For most applications, the model utilizes the depth-averaged hydrodynamics from RMA2.

The water quality model has been applied to:

- Define the horizontal salinity distribution
- Trace temperature effects from power plants
- Calculate residence times of harbors or basins
- Optimize the placement of outfalls
- Identify potential critical areas for oil spills or other pollutants spread

- Evaluate turbidity plume extent
- Monitor other water quality criterion within game and fish habitats
- Mixing zone definition
- Determine the limits of salinity intrusion
- Flushing analysis

Capabilities Of RMA4

RMA4 is a general purpose model designed to investigate physical processes which are responsible for the distribution of pollutants in the environment, and for testing the effectiveness of remedial control measures at high speed and low cost. The methodology is restricted to one-dimensional and two-dimensional systems in which the concentration distribution in the vertical dimension is assumed uniform.

RMA4 has the capability to:

- Read one-dimensional and/or two-dimensional geometry file from GFGEN, or allow manual specification of the geometry.
- Read one-dimensional and/or two-dimensional hydrodynamics from RMA2, or allow manual specification of the velocity field.
- Appropriately handle marsh porosity and wetting and drying from RMA2
- Account for rainfall/evaporation that was specified in RMA2
- Handle all one-dimensional flow control structures that are available in RMA2.
- Handle the concentrations related to a RMA2 simulation, which contained rainfall and/or evaporation.
- Re-start (Hotstart) the simulation from a prior RMA4 run and continue.
- Compute mass flux at geometry check line locations.
- Accept boundary condition concentrations by node, line, or mass loading.
- Model up to 6 constituents as conservative or non-conservative using a first order decay.
- Permit temporary storage of re-solution files designed to speed up the solution calculations when the velocity file recycles, such as during a simulation of many repeated tidal cycles.

Additionally, RMA4 has the capability to simulate advection-diffusion in the aquatic environment.

Limitations Of RMA4

The formulation of RMA4 is limited to one-dimensional (cross-sectionally averaged and depth-averaged) and two-dimensional (depth-averaged) situations in which the concentration is fairly well mixed in the vertical direction. It will not provide accurate concentrations for stratified situations in which the constituent concentration influences the density of the fluid.

Note: The accuracy of the transport model is dependent on the accuracy of the hydrodynamics (e.q. as supplied from RMA2). In particular, for RMA4 the shoreline boundaries are of critical importance

If you need to model water quality transport in three dimensional domains, refer to models such as TABS-MDS (previously referred to as WES RMA10).

Governing Equations

The CEWES version of RMA4 is a revised version of RMA4 as developed by King (1989). The generalized computer program solves the depth-integrated equations of the transport and mixing process. The form of the depth averaged transport equation is

Equation 1

$$h\left(\frac{\partial c}{\partial t}+u\frac{\partial c}{\partial x}+v\frac{\partial c}{\partial y}-\frac{\partial}{\partial x}D_x\frac{\partial c}{\partial x}-\frac{\partial}{\partial y}D_y\frac{\partial c}{\partial y}-\sigma+kc+\frac{R(c)}{h}\right)=0$$

where

h	=water depth
c	= concentration of pollutant for a given constituent
t	= time
u, v	= velocity in x direction and y direction
D_x, D_y,	= turbulent mixing (dispersion) coefficient
k	= first order decay of pollutant
σ	= source/sink of constituent
R(c)	= rainfall/evaporation rate

Term Descriptions:

1st term	= local storage
2nd term	= advection term (x)
3rd term	= advection term (y)
4th term	= dispersion (x)
5th term	= dispersion (y)
6th term	= local sources of mass substance
7th term	= exponential decay
8th term	= rainfall/evaporation effects

The basic governing equation for RMA4 is the same as for the sediment transport model, SED2D. The differences between the two models lies in the source/sink terms.

The equation is solved by the finite element method using Galerkin weighted residuals. As with the hydrodynamic model RMA2, the transport model RMA4 handles one-dimensional segments or two-dimensional quadrilaterals or triangles. Curved element edges are also supported. Spatial integration of the equations is performed by Gaussian techniques and the temporal variations are handled by nonlinear finite differences consistent with the method described for RMA2. The frontal solution method is also used in RMA4, as with the other programs in the TABS-MD system, to provide an efficient solution algorithm.

The boundary conditions for the RMA4 equations can be specified in several ways.

- The boundary concentration may be specified absolutely at a certain level regardless of the flow direction.
- The concentration can be specified only when the water is entering the model.
- A mixing zone may be specified just beyond the model boundary to provide the possibility of reinstatement of the constituent into the model that may have crossed the boundary earlier.

Within the one-dimensional formulation of the model, there is a provision for defining the constituent concentration mixing and transport at control structures as they may have been specified in RMA2. These allow for either a flow through condition, as for example a weir type flow, or for a mixing chamber type of flux, which would be appropriate for a navigation lock.

For a more detailed description of the constituent transport model, RMA4, see King and Rachiele, 1989.

Reference: Governing Equations for the Mathematical Model RMA4, Appendix H-A, Ian P. King, Richard R. Rachiele, Resource Management Associates, January 1989

Element Types Supported

RMA4, like other TABS-MD models is capable of supporting different types of elements within the same computational finite element mesh. The types of elements fit into three basic categories:

- Two Dimensional Elements
- Element Types Supported
- Special Elements

Note: Element edges may be either straight or curved (isoperimetric).

Two Dimensional Elements

Two dimensional elements are the customary type used with RMA2 and may be either triangular or rectangular in shape. A two dimensional element possesses a length and a width, determined by the positions of the corner nodes which define the element. The depth at any location within a two dimensional element is obtained by interpolating among the depths of the corner nodes which define the element. Elements may be either straight or curved (isoparametric).

Rectangular Element Triangular Element

One-dimensional Elements

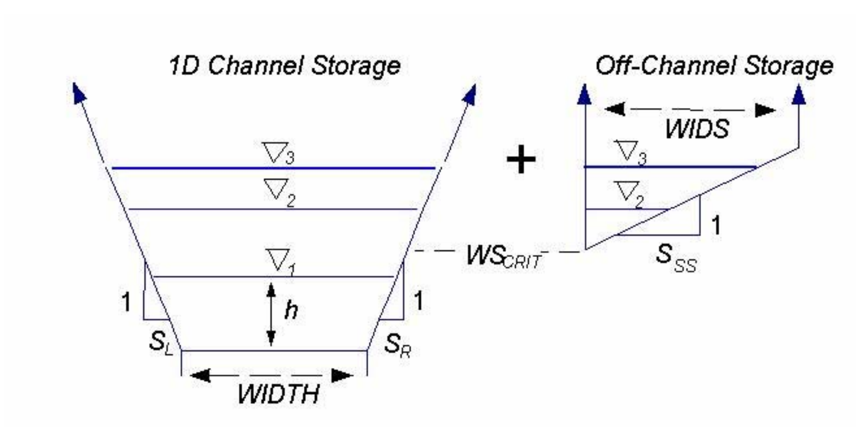
A one-dimensional element is a simplified element, which is composed of two corner nodes and one midside node. The Finite Element Governing Equations for one-dimensional elements are based on a trapezoidal cross section with side slopes, and an off channel storage area. The depth at any location along a one-dimensional element is obtained by interpolating between the depths of the two corner nodes which define the element.

The Basic One-dimensional Element

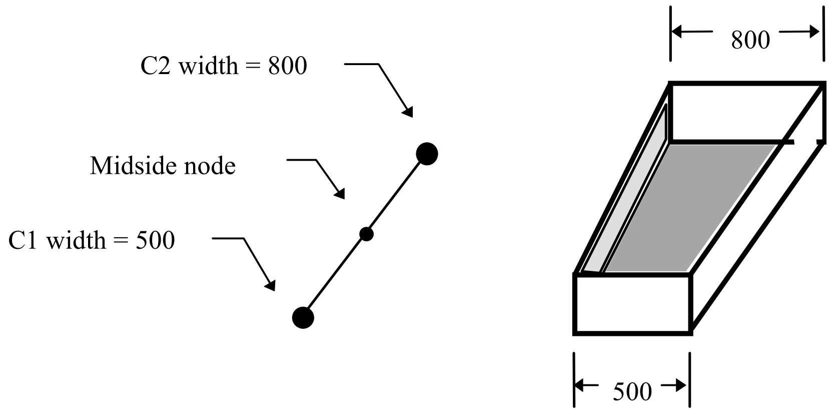
The basic one-dimensional element is composed of two corner nodes and one midside node, and may be either straight or curved.

The numerical model's Governing Equations for one-dimensional elements are based on a trapezoidal cross-section with side slopes and off-channel storage. To describe the trapezoidal cross-section, you must assign for each one-dimensional corner node a bottom width (when the depth=0), a left and right slope (S_L and S_R). The designation of left and right slope is arbitrary. If the values of S_L, S_R, and the off channel storage width are zero, the total trapezoidal shape reduces to a rectangle.

If additional off channel storage contributions are needed, they may be added by assigning an off channel storage width (WIDS), critical water surface elevation to activate off channel storage (WS_{CRIT}), and an off channel storage slope (S_{SS}). If the values of S_{SS} and WS_{CRIT} are not specified, the enhanced off channel storage features degrade to that of the previous versions of RMA2 (prior to version 4.52). All slopes reference a given distance for one unit of rise. A representative cross section of 1D channel storage and off-channel storage is shown below.



A basic one-dimensional element can have a different width at each corner node. A basic straight sided element with zero side slopes, but different width assignments at each corner (figure on left) will have a shape that looks like the figure on the right:



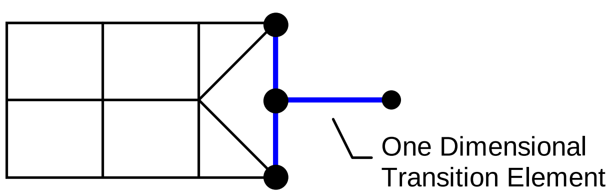
Special Elements

Some one-dimensional elements serve special purposes. These elements fall into three categories:

- Transition Elements
- Junction Elements
- Control Structure Elements

Transition Elements

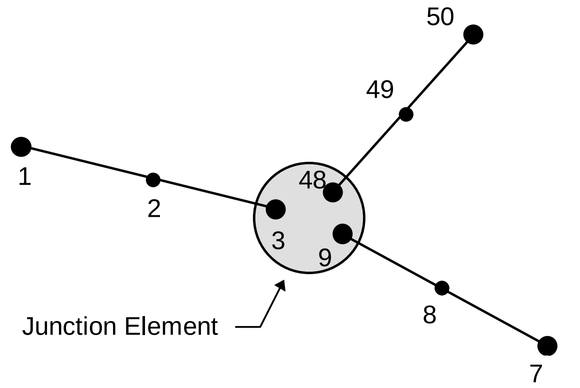
A transition element is required anytime you want to use a one-dimensional element in your mesh. This type of one-dimensional element makes the transition between the two dimensional elements and the one-dimensional elements.



For additional information on Transition elements, see the RMA2 and the GFGEN reference manual.

Junction Elements

A Junction element is a special one-dimensional element used to describe the proper characteristics where *three or more* one-dimensional elements intersect. Junction elements are defined using IMAT values of 901, 902, or 903. The junction element *is* the *point* where the other one-dimensional elements connect. There are as many nodes defining this junction element as there are one-dimensional elements connecting to it.



GE elem# 3 9 48 0 0 0 0 0 imat#

For example, if there are 3 one-dimensional elements connecting at a point, then that point is the junction element, and it is composed of 3 nodes.

Note: The functional limit of the model on the number of elements entering a junction is 8, as dictated by the number of nodes in an element.

For additional information on Junction elements, see the GFGEN and/or RMA2 reference manual.

Types of Junction Elements

The junction element is typically assigned during the RMA2 hydrodynamic simulation. There are three categories of Junction elements:

- Water Surface Junction (IMAT = 901)
  All water levels match at the junction.
- Total Head Junction (IMAT = 902)
  All total energy heads match at the junction.
- Momentum Junction (IMAT = 903)
  Momentum is conserved in the primary channel. The first 2 nodes in the junction element define the primary channel within which momentum will be conserved. Water levels for junction ends of the remaining elements all are set to the average water levels of these first two nodes.

Control Structure Elements

A Control Structure element is used to simulate obstructions in the flow path, such as weirs, dams, flood gates, etc. A material type greater than or equal to 904 activates the control structure logic. The modeler must specify the identifying type of each structure along with the coefficients of the equation defining that type (RMA2, FC card). The control structure typically is assigned during the RMA2 hydrodynamic simulation. The concentration mixing exchange parameters for each control structure type are specified within the RMA4 run control with a CS card(s), as described on page 91.

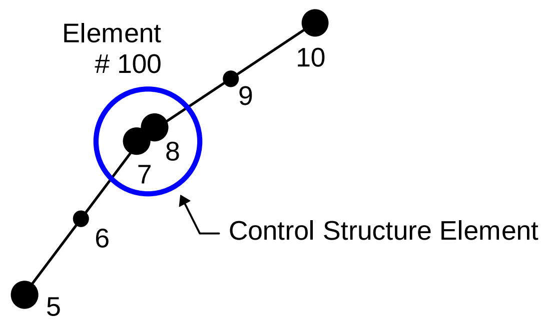
Types Of Control Structure Elements

There are several types of control structures elements:

- Type 1, Point Source of flow, like a pump or storm drain
- Type 2, Flow is a reversible function of head loss, like an open culvert
- Type 3, Flow is an irreversible Type 2, like a flap culvert
- Type 4, Flow is a function of water surface elevation, like a weir
- Type 5, Type 2, with head loss as a function of flow.
- Type 6, Flow is an irreversible Type 5.
- Other types may be developed later.

1D Control Structure

A one-dimensional Control Structure element is a single point which contains two nodes and has an IMAT value ≥ 904. The order of the node numbering at a Control Structure element should be that the side with higher elevation comes first, then the side with the lower elevation. This is generally the “upstream” side of the structure followed by the “downstream” side.



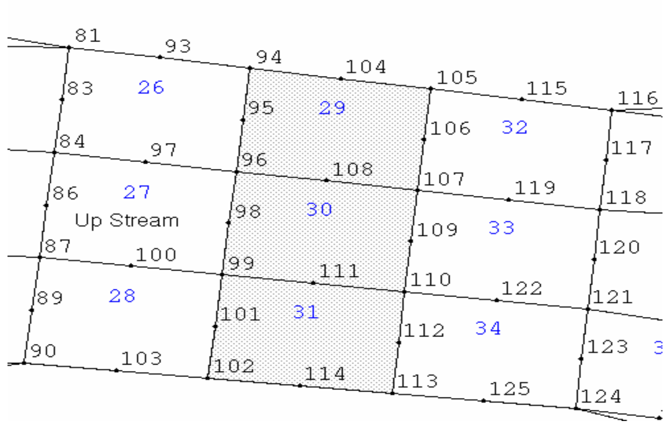
GE 100 7 8 0 0 0 0 0 <u>904</u>

2D Control Structure

The 2D control structure element is available for versions 4.52 of RMA4 or higher.

A two dimensional Control Structure elements have a unique connection table and has an IMAT value ≥ 904. The 2D control structure typically is assigned during the RMA2 hydrodynamic simulation. The selection of the type of control structure (RMA2 FC Card, NJT) is important. You will need to record the width of each structure because the discharge referenced in the equations for 2D control structure equations are based upon the <u>per unit width.</u>

What makes the 2D Control Structure element construction unique is that the element has to be numbered such that the upstream edge lies in nodal values 1-3 of the element connection table. Similarly, the down stream edge must fall in positions 5-7 of the element connection table. The user must verify that this was done correctly before proceeding. The mid side nodes connecting the “upstream” side of the structure with the “downstream” side of the structure must be set to zero (4^{th} and 8^{th} nodal value in the connection table). In the example given below, there are three elements (29-31) that represent three 2D control structures located side-by-side spanning from bank line to bank line. To convert these elements to control structure elements, the place holders for mid side nodes 104, 108, 111, and 114, are filled with zeros. In this example, the control structures were defined with a material type of 904, as shown in the GE-Cards by the bolded fields.



GE	29	94	95	96	**0**	107	106	105	**0**	**904**
GE	30	96	98	99	**0**	110	109	107	**0**	**904**
GE	31	99	101	102	**0**	113	112	110	**0**	**904**

For additional information on Control Structure elements, see the GFGEN reference manual.

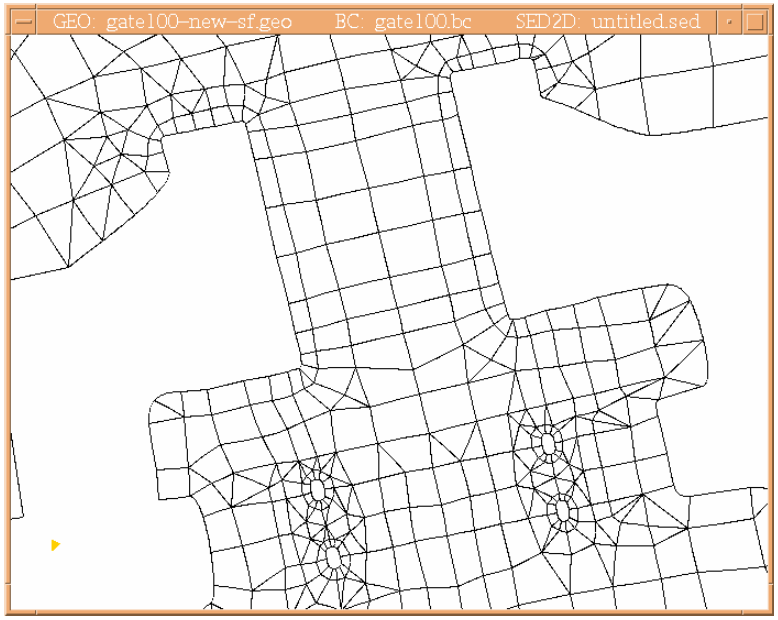
Curved Element Edges

Although it is not required to prescribe curved (isoperimetric) external boundaries, they may be used to achieve aesthetics when viewing the mesh, add length without additional resolution, and to aid in mass conservation for transport applications. Curving should be accomplished prior to generating the hydrodynamic solution.

It is very important to have a mesh that follows the "boundary break angle of < 10 degree rule" and "smooth curved exterior edges" for either a RMA4 water quality or SED2D sediment transport simulation.

An element edge may be *curved* by moving the (x, y) location of the midside node. See the GFGEN and/or RMA2 reference manual for more information.

Example of a Properly Curved Exterior Boundary.



Note: SMS will display curved elements if the curved *mesh edges* are activated in the mesh display options.

What's New And What's Different

During the process of writing this manual, the RMA4 source code has been reviewed many times. As a result, enhancements have been made and many code errors have been eliminated.

Additions

The following user controls have been added to RMA4 version 4.50

- MC Card Added - Mass Conservation Check added.
- PE Card Added - Auto Peclet for Diffusion
- RA Card Added - Rainfall And Evaporation
- Off-Channel Storage logic for compatibility with RMA2 v 4.52

The following enhancements have been added to RMA4 version 4.54

- Diagnostic file, "r4negconc.dat" created to flag negative concentration
- Expiration date added
- Dynamic Memory Allocation added.

The following modifications were added to RMA4 version 4.56

- DO/BOD with reaeration

Modifications

- TP Card was modified to permit additional print control

Deletions

- GC card was replaced by the GCL card
- BLN input option, nodal mass load, was removed with version 4.5

Corrections

- Corrected erroneous sign when decay coefficient is used

System Requirements

Because RMA4 is written in standard FORTRAN 77 code, it can be compiled and executed on many different types of computer systems. System requirements such as RAM and disk space will differ depending upon the size of your project. The size of your project may require you to choose to use one type of system over another, or require an upgrade of your current system. In general, you can use the table below as a guide.

Max Number Of Elements	Required RAM (Mbytes)
5000	8
8000	16
15000	32

Note: The RMA4 source code "include file" must be edited to re-dimension arrays pertaining to the number of elements, nodes, etc., if the current dimensions are not adequate for your project. See "Redimensioning RMA4" on page 127. The User may alter array sizes at run time by using the dynamic memory allocation input file “r4memsize.dat”

In addition to RAM, the processing power, or speed of your system is also an important consideration when using RMA4. Although there are ways to speed the computation (see “Speeding Up The Computation” on page 68), RMA4 can require a large amount of computational power during a simulation, and can take a very long time to complete the run. Although RMA4 is much faster than RMA2 for each time step, the problem typically requires a much longer simulation time to determine pollutant transport than is required to define a representative set of hydrodynamics. Always seek a powerful processor to yield an efficient computational environment.

In some cases, machine precision may become an issue. If you suspect RMA4 will encounter very large or very small numbers during a simulation, be aware that machines with a smaller word length, such as PC’s, may not perform to your expectations.

For additional information, see “Performance Enhancements” on page 125.

Personal Computer Systems

The most commonly used personal computer systems on which RMA4 is run are the "IBM compatible" PC, and the Macintosh. Remember that the size of your project dictates the amount of RAM your system will actually need.

PC's (IBM Compatible)

The minimum requirement for a PC is a 386 CPU with 4 Mbytes of RAM, and 20 Mbytes of free disk space, and the DOS extender program DOS4GW.EXE from Rational Systems™.

If you do not have enough physical memory to run RMA4, and you have a windowing program such as Microsoft™ Windows™, which allows the use of DOS virtual machines, or DOS windows, you may be able to run RMA4 in this environment. You need to be sure under this environment that the size allocated for *virtual memory* will provide you with enough total RAM to execute RMA4. Be aware that RMA4 may run much slower with this than if your system actually had enough physical RAM.

A more productive environment would include a configuration of a 3.0 Ghz processor, 2 GB of RAM, 200 GB hard disk drive, and a data back up device.

Macintosh

The minimum requirement for a Macintosh is a 68030 processor, 4 Mbytes of RAM, and 20 Mbytes of free disk space.

Mini Computer Systems

Workstations are a good choice for running RMA4. They provide a stable platform without the computational costs associated with many mainframe computer systems. Today's workstations exceed 600 Mhz processor speed and are normally shipped with at least 1.0 Gigabyte of memory and 200 gigabytes of disk space. This configuration would be capable of modeling a moderately detailed RMA4 simulation.

Mainframe/Supercomputer Systems

If you have access to a mainframe, or supercomputer, it may be very useful when running RMA4. Many are faster and use more precision than do some workstations. Be sure, however, to have enough available disk space, at least 20 Mbytes, allotted to your userid for installation and storing RMA4 run output.

Using RMA4

Assumed User Knowledge Base

RMA4 can be used by engineers and scientist to solve water quality transport problems. Users are cautioned that the program is relatively easy to use but somewhat more difficult to use properly. Persons using the program are assumed to be familiar with using a computer system. Knowledge of basic concepts in numerical methods is necessary.

The modeling of dissolved oxygen (DO) and biological oxygen demand (BOD) requires a background in the environmental sciences.

The RMA4 Modeling Process

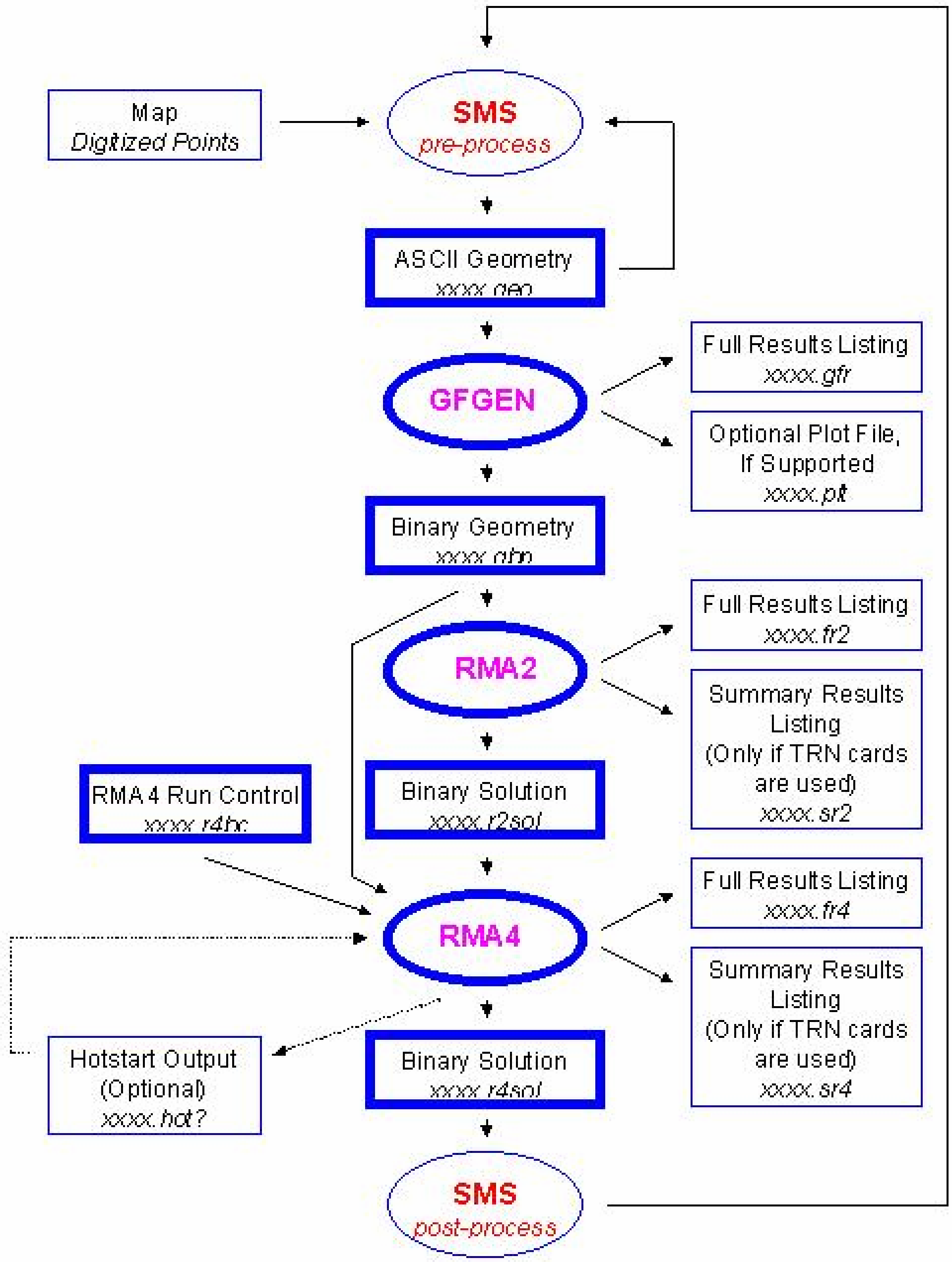
Like all of the TABS-MD models, RMA4 modeling process requires a basic set of input files and output files to conduct the simulation. The basic file essentials include: run control instructions including boundary conditions and pollutant loads, geometry, hydrodynamics, and concentration output.

RMA4 typically uses a hydrodynamic solution from RMA2 which has been obtained from the exact same geometry to be used for RMA4. An RMA4 simulation is always run in transient, i.e., dynamic mode, regardless of whether the RMA2 hydrodynamic results are steady state or transient. If the RMA2 hydrodynamics are steady state, these same results are reapplied at every time step of the RMA4 simulation. If the RMA2 hydrodynamics are transient (dynamic), you have control of the starting and stopping time of the RMA2 solution to be applied to the RMA4 simulation. Again, the hydrodynamics are rewound to the designated starting point if the RMA4 simulation is longer than the supplied hydrodynamics.

For information on the input and output files, see “Data Files” on page 25.

The following flow chart illustrates the RMA4 modeling process. Items with bold borders are required, others are optional.

Flowchart of RMA4 Modeling Process



How RMA4 Works

RMA4 makes use of an advection-diffusion equation to obtain a solution. From a set of initial conditions, RMA4 calculates concentrations for a specified number of constituents.

RMA4 has the capability for the simulation of the advection-diffusion process in the aquatic environment. The methodology is restricted to two dimensional systems in which the vertical concentration distribution (in the third dimension) is assumed uniform. Although this model is presented as an application to water quality problems, the technique is applicable to a large class of other environmental transport problems.

To use RMA4, the user must supply the basic physical description of the system (which is usually generated by SMS), the velocity fields (which is usually generated by RMA2), and the boundary conditions and pollutant loads. The output from the model consists of tabulated values (for viewing in a spreadsheet program) and binary solutions file (for viewing within SMS) of pollutant concentrations throughout the system for time steps during the simulation. The model will treat pollutants either as conservative or non-conservative using a first order decay (see "FQC Card: Fluid Qualities - Decay Control" on page 98).

The model is written in FORTRAN 77, is compatible with FORTRAN-90, and uses the finite element method of numerical solution. The program has a number of user options (explained in the RMA4 control cards) and can be operated with or without the time terms activated in the governing equations. Currently the model is most valuable for the investigation of physical processes which are responsible for the distribution of pollutants in the environment and for testing the effectiveness of remedial control measures at high speed and low cost.

Dynamic Memory Allocation

Effective in July 2002 with version 4.54 of RMA4, the code was converted to FORTRAN 90 dynamic memory allocation. This allows the user the ability to control the dimensionality of the model without having to recompile.

The file “r4memsize.dat” allows the user to customize the dimensions of the RMA4 model..

By placing the file named “r4memsize.dat” in the same directory as the RMA4 run control file (.bc file), the dimensions of the model can be customized for your project specific requirements and accommodate your personal computer’s memory/disk space specifications.

The default memory model is set for approximately 2 gigabytes of random access memory (RAM). The user may customize the memory requirement by modifying the third line of the file “r4memsize.dat”. A representative sample of the contents of this file is shown below. The first line contains the word “user”, lower case, in columns one through four. The second line is a character string that serves as a header for the third line. The third line of data are integer values corresponding to variable names in the header list. These integer values are list directed, so at least one space or a comma must separate each variable.

Example: Contents of “r4memsize.dat”

```
user
MND   MEL   NBS     MFW MSV   MSTEP MPB    MBNP MBB MCC MCCN MQAL MFCS
40000 20000 5000000 500 40000 34010 340010 300   10  100  300   6 20
```

Where the dimension variables are defined as

MND	Maximum number of node numbers
MEL	Maximum number of element numbers
NBS	Maximum buffer size when using the Frontal solver

MFW	Maximum front width
MSV	Always set equal to MND
MSTEP	Maximum number of time steps within the simulation
MPB	Maximum summary listing buffer (TRN-card nodes * MSTEP)
MCC	Maximum number of continuity check lines (GCL strings)
MCCN	Maximum number of nodes on a check line (includes corner & midside nodes
MQAL	Maximum number of quality constituents (usually = 1)
MFCS	Maximum number of flow control structures

Modes Of Operation

At present there are three modes of operation for running RMA4. These modes are:

- Running In Interactive Mode
- Running In Batch Mode
- Running In Auto Mode (Simulation Super File)

Typically RMA4 is set to automatic, simulation super file, mode for compatibility with the simulation super file saved by SMS

RMA4 may be executed either in *interactive mode, batch mode, or automatic mode*. The mode is determined by the value of the `IBATCH` variable within the program. You can change the mode only if you have the source code and a FORTRAN compiler.

Running In Interactive Mode

When the IBATCH variable is set to zero, the model is set to run interactively. To run interactively, execute RMA4 directly and answer the series of questions referring to input and output file names. For instructions, enter a '?'.

The example table below llustrates the startup procedure for running a RMA4 hotstart simulation in interactive mode. Note that the question regarding a particular file will only be asked only if that file was activated on the $L card. You may choose to null or prevent that file from being opened by typing '**null**' in response to a file name prompt. The italics indicate the user response. For information on RMA4 files, see "Data Files" on page 25.

Example of a RMA4 interactive response session

```
===== RMA4 VERSION 4.50   1D AND 2D CAPABILITY =====
===== LAST MODIFICATION DATE:  07-15-2000      =====
RMA4 Program Dimensions for this executable:
     MAX NODE                  =    33000
     MAX ELEMENT               =    11000
     MAX FRONT WIDTH           =      550
     MAX CONTINUITY CHECKS     =      150
     MAX TIME STEPS            =     1500
     SCRATCH BUFFER            =  2000000
     MAX SUMMARY PRINT BUFFER  =    24000
     MAX CONSTITUENTS          =        6
```

ENTER RUN CONTROL INPUT FILE NAME
river2.r4bc
ENTER FULL PRINT OUTPUT FILE NAME
river2.r4fp

RMA4 VERSION 4.50 READING INPUT DATA ... UNIT=   3
T1   RMA4 example
T2   Barbara P. Donnell 10-12-1995, and again in 4-7-2000
T3   Medora Crossing - RMA4 run control - HOTSTART thru hour 24.0
CO   geo r2sol hoti hoto altbc r4sol fp   sp
$L   10   20   30   0   31   32   33   34
<< CAUTION >> Alt BC file is turned off by $L

ENTER SPECIAL PRINT OUTPUT FILE
river2.r4sp

ENTER INPUT GEOMETRY FILE (binary)
river.geobin

ENTER VELOCITY INPUT FILE (RMA2 binary solution)
river_ss.r2sol

ENTER BINARY INPUT RESTART/HOTSTART FILE (binary)
river1.r4hot

ENTER OUTPUT RESTART/HOTSTART FILE (binary)
river2.r4hot

ENTER FINAL RMA4 RESULTS FILE (binary)
river2.r4sol

Pseudo-Interactive Mode

The program may be run in pseudo-interactive mode. This is useful on most unix and mainframe computers, such as the Cray T3E located at the WES High Performance Computer Center (HPC).

Because of the way data are stored and accessed on the WES HPC, including the Mass Storage Facility (MSF), and the changing rules regarding the amount of memory available per processor, there is not available a straight forward way to describe the execution of the TABS-MD models on a super computer.

For a pseudo-interactive mode, the IBATCH variable is set to interactive (IBATCH=0). If you have prepared a run file, containing the input file names in the proper order, you can free up the terminal session for other activities. To run the RMA4 model in pseudo-interactive mode, the following commands may be used. The direct (<) command, re-direct (>) command, background run request (&), and the optional unix C-shell *nohup* utility, which ignores hang-ups and quits.   Used together, these provide a means to capture the “screen” and safeguard to insure that the RMA4 continues to execute if the terminal connection is dropped. The actual command line would be similar to the following.

nohup RMA4v450.exe < *run_file_name* > *output_file_name &*

Where the run file name would satisfy the order dependent requests for the file names. An example of the contents of the run_file_name follows. This particular example was for a RMA4 hotstart simulation.

Example of a pseudo interactive run file.

```
river2.r4bc
river2.r4fp
river2.r4sp
river.geobin
river_ss.r2sol
river1.r4hot
river2.r4hot
river2.r4sol
```

Running In Batch Mode

When RMA4 is compiled in batch mode (IBATCH=1), the model is ready to be used on a large mainframe computer system, such as the Cray T3E at the Waterways Experiment Station. On the WES Cray computers and other mainframes, the batch mode execution of RMA4 requires computer specific job control language. Since the specific syntax changes frequently, it will not be presented in this document. The WES Information Technology Laboratory (ITL) help line is available to assist super computer users. The ITL High Performance computing assistance help line can be reached by calling 1-800-LAB-6WES extension 4400, option 1.

The advantage of running in batch mode is it generally allows access to more CPU time and more system RAM. Another advantage of batch mode is that it does not tie up your terminal. Once the job has been submitted to the system, your terminal is free and available for you to perform other tasks.

Note: The WES High Performance Computer Center (HPC) has established policies and procedures that may change. Therefore, be sure to check with the HPC before planning and/or executing projects with hard milestone dates that could be affected by the operational demands on the DOD HPC.

Running In Auto Mode (Simulation Super File)

Automatic Super Mode in RMA4 (version 4.5 or higher) was designed to be compatible with the simulation super file concept in SMS version 7.

When RMA4 is compiled in automatic super mode (IBATCH=2 or 3), all input/output file names are controlled based upon a key-word character string, as shown in the table below. The user either edits the simulation super file created by SMS or creates a super file containing a list of key-words and file names. The $L card still activates the existence of the input/output files.

The key-word must start in column one and be spelled exactly as shown, note that both upper and lower case is permitted. The user does not have to be overly concerned with the order in which files names are provided. The only order dependency is for the FILE_PATHWAY key-word which sets the directory information defining the pathway for all subsequent files. To de-activate a file I-O request, simple enter the word, "**null**" (in lower case), for the file name.

When RMA4 executes, the only interactive question presented to the user is a request for the name of the simulation super file (*site.sup* in this example). The actual command to run the RMA4 model in this mode could use the direct (<) and re-direct (>) commands, as shown. The unix C-shell *nohup* utility and (&) background request is optional.

nohup RMA4v450.exe < *site.sup* >site.screen &

Super File Key-Word	File Name	Description
FILE_PATHWAY	/disk2/proj/	Sets the pwd for these I/O files
R4_BC	site.r4bc	RMA4 run control BC file, (input)
R4_FRL	site.r4frl	RMA4 full results listing
GEOMETRY_BIN	site.gbn	GFGEN binary geometry, (input)
R4_HOTIN	site.r4hot1	RMA4 initial condition hotstart, (input)
R4_HOTOUT	site.r4hot2	RMA4 binary hotstart output
R2_SOL	site.r2sol	RMA2 binary hydrodynamic solution file (input)
R4_SOL	site.r4sol	RMA4 solution
R4_SRL	site.r4sum	RMA4 summary results listing via TRN-Cards
R4_ALTBC	site.r4altbc	RMA4 alternate BC transient file, (input)
R4_PARAMETER	site.r4param	RMA4 auto Diffusion coefficient selection

Example of a project specific (for the Craney Island Project) super file for a RMA4 simulation

```
R4_BC        craney_v45.r4bc
R4_FRL       craney_v45.r4fp
R4_SRL       craney_v45.r4sp
GEOMETRY_BIN craney_sfx.geobin
R2_SOL       craneyr.r2sol
R4_HOTOUT    craney_v45.r4hot
R4_SOL       craney_v45.r4sol
R4_PARAMETER craney_v45.r4auto
```

Guidelines For Obtaining A Good Solution

All aspects of the geometry, the RMA2 hydrodynamics, and the RMA4 numerical model simulation must run in harmony. In addition, the RMA4 run control (boundary condition) file must contain the proper information if the simulation is to be successful. The RMA4 interface was made available in July 2000 for version 7 of SMS. It is also permissible to use a text editor to create the RMA4 run control BC file.

Note: The first requirement for a successful RMA4 numerical model is preparing a good mesh with emphasis on smooth wet/dry boundaries. Additional guidelines can be found in detail in the GFGEN geometry manual.

Have A Good Hydrodynamic Solution

Be sure you have a good solution from RMA2. The RMA2 solution is used as input to RMA4 and is a critical component of the constituent modeling process. It is unrealistic to expect the RMA4 model to produce answers that are better than the hydrodynamics from which they were derived. Take for example the situation where an average daily velocity field is input to the RMA4 simulation with hourly pollution loading. The model will calculate concentration on an hourly basis, but the effects of current shifts of less than daily duration will not be represented. There are other considerations as well, for more information, see “Guidelines For Obtaining A Good Solution” in the RMA2 Users Guide.

Pre-RMA4 Simulation Check List

After the mesh has been constructed and the hydrodynamic solution has been obtained from RMA2, the following check list can be helpful prior to running RMA4. Be sure to:

- If marsh porosity effects of wetting and drying are required, they will have to be predefined in the RMA2 hydrodynamics run.
- If rainfall or evaporation effects are required, they will have to be predefined in the RMA2 hydrodynamics run, and provided as input in the RMA4 water quality run control file using the RA card. Be sure to use the same rates and to check the units.
- Label the T3 card with a descriptive title for this simulation.
- Verify that the machine identifier on the $M card is correct for your computer system.
- Set the appropriate input/output switches for file control on the $L card.
- Specify diffusion coefficients for every element material type defined in the mesh (DF card) or employ the automatic assignment feature (PE card).
- Set the RMA4 timing control on the TC card.
- Set the hydrodynamic timing control on the TH card, making a point to avoid the spin-up section of the hydrodynamic solution.
- Check that the initial condition (either read from the IC card or from a hotstart file) is representative of the modeling domain.
- Double check your units. The GFGEN geometry and RMA2 hydrodynamic solution units must agree with the input RMA4 parameters (SI-card and others). Otherwise you must use the proper scale factors to have the geometry (GS card), and the hydrodynamics (HS card) converted to the desired units. Unless otherwise stated, all RMA4 input parameters and boundary condition units must be consistent with the geometry and hydrodynamic input files after the designated scaling factors have been applied.
- Set the decay coefficient for each constituent on the FQC card.
- Evaluate the location and type of boundary conditions with the BC card and BL card.
- To determine the RMA4 model spin-up, look for evidence of equilibrium concentrations.

Maintaining Stability

RMA4 will usually obtain a solution unless there is something drastically wrong. However, be aware that, unlike RMA2, RMA4 does not use an iterative process to arrive at a solution, hence there are few indicators in the full results listing as to how well the model performed. If you observe negative concentrations or concentrations which are exceedingly out of range from the expected, you may be experiencing oscillation problems. If oscillation problems persist after you have checked the mesh characteristics and pre-simulation check list, you should examine the solution to determine where the simulation *first* appears to begin oscillating. With a graphical user interface such as SMS, you can view the various time steps from the run to find the location in the mesh, and the time, where the problem begins.

Reasons For Instability

There are basically three causes for instability:

- Geometry Related Issues
- Inappropriate Diffusion Coefficients
- Re-Solve Error

Geometry Related Issues

The most common geometry problem is sharp wet/dry boundary break angles in excess of 10 degrees. Applying sufficient resolution or curving the edges on the wet/dry boundary is required to provide a smooth boundary. Sharp boundary break angles are not only an issue for the exterior of the mesh domain, but must also be considered at any interior locations which may become dry, turning these locations into temporary external boundaries.

Remember that adequate resolution is also required in locations of sharp concentration gradients. Geometry related issues should be addressed, rectified, and the RMA2 hydrodynamic model re-run so that the velocity field will reflect the modifications.

If you failed to recognize and address these locations prior to running RMA4, you may consider increasing the diffusion coefficients in these areas. The appropriate value for the diffusion coefficients is related to the size of the elements.

Inappropriate Diffusion Coefficients

Every material type in the mesh must be assigned a diffusion or mixing coefficient. If this information is missing for any one material type, unexpected and unreasonable results may be calculated.

If the diffusion coefficients are too small in relation to the size of the elements and the flow within those elements, oscillations can occur. The classic symptom is to experience higher concentrations within the mesh than were specified as boundary conditions. Another symptom is the existance of negative concentrations. Negative concentrations larger than -1e-4 are indicative of numerical oscillation and should be addressed.

If the diffusion coefficients are too large, the computed concentrations will be smeared, sluggish, and highly diffused.

For guidelines on setting the diffusion coefficients see "Specifying The Diffusion Coefficients" on page 34. These guidelines are intended to provide a starting reference point. You may need to alter the diffusion coefficients to achieve stability.

Re-Solve Error

An input error for the re-solve technique that caused an inappropriate re-solution record to be read for a given time step would cause a shock to the simulation and render oscillations in the solutions. Usually these types of errors enlarge with time.

Critical Check Points

First determine if the solution is numerically stable and realistic. A critical check point is to determine if the maximum computed concentration exceeds the maximum concentration attributed by the specified boundary condition(s). Another critical check point is to determine if negative concentrations are present (found in the full results listing file, screen file, diagnostic file, or by graphical analysis). A poor mesh network with insufficient resolution and or sharp breaks along boundary element sides will cause unexpected loss/gain of total mass.

Basic Operation

Data Files

RMA4 may read and write several files during a simulation. The number and type of files depends upon choices you make about how the simulation will run and what type of information you want to see in the results.

The $L card is used to specify the files that will be utilized by RMA4. The types of files RMA4 uses consist of:

- Run Control Files
- Geometry File
- Hydrodynamic Solution File
- RMA4 Hotstart File
- Results Listing Files
- Solution File
- Re-Solve Files
- Diagnostic Files

For a normal run, you should specify as input a run control file and the binary geometry file from GFGEN and the binary hydrodynamic solution file from RMA2, and as output a full results listing and a solution file. See "The RMA4 Modeling Process" on page 13 for additional information.

Run Control Files

The RMA4 run control file is normally named with the extension **".r4bc"**. There are many options available to you when running an RMA4 simulation. The run control file (a.k.a. boundary condition file) is what you use to tell RMA4 how to run the simulation. Every action taken by RMA4 is defined or modified in the run control file.

In addition to the run control file, you may specify an alternate file for dynamic boundary condition data. This additional file is used in conjunction with the primary run control file. The $L card is used to specify an alternate boundary condition file.

Once control has been transferred to the alternate boundary condition file, the primary run control file cannot be accessed by the model again during the current execution.

Note: It is recommended that you use the primary run control file for all RMA4 input. The use of an alternate dynamic boundary condition file is available but not supported. Current versions of SMS do not save the RMA4 run control file.

Example of a typical RMA4 run control file.

Title Card
English Units
Comment
I/O Control
Computer Type
Print Control
Summary Request

Fluid Quality

Geometry Check
Lines

Init Concentration
Peclet
Re-Solve OFF
RMA4 time
RMA2 time
Inflow-Conc

End of a Time Step
STOP the run

```
T1 RMA4 example - ENGLISH UNITS
T2 Donnell:  4-14-2000
T3 Medora - RMA4hot, run 24 hr, PE=20
SI   0
CO  geo r2 hotn abc r4sol hoto  fp  sp iwpar
$L  10  20   30   0    31   32  33  34   40
$M   4
TP   1  1   12  -1  0  0
TRN  7  55  88  122  156  208  296  424
FT  17.0
FQ     1    0
FQC    0
GCL    1    1    3    5    7    9   11    -1
GCL    2   13   15   18   21   24   27    -1
GCL    3   30   32   35   38   41   44    -1
GCL    4   47   49   52   55   58   61    -1
GCL    5   64   66   69   72   75   78    -1
GCL    6   81   83   86   89   92   95    -1
GCL    7   98  100  103  106  109  112    -1
GCL    8  115  117  120  123  126  129    -1
GCL    9  132  134  137  140  143  146    -1
GCL   10  149  151  154  157  160  163    -1
GCL   11  166  168  171  174  177  180    -1
GCL   12  203  205  208  211  214
GCL       217  220        -1
GCL   13  292  294  297  300  303
GCL       306  309  312   -1
GCL   14  359  361  364  367  370
GCL       373  376        -1
GCL   15  419  421  424  427  430
GCL       433  436        -1
IC    1   0.0
PE    1    20  1.00   1   1
RE   -1     0     .25
TC   0.  1.0   25  24.0   -1
TH   0    0
BCL  1    100  0   0
END
END
 . .  many END cards were deleted for readability
END
STOP
```

Geometry File

It is possible to code the entire mesh in the RMA4 run control file. However, this is highly discouraged. The ability to code mesh geometry for RMA4 is intended for making minor geometry modifications when testing a proposed change to the mesh.

The mesh geometry which RMA4 will use is normally defined in a binary file produced by the Geometry File Generation program, GFGEN. This mesh geometry file consists of the nodes and elements that define the size, shape, and bathymetry of the study area.

RMA4 will not read a text, or ASCII, type file as input for the mesh geometry. If you do not have a binary geometry file, and you do not want to code the mesh geometry in the RMA4 run control file, you must run GFGEN to obtain the binary geometry file before running RMA4. The $L card is used to include the geometry file in the simulation run.

For more information regarding the geometry file, see the GFGEN User's Manual. For a detail description of the binary geometry file structure, see "GFGEN Binary Geometry File Format" on page 139.

For small test cases, it is possible to input the element connection table and other geometry information for each node with user input on the GE-, GNN-, and GWN-cards.

In most applications, the finite element network used for RMA2 and RMA4 should be identical. Therefore the network should be designed to address both hydrodynamic and quality issues.

Hydrodynamic Solution File

The velocity field that RMA4 will use is normally defined in a binary file produced by the TABS-MD RMA2 hydrodynamic model. The $L card is used to include the RMA2 hydrodynamic binary solution file in the simulation run. For more information on the binary RMA2 solution file structure, see "RMA2 Binary Solution (u,v,h) File Format" on page 140.

For simplistic test cases, it is possible to input the velocity and water depth at each node with user input on the HD-, HU-, and HV-cards.

RMA4 Hotstart File

Hotstarting may be desired when you have a limit on run time for a simulation, or you only want to retain certain time intervals of the solution. A Hotstart file is used to preserve the critical information (derivatives, etc.) at the end of a simulation in order for the run to be restarted and continued at a later time.

A Hotstart file can be both an output file and an input file. The Hotstart output from run 1, for example, is typically used as the Hotstart input for run 2. When a Hotstart file is used as input, it defines the initial conditions for the new simulation.

For more information on the binary hotstart file structure, see "RMA4 Binary File Format" on page 141.

Results Listing Files

The listing files can contain a plethora of information pertaining to the simulation results. Upon normal completion of an RMA4 simulation run, a results listing file, or files, may be written, depending upon the settings of the parameters that pertain to results listing files on the $L card.

For details on controlling the length of the results listing files, see the "TP Card: Trace Print by Constituent for All Nodes" on page 122.

Full Results Listing

If a full listing of results is desired, use the $L card to specify a full listing file. The TP card is used to specify the amount of detail and what types of information will be written in the file. The full results listing file may contain: input run instructions, geometry and hydrodynamic information for each node, initial conditions, wetting and drying status, specified boundary conditions, and the nodal concentration results for each requested constituent.

See "Understanding The Full Listing File" on page 47 for a more in depth description of this file.

Summary Results Listing

When you only need a summary of the simulation results at specific nodes, you can request a summary listing file using the $L card. The nodes to be included in the summary listing file are specified using the TRN card. The summary results listing file contains

- A tabular listing of concentrations for all active constituents for each node listed on the TRN card at each time step

Tip: Information in the summary listing can be imported into a spreadsheet program for plotting and further analysis.

For more information on the summary results listing, see “Understanding The Summary Listing File” on page 59.

Solution File

RMA4 will write the final solutions from its calculations to a solution file. The solution file is a binary file that, upon normal completion of the RMA4 simulation run, contains the results of computations for all time steps defined in the run control file (see “Run Control Files” on page 25). The data in the solution file can be graphically analyzed using a compatible post-processor, such as SMS.

The solution file contains the following types of information for each node in the mesh:

- The computed concentration for all active constituents for each node in the mesh for each time step saved

The solution file also contains the material type number for each element. The material type number for a completely dry element will be written as a negative number.

Re-Solve Files

The re-solve technique is an advanced feature that is completely optional. This capability was added to permit speed up of long term RMA4 simulations. The re-solve capability is of value whenever the hydrodynamics are either steady state or have a repeatable cycle for the duration of a long-term RMA4 simulation.

The technique saves the global matrix for each time step in a set of files known as the re-solve files. Typically these files are named "Q001.00**n**", where **n** is an integer starting with 1 and sequentially increments to the total number of time steps that define the repeatable hydrodynamic cycle. Each resolve file size is dependent on the dimensions of the parameter variables NBS and MSV. Therefore this technique generates many large files that can consume a large amount of disk space.

For an example and further discussion of the re-solve capability, see "Speeding Up The Computation" on page 68, specifically the section Save the Global Matrix and Re-Solve.

Diagnostic Files

There are several RMA4 files available, which may provide valuable diagnostic information.

Auto Parameter Summary File

This file contains a one line summary of the minimum and maximum diffusion coefficients calculated for each time step. These statistics are tabulated in this summary file for quick viewing. To obtain this file, set the IWPAR variable to a positive value on the $L card.

Example: Auto parameter summary file contents

```
RMA4 Time=   0.5000  Tot #elems=  7362    Auto-PEC=1   METRIC=0
 MIN Diff=  0.01354  at Elem=  3679  imat=  6  GPEC(imat)=  4.0
 MAX Diff= 45.87533  at Elem=  4031  imat=  2  GPEC(imat)= 12.0

data repeats for each time step simulated
```

Auto Parameter Assignment SCAT2D File

If the user desires to know the details of the automatic diffusion coefficient calculations, then set the ITRACE variable (TP-card) to a non-zero value. The file "r4autodiff.dat" file is created in a 2DSCAT format compatible with SMS which will display the computed diffusion coefficient at the centroid of all elements (excluding control structures and junctions). The table below shows the type of information saved to the "r4autodiff.dat" file. The file is rewound during each time step; consequently only the last simulated time step is saved to this diagnostic file.

SCAT2D

XYD ## Auto-Diff 4 GPEC IMAT X-Diff Y-Diff Elem time/nstep

X-Coord	Y-Coord	GPEC	IMAT	X-Diff	Y-Diff	Elem #	Time/ Nstep
2741224.0	196357.8	10	1	18.381	18.381	1	0.5 / 1
2748797.5	198554.9	10	1	19.023	19.023	2	0.5 / 1
Continued						last	0.5 / 1

In each case the value of ## , the total number of elements requiring a diffusion coefficient assignment, in the SCAT2D file must be checked for accuracy and may have to be modified by hand, because the total number of entries is not necessarily known apriori.

Negative Concentration SCAT2D File

The file "r4negconc.dat" file is created in a SCAT2D format compatible with SMS which can highlight areas within the model domain where negative concentration values are being computed. This file is created whenever the ITRACE variable (TP-card) is non-zero and negative concentrations have been computed. The table below shows an example of the type of information saved in the "r4negconc.dat" file. In this example, there were two nodes which had very small negative concentrations computed for hour=6.0, time step=12 and one occurrence for hour=7.0, time step=13.

```
SCAT2D
XYD  ##   Neg_Conc 5  CONC  u-VEL   v-VEL   DEPTH   NDRY   NODE#  time
7920.3   7544.6   -.0114   .006   .081   0.69   1   7054   6.0 / 12
7917.8   7544.6   -0.007   -.001   .083   0.69   1   7055   6.0 / 12
7920.3   7544.6   -0.008   .008   .099   0.56   1   7054   7.0 / 13
```

For the negative concentration diagnostic file the value of ##, in the SCAT2D file, which represents the total number of detected negative concentration nodal entry values, must be checked for accuracy and will have to be modified by hand, because the total number of entries is not known apriori. For this example ## would be equal to 3.

Using Titles

The ability to add Titles in the RMA4 run control input file provides a means to describe the data which is being modeled. Titles are specified using the T1 and T2 cards and a T3 card.

Note: A Title card must be the *first* card in the run control file, otherwise RMA4 will not recognize the file as valid input.

Including Title Information

Enter the Title and descriptive information about your data on T1, T2, and T3 cards. You may use as many T1 and T2 cards as you wish, and card order is unimportant. Be sure to end your set of Title cards with a T3 card.

The Last Title Card

The last Title card is the T3 card. Only one T3 card is allowed and it must be the very *last* title card. RMA4 reads the '3' to mean the *end* of the Title cards.

Tip: The information on the T3 card is retained by RMA4 and is written into the binary solution file header. Use the T3 card to your advantage. Supply information which will allow your solution file to be more easily identified in the future.

What Kind Of Computer Do You Have?

Why Does it Matter?

Because different computer systems may store and retrieve information in different ways, RMA4 needs to know the type of system on which it is running so it can properly transfer information on the system.

RMA4 solution files and buffer files are written in a binary form. Binary files are strictly associated with the type of system upon which they are created. Word size and record length may be different from one system to the next.

What To Do About It?

Tell RMA4 what type of system on which you will be running by providing a value for the machine identifier with the $M card. The type of computer you specify determines how temporary buffer files will be written and read.

The $M card is necessary if the re-solve technique is activated or if your system does not have enough memory available for the simulation. In this case, RMA4 will write temporary buffer files to your disk. See the chapter entitled "**Performance Enhancements**" on page 125 for additional information on temporary buffer files.

Note: The computer identification ($M card) is vital if there is insufficient memory allotted during execution of RMA4, which would require it to write temporary buffer files to solve the large matrix.

Example: If you are running RMA4 on a DOS or Windows based PC, the machine identifier value should be 1.

```
 .
CO   Running on a DOS or windows based PC
$M   1
 .
 .
```

Specifying Units

RMA4 may be run in either all english or all metric units. Therefore it is necessary to either provide the geometry and the hydrodynamic input field in units of (1) feet and feet/second, or (2) meters and meters/second. The ability to scale the geometry (GS-card) and hydrodynamic files (HS-card) are made available in the RMA4 run control in order to make these data sets unit consistent with other RMA4 run parameters.

The optional SI card in the RMA4 run control, does <u>not</u> cause unit conversions on the geometry or hydrodynamic input to be performed. Those unit conversions are controlled by the GS and HS card. The presence of the SI card will activate <u>some</u> cross reference unit checks, scale the rainfall/evaporation, and label the output files appropriately.

Similarly *all* RMA4 input parameters and boundary condition specification units must be consistent with the input geometry and velocity field, after unit conversion scale factors have been applied. The only exception is the rainfall/evaporation units, since rainfall is typically reported in a smaller length scale and in terms of hours.

The boundary condition specifications are in terms of *mass per cubic-length*. This generalization permits the user to choose the desired units for the pollutant. As long as the units of concentration are consistent for the initial conditions and boundary conditions, any measure of concentration may be used. For your convenience, a unit's summary table is listed below.

Type of Input	English (SI=0)	Metric (SI=1)
Geometry	feet (ft)	meter (m)
Water Depth / Velocity	ft, and ft/sec	m, and m/sec
Volume of Control Structures	ft^3	m^3
Diffusion Coefficients	ft^2/sec	m^2/sec
Rain/Evaporation	in/hr	cm/hr
Concentration	units-of-mass/ft^3	units-of-mass/liter
Mass Loading	units-of-mass/sec	units-of-mass/sec (kg/sec)

For example, if "English" length units of feet are read from both GFGEN and RMA2 input files, then both the GS and HS cards are required to apply the scale factors (0.3048) to convert feet to meters. The RMA4 parameters should be input in metric units: meters for length, either kilogram or milligram for mass, and sec for time. If the mass input units were expressed in kilograms, then the RMA4 output concentrations will be in kg/m^3 (equivalent to ppt, parts per thousand). If desired, the SMS data calculator could then be used to multiply the concentration by 1000 to obtain units of mg/liter (equivalent to ppm, parts per million). If the mass input units were expressed in grams, then the RMA4 output concentrations would be in g/m^3 (equivalent to ppm, parts per million)

1 m^3 = 35.314667 ft^3
1 ppt = 1000 mg/liter = 1 g/liter = 1 kg/m^3
1 ppm = mg/liter = 1 g/m^3
1 ppb = $1x10^{-3}$ mg/liter = 1 μg/liter = 6.29 x 10^{-8} pounds-mass/ft^3

Obtaining The Mesh Geometry

As with all of the TABS-MD system numerical models, the finite element mesh is typically constructed with a graphical user interface, such as SMS, then processed through the GFGEN program.

A basic rule for constructing a finite mesh geometry to represent a given problem, is to move the boundaries of the model far enough from the area of primary interest such that if something is happening at the boundary its effect will be insignificant in the area of interest. When transport simulations follow a hydrodynamic study, it is imperative to consider water quality issues when defining the model domain and its corresponding boundary condition locations.

In almost all model applications the mesh geometry used in RMA2 and RMA4 should be identical. The one notable exception would be a deliberate IMAT=0 setting.

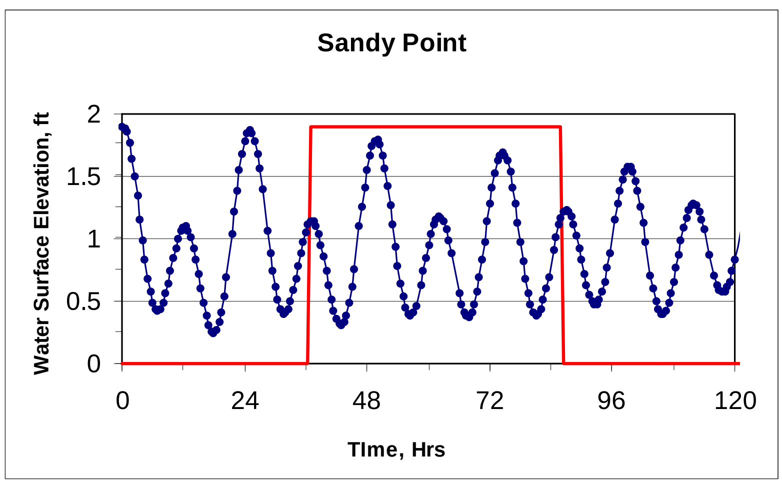
Obtaining The Velocity Field

The accuracy of RMA4's calculations is dependent on the proper temporal and spatial estimate of the velocity field. There are two alternatives for specifying the velocity field.

The first, a simplistic approach, permits you to enter an estimate of the input velocity and water depth representative of some behavior from field experience or statistical analysis for all nodes and for each time interval. The HD-, HU-, and HV- cards will be used to directly specify the flow field for the RMA4 simulation.

The second, and more accurate method is to supply the binary final results solution file from RMA2 as the flow field. If only 1 time step is available on the RMA2 hydrodynamic file, it will be read, rewound, and re-used for all RMA4 time steps. However, you must control the starting and ending time of the transient RMA2 velocity field (TH card) to be used in the RMA4 simulation. The start/stop controls allow the modeler to skip over spin up and/or choose a repeating cycle. Whenever the RMA4 selected delta time step does not match the RMA2 hydrodynamic delta time step, RMA4 will skip over or interpolate (which ever is appropriate) the RMA2 solution. If the RMA2 solution is shorter than the requested RMA4 simulation, the RMA2 results file will be re-wound and re-used again and again.

Example of hydrodynamic time control, TH card: TCORR=37, TEND=86 hrs.



Note: Re-running RMA2 with a smaller time step is more accurate than RMA4 internally interpolating the velocity field linearly between time steps.

Specifying The Pollutant Data

The specification of the sources of pollutants requires a detailed description of the time history for each loading source. It is up to the user to determine these data, and typically they will come from a number of sources. For purposes of model validation or short term operational control, the data should be collected from field measurements and observations. For other studies, such as long term planning activities, typical operation schedules and average discharges conditions are appropriate.

Specifying The Diffusion Coefficients

RMA4 requires two coefficients, one for the x-direction (DX card or DF card) and the other for the y-direction (DY card or DF card), which reflect the influence of turbulent behavior in the convective field. These coefficients are sometimes referred to as dispersion coefficient or diffusion coefficients. Although these coefficients have a somewhat artificial physical basis, they may be estimated from observed data. If no data exists, then the user may find these guidelines helpful.

- Use the automatic diffusion capability, Peclet method, within RMA4.
- Obtain guidelines from the pre-processor utility program **MAKE_EV_DF**
- Choose values known to be satisfactory in similar physical locations
- Choose values which are conservative in regard to the study's objectives
- Choose larger values as the detail in the velocity diminishes

For water quality studies, convection is usually the dominant transport process and modeling results may be relatively insensitive to the mixing coefficients. Therefore, it is often beneficial to run the model with a range of possible coefficients to determine how sensitive the predicted levels of quality are to these values.

There are two methods of assigning the diffusion coefficients, the modeler may directly specify them or activate the automatic calculation method.

Direct Specification

For the direct approach of specifying the diffusion coefficients, there must be diffusion coefficients for both the x- and y-direction assigned for each material type within the network. The only exceptions are for material types exceeding 900, which designate the control structures and junction elements. Therefore the direct specification requires either one DF card, or sets of the DX and DY cards for each material type.

If data is not available to help determine what are the appropriate starting values for the diffusion coefficients, a utility program called **MAKE_EV_DF** will calculate the average elemental size of each material type and create a table of recommended diffusion coefficients values for each material type in the mesh. The need for this utility program has been replaced by automatic methodologies discussed in the next section. For more information on **MAKE_EV_DF,** see "Obtaining Estimates for Diffusion Coefficients" on page 146.

The table below provides some representative ranges of diffusion coefficients.

Type of Problem	D, m^2/sec
Homogenous horizontal flow around an island	0.01-0.1
Homogenous horizontal flow at a confluence	0.03-0.01

Steady-State flow for thermal discharge to a slow moving river	0.02-1
Tidal flow in a marshy estuary	0.05-0.2
Slow flow through a shallow pond	0.0002- 0.001

Automatic Assignment by Peclet Number

The second way to assign the diffusion coefficient, D, is to allow the model to automatically adjust D after each time step, based upon a provided Peclet number, which is based upon the element size and calculated velocity within each element. The Peclet methodology is very similar to that of RMA2. The Peclet number is the nondimensional parameter developed by ratioing the advective terms to the diffusive terms in the governing equations.

Recall the formula for Peclet number (P) that was introduced in the RMA2 User's Guide, where P is recommended to be between 15 and 40.

Equation 2

$$P = \frac{\rho u dx}{E}$$

However, for RMA4, the Peclet number (P) takes the form

Equation 3

$$P = \frac{u dx}{D}$$

As can be easily seen, D_x (similarly for D_y) can be approximated as

Equation 4

$$D_x = \frac{E_{xx}}{\rho}$$

where:

Coefficient			English Units	Metric Units
ρ	=	fluid density	1.94 slugs/ft^3	998.46 kg/m^3
u	=	average elemental velocity calculated at the gauss points	ft/sec	m/sec
dx	=	length of element in streamwise direction	ft	m
E	=	Eddy Viscosity for RMA2	lb-sec/ft	Pascal-sec
D	=	Diffusion Coefficient for RMA4	ft^2/sec	m^2/sec

At present the Peclet method, is the only option for automatically calculating the diffusion coefficients.

If the diffusion coefficients were previously assigned by the direct method, mentioned in the previous heading, then the Peclet method will overrule these assignments. The Peclet capability is activated with the PE card, which specifies a global Peclet number to the entire mesh. The modeler my then choose to define a unique Peclet number to one or all material types with the PET card(s).

Assigning Initial Conditions

In the time-dependent mode of operation, the RMA4 model moves forward in time from some beginning, or initial condition. The concentrations specified at the starting time are called the **initial conditions**. The *calculated* concentrations at the beginning of a simulation are highly influenced by the initial conditions, but to a lesser degree as time progresses.

You may have access to extensive field measurements, or output from another model, to permit a good estimate of concentrations contours in the study domain at a particular time. There are two techniques available to assign an initial estimate of concentrations for the domain.

- Creating Distinct Initial Concentration
- Creating Gradual Initial Concentration
- Using a Hotstart as the Initial Condition

Creating Distinct Initial Concentrations

One practical straightforward approach for setting initial conditions is to start the simulation with an arbitrary global value for concentration and simply let the RMA4 model proceed with specified inputs until the state of the system is independent of the starting condition. For this technique, a rule of thumb for "how long is long enough" is to run the simulation for a period of approximately *twice* the time it takes for the velocity field to move across the area being simulated. After satisfying this criterion, the initial conditions should have little influence on the model's results. However, if you can initialize the model with realistic concentration contours, the best solution will be obtained in the least amount of computational time.

The global assignment may be the simplest method to create initial contours of concentration, however it may be improved by making use of the material types, element numbers, or node numbers to assign an initial condition. This may be achieved quickly with the IC card with the T option using element material types, if the material types were assigned with the intent of using them to apply the initial concentration values.

Creating Gradual Initial Concentrations

Another technique for obtaining initial contours of concentration is to let the RMA4 model calculate gradual concentrations starting from the contaminant source point.

The strategy is as follows:

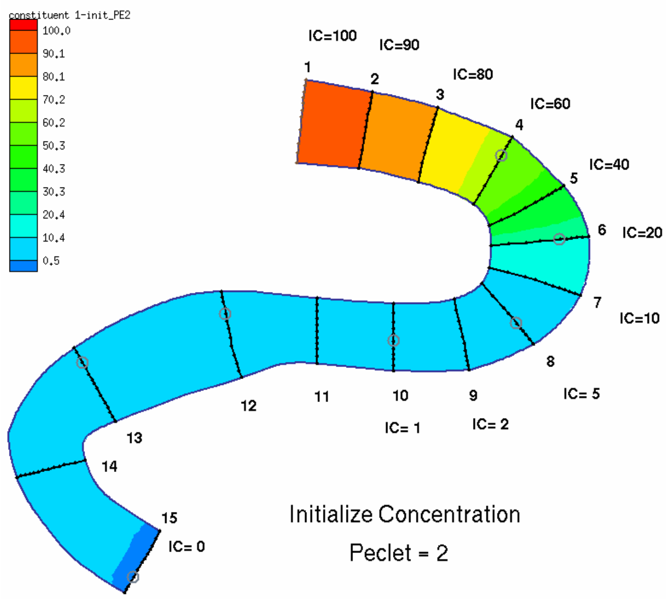
- Initialize the entire model domain with the IC card to permit a representative starting concentration.
- Strategically assign continuity check lines throughout the model domain.
- On the TC card, set the delta time step to a value of zero and the steady state flag to a value of one.
- Define a boundary condition with a negative continuity check line number at pertinent locations using the BC card with the L option.
- Run the RMA4 model for one time step and save both the solution and the Hotstart file.

- Examine the concentration contours of the RMA4 solution to determine if the continuity check lines are assigned and located appropriately. Some experimentation may be necessary.
- Once you are satisfied with the initial condition concentration patterns, then Hotstart RMA4 and proceed with the simulation.

Example:RMA4 Run Control for Gradual Initialization of Concentration.

Gradual Initialization	T3 Medora -RMA4 Init with GCL & PE-Card=2
Set I/O Files	SI 0
	CO geo r2 hoti abc sol hoto fp sp parm
	$L 10 20 0 0 31 32 33 34 40
	$M 4
Print control	TP 1 1 12 1 -1 0
Summary print nodes	TRN 7 55 88 122 156 208 296 424
	FT 17.0
	FQ 1 0
1 constituent, no decay	FQC 0
	GCL 1 1 3 5 7 9 11 -1
Define 15 geometry	GCL 2 13 15 18 21 24 27 -1
check lines	GCL 3 30 32 35 38 41 44 -1
(see figure)	GCL 4 47 49 52 55 58 61 -1
	GCL 5 64 66 69 72 75 78 -1
	GCL 6 81 83 86 89 92 95 -1
	GCL 7 98 100 103 106 109 112 -1
	GCL 8 115 117 120 123 126 129 -1
	GCL 9 132 134 137 140 143 146 -1
	GCL 10 149 151 154 157 160 163 -1
	GCL 11 166 168 171 174 177 180 -1
	GCL 12 203 205 208 211 214
	GCL 217 220 -1
	GCL 13 292 294 297 300 303
	GCL 306 309 312 -1
	GCL 14 359 361 364 367 370
	GCL 373 376 -1
	GCL 15 419 421 424 427 430
	GCL 433 436 -1
Set Init Conc=0	IC 1 0.0
Set Peclet=2	PE 1 2 1.00 1 1
Set DELT=0,	RE -1 0 .25
TTSWITCH=1	TC 0 **0** 1 0 **1**
	TH 0 0
	BCL **-1** 100 0 0
Specify concentration	BCL **-2** 90 0 0
BCL with the check	BCL **-3** 80 0 0
line marked as	BCL **-4** 60 0 0
NEGATIVE	BCL **-5** 40 0 0
	BCL **-6** 20 0 0
	BCL **-7** 10 0 0
	BCL **-8** 5 0 0
	BCL **-9** 2 0 0
	BCL **-10** 1 0 0
	BCL **-15** 0 0 0
	END
	STOP

Example of a RMA4 Initialization of Concentration



Using a Hotstart as the Initial Condition

Data on the IC card(s) are ignored when Hotstarting.

Any initial conditions supplied by the user on the IC card(s) are ignored when Hotstarting because they are superseded by the read from a Hotstart file. A hotstart is generated from a previous RMA4 simulation. Hotstarting is activated by specifying a positive number for the IHOTN and/or IHOTO variables on the $L card.

The Hotstart binary file is unique in that the hotstart output file for one simulation becomes the hotstart input file defining the initial conditions of the continuation run.

Creating A Hotstart File

Saving a hotstart file is simple. You need only to specify a positive value for IHOTO variable on the $L card. When you run RMA4 you will be required to assign a name for your Hotstart output file.

How to Resume A Simulation Using Hotstart

Using a previously generated RMA4 hotstart file as a startup initial condition is also very simple. Simply specify a positive value for IHOTN variable on the $L card. When you run RMA4 you will be required to assign a name for your Hotstart input file. It is optional at this point whether you wish to request to create another hotstart output. The quality of these initial conditions is completely dependent on the conditions of the RMA4 simulation that created the hotstart.

You must also specify the number of boundary condition sets (END cards) to skip in the RMA4 run control file (ISKIP_END variable on the BCC card). Obviously if the run is just being initiated from the beginning, this variable is zero.

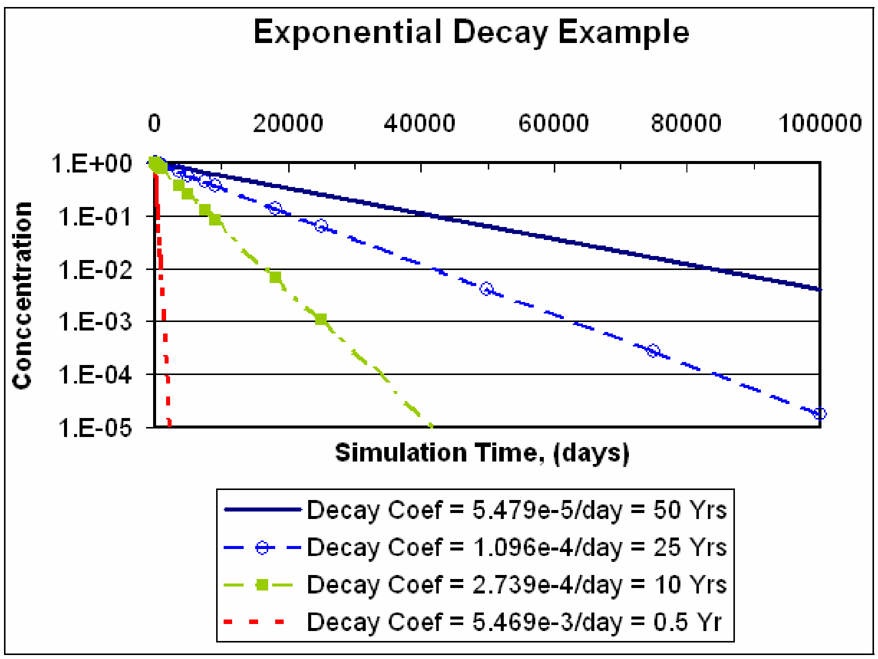
Specifying Fluid Qualities

The FQ and FQC cards are used to specify fluid qualities. The user has the option of transporting from 1 to 6 quality constituents with or without decay. Typically the default number of quality constituents is one, decay is zero, and the dissolved oxygen (DO) and biological oxygen demand calculations (BOD) are off.

Decay Coefficients

The model will treat pollutants either as conservative or non-conservative using a first order decay. The decay coefficient for each constituent is assigned on the FQC card in units of day^{-1} (see "FQC Card: Fluid Qualities - Decay Control" on page 98).

To run RMA4 in conservative form, set the decay coefficient to zero. For non-conservative simulations a positive decay coefficient is supplied for each active constituent being simulated. A small decay coefficient results in slow decay. For example, a small decay coefficient equivalent to 50-years or 18250 days, would equal 5.479e-5/days. As shown in the graphic, an initial concentration of one would be decreased to .0042 after 100,000 days. If a decay coefficient equivalent to 0.5 years or 5.469e-3/days were applied, this would cause the initial concentration of one to be decreased to .0042 after 1000 days. A larger decay coefficient causes rapid decay. For example, a larger decay coefficient of 1/day would cause the initial concentration of one to be decreased to .0067 after only 5 days.



The exponential decay equation follows.

Equation 5

$$\frac{C_i(t)}{C_i(t_o)} = e^{-[XKCOEF_i(t-t_o))]}$$

where

$XKCOEF_i$ = Decay Coefficient for constituent *i*

C(t) = concentration at present time t, for a given constituent *i*

$C(t_o)$ = concentration at previous time t_o, for a given constituent *i*

Dissolved Oxygen and Biological Oxygen Demand

Dissolved Oxygen (DO) and Biological Oxygen demand (BOD) are modeled as coupled constituents. In RMA4, this coupling is given as a simplified relationship: The BOD decay is given as a first order function of the BOD concentration, and the DO decay is given as a first order function of both the DO and the BOD concentration. The DO calculation also includes a term that simulates the entrainment of atmospheric oxygen, as a function of depth, water velocity, and water temperature.

DO/BOD Equations

The DO decay and production terms are given as follows:

Equation 6

$$\frac{\partial C_{DO}}{\partial t} = -K_{DO}C_{DO} - K_{BOD}C_{BOD} + K_{DOE}\left(CS_{DO} - C_{DO}\right)$$

The BOD decay term is given as follows:

Equation 7

$$\frac{\partial C_{BOD}}{\partial t} = -K_{BOD}\beta_{WT}C_{BOD}$$

where

K_{DO} = the DO decay coefficient @ 20°C (1/day)

C_{DO} = the DO concentration (mg/l)

K_{BOD} = the BOD decay coefficient @ 20°C (1/day)

C_{BOD} = the BOD concentration (mg/l)

K_{DOE} = the DO atmospheric reaeration coefficient (1/day)

CS_{DO} = the DO saturation concentration (mg/l)

β_{WT} = a correction factor for the water temperature

The values of K_{DO} and K_{BOD} are lumped parameters that include all decay effects except for atmospheric reaeration, which is handled with K_{DOE}. .

The values of β_{WT}, $CS_{DO,}$ and K_{DOE} are calculated in RMA4 using the following empirical relationships:

Equation 8

$$\beta_{WT} = (1.047)^{(20-T)}$$

$$CS_{DO} = 14.55 - 0.3822T + .005426T^2$$

$$K_{DOE} = K_{DOE.C} + K_{DOE.W}$$

$$K_{DOE.C} = 3.8966\frac{\sqrt{U}}{d^2}$$

$$K_{DOE.W} = 3.7843\, u_{*W} \text{ for } u_{*W} < 0.0207 \text{ m s}^{-1}$$

$$K_{DOE.W} = 15.8112\, u_{*W}^2 \text{ for } u_{*W} > 0.0207 \text{ m s}^{-1}$$

where

$K_{DOE.C}$ = the DO atmospheric reaeration coefficient from current (1/day)

$K_{DOE.W}$ = the DO atmospheric reaeration coefficient from wind (1/day)

u_{*W} = the wind shear velocity (m/s or ft/s)

$U_{W.10}$ = the wind speed at 10m elevation (m/s or ft/s)

T = the water temperature, as given on the FT card (°C)

U = the water velocity magnitude (ft/s or m/s as set by SI-Card)

d = the water depth (ft or m as set by SI-Card)

The wind reaeration coefficient is taken from Chu and Jirka (2003)

How to Activate DO/BOD calculations

The dissolved oxygen (DO) and biological oxygen demand (BOD) calculations are activated by FQ card settings. To make DO/BOD calculations with RMA4, set the number of constituents to two (NQAL=2), and toggle on the switch (IDOS=1) on the FQ card. This will establish the dissolved oxygen as constituent number 1, and biological oxygen demand as constituent number 2.

On the FQC card, set the values of the DO and BOD decay coefficients (K_{DO} and K_{BOD}). On this card, XKCOEF(1) = K_{DO}, and XKCOEF(2) = K_{BOD}.

Specifying Boundary Conditions

A concentration boundary condition should be applied at every potential inflow boundary location.

There are two main categories of boundary conditions for RMA4:

- Concentration Boundary Condition
- Mass Loading Boundary Condition

Concentration Boundary Condition

Concentration may be specified by node, or across a check line, for each of the constituents (1-6). The assigned value is held to its specified value for all portions of the simulation when the flow direction is into the model. If the flow direction is out of the model, the exit value is the calculated concentration at that location. The user may over ride the default, by forcing the exit concentration to be the specified value by flagging the continuity line as negative on the BC card.

Later, if the flow direction is once again directed into the model, the specified concentration is reactivated at the boundary. This strategy is not always satisfactory for a boundary flow reversal situation, because unrealistic sharp concentration gradients may result.

For the tidal boundary or any flow reversal boundary condition, the strategy of a boundary-mixing chamber will eliminate abrupt concentration gradients.

Boundary Mixing Chamber

Boundary mixing is optional and may be activated by assigning the variable IBCFT on the BC Card to 1. The mixing chamber factor (BCFCT) is a real value ranging between 0 and 1, which serves as a multiplier to the actual computed concentration. Where a 0 means to hold the boundary condition specification (100%) when flow comes into the model, and a 1 means that a series of 10 buffers (default value) will *remember* the calculated concentration at the previous 10 time steps, apply a mixing algorithm, and gradually feed the appropriate concentration to the model when flow comes into the model after an outflow.

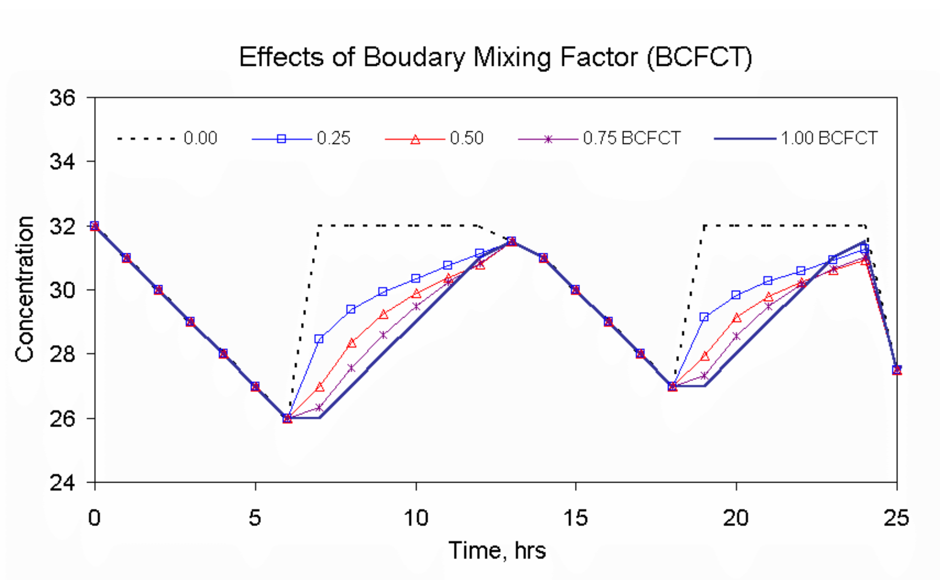
The boundary mixing chamber factor with a value greater than zero reduces the likelihood of a numerical shock resulting from sudden changes of concentration at the boundary within one time step, and is a pseudo successful means of specifying a boundary condition that is located too close to the upstream boundary condition to remain unaffected by it.

For example, consider this scenario of a simulation where there is fresh water specified at the upstream and salt water specified at the tidal boundary. Suppose the first few time steps have a boundary inflow of 32 parts per thousand, and then the flow reverses such that fresher waters begin to exit normal to the boundary condition. The use of boundary mixing chamber allows for the model to effectively remember the concentration of water that has left the model during the previous time steps. To accomplish this, each boundary node is allocated a series of mixing "chambers" into which the concentrations are stored.

As water leaves the model the current concentration of the water leaving the model at that node is stored in chamber 1 and the chamber concentrations from the previous time step are shifted (advected) to the next higher chamber number. Then the chamber-mixing factor (BCFACT) is used to perform some level of mixing between adjacent cambers at the end of each time step. The last mixing chamber (#10 by default) is where the actual user-specified boundary condition is applied. That chamber's concentration is never altered.

When the water turns and begins to flow back into the model then the chamber concentrations are shifted toward the next lower chamber number and the mixing factor again is applied.

A graphic showing the effects of the mixing chamber factor (BCFCT) is presented below. Even a small mixing factor (.25) greatly improves model performance.



NOTE: The number of buffers available is controlled by the PARAMETER variable MBB. If the default value of 10 is not satisfactory, this parameter may be adjusted prior to the compilation of the RMA4 program.

Mass Loading Boundary Condition

The BL card is used to specify mass loading of a pollutant. Mass loading, for each of the constituents (1-6), may be specified globally, or by individual element, or material type on the BL card.

The advantage of specifying a mass loading is that the user has control over the total mass that is input to the model.   If an internal point boundary condition were allowed the concentration is fixed at the specified value, but the effective total input to the model is impossible to calculate for all practical purposes. The total mass input for a point source would be:

Equation 9

$$M = \int_{t}^{t+\Delta t}\int_{\Omega} d(C_{ps} - C_n)\,d\Omega\,dt$$

where:

d	=	water depth
t	=	time
Δt	=	time step
M	=	total mass input into the model between time t and t+Δt, kg
C_{ps}	=	concentration solution with internal point source, kg/m^3
C_n	=	concentration solution without internal point source, kg/m^3
Ω	=	the model domain comprised of all elements that include the node where the point source is applied

In order to estimate the total load using the internal concentration specification two separate runs would have to be made and the integral above evaluated, which is nontrivial. Because of the nontrivial nature of a point source specification, the capability of a "BLN" input was removed with RMA4 version 4.5.

However, mass loading by element is a easier concept. If the BLE card is used the user specifies the total mass loading rate ***m*** in mass/sec (kg/sec in this example) for a given element. The model internally converts the specification into a concentration loading, **L**, by dividing this by the volume of the element (rather than node) into which the loading is applied.

Equation 10

$$L = \frac{m}{\int_{\Omega} d \;\; d\Omega} = \frac{m}{Volume}$$

After the conversion, the units of the loading are now in kg/m^3/sec. The total mass input into the model during a time step is then simply

Equation 11

$$M = \int_{t}^{t+\Delta t} m\; dt$$

If m is a constant then $M = m\,\Delta t$

If a boundary mass loading by material type (BLT card) option is requested, the user still inputs the mass loading rate ***m*** in mass/sec to be applied to <u>each</u> individual element that has the desired material type. The BLT option is simply a shortcut method of specifying numerous BLE specifications.

Rainfall / Evaporation Concentration Boundary Condition

Whenever the hydrodynamic field, used as input to the RMA4 simulation, had a rainfall and or evaporation criterion specified, then the user must supply the water quality transport model with the same rate that was used during the hydrodynamic model. Once the rate is provided, the RMA4 model will know to supply a zero concentration rate for a rainfall event and to extract the ambient concentration for an evaporation event. The RA card is used to specify an inflow rate (+ for rain, - for evaporation) to all elements involved in the rainfall/evaporation specification. Typically the rainfall/evaporation boundary condition is applied to long term salinity simulations.

The input units are *in/hr* or *cm/hr* as appropriate for the English or metric flag dictated on the SI card of the RMA4 run control file. Although the rainfall/evaporation rate input units are in terms of hours, the RMA4 model will internally scale the input rates to be in terms of seconds. For instance, if the SI card was set to one (metric), then the input rate of *cm/hr* will be converted to *m/sec* for consistency of RMA4 internal units. Similarly, if the SI card was set to zero (English), then the input rate of *in/hr* will be converted to *ft/sec*.

Interpretation Of Results

Overview

RMA4 is capable of generating more data in one simulation than a single person could absorb in a month. The key to understanding the results from the model lies in requesting to see key segments of information, such as: initial conditions, continuity checks, and mini-statistics. Most of these key segments are located in both the full results listing file and the screen output. At this time additional global and detailed information can be obtained from the graphical user interface SMS, or from a post-processing plotting utility program, such as Microsoft EXCEL.

To become an informed RMA4 user, you should be aware of what is included within the three types of RMA4 output:

- Full Results Listing
- Summary Results Listing
- RMA4 Solution

Understanding The Full Listing File

The Full Results Listing file is the most comprehensive analysis tool available in RMA4. Although it is not as complex as the full listing file created by RMA2, it contains some informative data. If a full listing of results is desired, use the $L card to specify a full listing file. The TP card is used to specify the detail and the types of information written in the file.

The structure of the full results listing file consists of some initial data, which is only printed once, and information, which is updated at every time step. The section marking the status of the velocity file and wet/dry updates mark the beginning of a new time step.

The full results listing file can be separated into these sections:

- Model Information
- Input Interpretation
- Element Connection Table
- Nodal Information and Initial Conditions
- Dynamic Simulation Progress And Statistics

The easiest way to locate these sections is to place the listing file into a text editor which has a search function and search for key words which are unique to each section. These key words are presented below within the description of each section.

Model Information

General model information appears first in the full results listing. This information is always included and cannot be switched off.

Version Number And Modification Date

The version number and last modification date are provided here. This information can be very important when requesting technical support.

To find this section in the full results listing file, search for the key word "**RMA4 VERSION**".

Here is a typical representation:

```
======   RMA4 VERSION 4.50   1D AND 2D CAPABILITY
======   LAST MODIFICATION DATE:  05-18-2001
```

Program Dimensions

The program array dimensions used for the simulation are listed here. If RMA4 cannot run using your data, you should verify that the program dimensions are large enough to accept your data. Generally, check the number of **nodes** and **elements**, and also the **front width**.

To find this section in the full results listing file, search for the key word "**EXECUTABLE**".

Here is a typical representation:

```
   RMA4 PROGRAM DIMENSIONS FOR THIS EXECUTABLE:
        MAX NODE                     =        33000
        MAX ELEMENT                  =        11000
        MAX FRONT WIDTH              =          550
        MAX CONTINUITY CHECKS        =          150
        MAX NUMBER OF NODES/GCL-LINE =          300
        MAX TIME STEPS               =         1500
        MAX BUFFER SIZE              =      2000000
        MAX SUMMARY PRINT BUFFER     =        24000
        MAX CONSTITUENTS             =            6
      ..IBatch Switch                =            2
```

Tip: The dimension labeled **MAX BUFFER SIZE** influences the number of temporary files which are written at each iteration. You may be able to improve the performance of RMA4 by increasing this dimension in the RMA4 source code. For details, see "Setting the Optimal File Buffer Size " on page 126.

File Names

The file names supplied to an interactive RMA4 simulation at startup are listed at the beginning of the full results listing file. The types of files, of course, depend upon the run options which were selected in the run control file.

To find this section, search for the key word "**FILE**".

Here is a typical representation for a hotstart run:

```
FILE INFORMATION:
-----------------
 RMA4 RUN CONTROL (bc) input [ascii]= project.r4bc
 Full Print listing file [ascii]    = project.r4fp
 RMA4 summary results file [ascii]  = project.r4sp
 GFGEN output geometry [binary]     = project.geobin
 RMA2 hydro solution [binary]       = project.r2sol
 RMA4 HOTSTART input [binary]       = project_ini.r4hot
 RMA4 HOTSTART output [binary]      = project.r4hot
 RMA4 solution output [binary]      = project.r4sol
```

Input Interpretation

An interpretation of the data cards used in the run control file appears in the full results listing after the Model Information section. This information is always included and cannot be switched off.

This section includes

- Geometry Input Summary
- Run Control Parameters

Geometry Input Summary

This small section of the full results listing file is to inform you as to what RMA4 has read from the GFGEN binary geometry file. The geometry title and the GFGEN version number are provided here, but most important are the network statistics which you use to verify that RMA4 has properly read your geometry data.

To find this section in the full results listing file, search for the key word "**NETWORK**".

Here is a typical representation:

```
 NETWORK INFORMATION FROM GFGEN [binary]:
 ----------------------------------------
     Title:M3dora Crossing Steady State, Curved sides
     Units from GFGEN banner           = ENGLISH
     Number of elements      (min/max) =       1     128
            Unused elem  =       0
            1D elements  =       0
            2D elements  =     128
               Triangles =       3
               Rectangles=     125
     Number of nodes         (min/max) =       1     437
     Number of materials     (min/max) =       1       3
     Number of neg or zero materials   =       0
     Number of Control Structures      =       0
            Structure types (min/max)  =       0       0
     Number of 1D Junctions
            Junction types (min/max)   =       0       0
     Reorder info for first element    =       1
     Reorder info for last element     =     123
```

Run Control Parameters

This section of the full results listing file contains several areas pertaining to the different aspects of run control. It includes information such as:

- The machine type identifier
- General control parameters
- Geometry and Hydrodynamic scales
- Logical unit number assignments for active files
- Time controls and run specifications
- Re-Solve Controls
- Special Print instructions
- Diffusion and decay coefficients
- A list of assigned geometry (continuity) check lines.
- Marsh Porosity Declaration

Machine Type Identifier

The value assigned to the machine type identifier is *only important if* there is insufficient internal memory (RAM) to put the entire matrix conversion process in memory. Otherwise stated, the proper machine identifier mush be specified to match the computer solving the simulation if buffer files are written.

To find this section in the full results listing file, search for these key words "**MACHINE TYPE IDENTIFIER**". To determine if buffer files were written, search for the text "**BUFFER BLOCKS WRITTEN**".

General Control Parameters

General information regarding the type of simulation to be conducted.

To find this section in the full results listing file, search for these key words of text "**GENERAL CONTROL PARAMETERS**".

Geometry and Hydrodynamic Scales

Reports the scaling values to be applied to both the input geometry and hydrodynamic files. These parameters were assigned by the GS and HS cards.

To find this section in the full results listing file, search for these key words of text "**SCALE FACTOR**".

Logical Unit Number Assignments For Active Files

This section reports the files which were activated on the $L card. The corresponding logical unit numbers associated with each file is reported.

To find this section in the full results listing file, search for the key words "**LOGICAL UNIT**".

Time Control and Run Specifications

The time controls for both the simulation (TC and TO cards) and for manipulation of the hydrodynamic file (TH card) are reported. Print controls (TP card) are also found in this section.

To find this section in the full results listing file, search for the key words "**TIME CONTROL**".

Re-Solve Control

The re-solve controls specified on the RE-card are reported in this section. The re-solve capability is a means to speed up the computational aspects of the simulation.

To find this section in the full results listing file, search for the key word "**RE-SOLVE**".

Print Request

The interpretation of the input provided on the TP, TR, and TQ cards are listed herein.

To find this section in the full results listing file, search for the key word "**PRINT**".

Diffusion and Decay Coefficients

The interpretation of all input provided on the DX, DY, DF, PE, and FQC cards are summarized in this section.

To find this section in the full results listing file, search for the key words "**DIFFUSION**" and/or "**DECAY**".

Here is a typical representation if the diffusion coefficients are applied directly through the run control:

```
DIFFUSION COEFFICIENTS ARE READ
-------------------------------
MATERIAL#      DIFF-X      DIFF-Y (length^2/sec)
     1         1.0000      1.0000
     2         1.0000      1.0000
```

Here is a typical representation if the diffusion coefficients are automatically generated with the Peclet methodology:

```
AUTO DIFFUSION BY PECLET (PE-Card):  IPEC Switch=1
----------------------------------
MATERIAL#   GPEC(imat)   DIFF-X   DIFF-Y (ratios)
     1          10         1.00     1.00
     2          10         1.00     1.00
```

Continuity Check Lines

A list of the geometry continuity check lines assigned with GCL cards are listed. Note that the mid side numbers have been added to the check lines for this listing.

To find this section in the full results listing file, search for the key words "**CONTINUITY CHECKS**".

Marsh Porosity Declaration

There is not a way to retrieve any information from the RMA4 results listing regarding the marsh porosity parameters, which may have been used during the hydrodynamic simulation.

To find this section in the full results listing file, search for the key words "**MARSH POROSITY**".

Element Connection Table

The element connection table contains detailed information on all the elements in the mesh. Generally the elemental data is read as a binary file, which was created by the Geometry File generation program, GFGEN. However, this is not always the case. Both data from GFGEN and those provided as input from the GE, GNN, GV, and GW cards are summarized in this section.

The element connection table lists all of the element numbers along with their associated node numbers, as well as the material type identifier. Additionally, this table will allow you to determine the nodes which make up an element, the material type, the elemental area, and elemental ordering sequence.

The element connection table echo can be illiminated by setting the IREPRT variable on the TP card to zero. It can be reduced in size by setting the IREPRT variable on the TP card to a large number. This causes a echo of every IREPRTth elemental value.

```
 ELEMENT CONNECTIONS AND MATERIAL NUMBERS   KITH=IREPRT increment= 50
 ----------------------------------------
 ELEMENT            NODES(counterclockwise)       TYPE ORDER AREA(sq-L)
 ------- ----------------------------------------- ---- ----  ----------
       1     1    12    13    14    15    16     3     2     3     1  3381589.5
      51   183   184   185   186   168   167   166   182     2    51  1882903.4
     101   320   319   317   346   347   348   344   343     1   101  1256870.3
     where Max Elem= 128
```

To find this section in the full results listing file, search for the key word "**CONNECTIONS**".

Note: An element with *negative* area indicates that its nodes are listed in reverse order. Be sure the nodes comprising the element are listed in a counter-clockwise order. Alternatively, elements with zero area may be a 1D control structure, 1D junction or an element whose material type is zero.

SMS 8.0

SMS Note: Safeguards within the SMS mesh generator help prevent you from making common network errors, such as, an element with a negative area, and other element connection errors. However, connection table errors are indeed fatal errors which must be corrected.

Nodal Information and Initial Conditions

Irregardless of whether the initial conditions reflect a coldstart or hotstart condition, they will be reported in the full results listing unless the IREPRT variable on the TP card is zero. The report can be reduced in size by setting the IREPRT variable to a large number. This will only report every IREPRTth nodal value.

The coordinates, elevation, velocity, and concentraton values of each node in the computational domain for each constituent (1-6) is provided.

To find this section in the full results listing file, search for the key words "**NODAL INFORMATION or INITIAL CONNECTIONS**".

```
 NODAL INFORMATION     KITH=IREPRT increment=     1   Max Node=     43
 -----------------
 NODE   (X-,Y-)COORDs 1D-WIDTH   SS1   SS2 STORAGEW  WSELV-CRIT OFF-CHAN-S
 -----  ------------  -------- ----  ---- --------  ----------  ---------
    1   1000.0  200.0    0.000   0.0   0.0    0.0       0.00      0.00
    2   1000.0  100.0    0.000   0.0   0.0    0.0       0.00      0.00
    3   1000.0    0.0    0.000   0.0   0.0    0.0       0.00      0.00
    4    937.5    0.0    0.000   0.0   0.0    0.0       0.00      0.00
    5    875.0    0.0    0.000   0.0   0.0    0.0       0.00      0.00
    etc.

 NODAL VELOCITIES AND INITIAL CONDITIONS (after scaling)
 -------------------------------------------------------
 NODE   WDEPTH     u-VEL    v-VEL   CONC(1) ... CONC(6)     WTEMP
          (L)      (L/s)    (L/s)   (mass/cu-L)            (DEG C)
-----   ------    ------   ------   -------------------    -------
   1    10.000    1.028    0.000     30.000 ...    0.0      15.0
   2    10.000    1.028    0.000     30.000 ...    0.0      15.0
   3    10.000    1.028    0.000     30.000 ...    0.0      15.0
   4    10.001    1.026    0.000     30.000 ...    0.0      15.0
   5    10.002    1.024    0.000     30.000 ...    0.0      15.0
  etc.
```

Dynamic Simulation Progress And Statistics

This section repeats for each computational time step.

This section contains information which shows the model transient input and output for each time step solved during the simulation.

This information is always included and cannot be switched off. Detailed information about each iteration is provided. This is valuable information you can use to help find the cause of problems.

These items are repeated for each time step.

- Velocity File Status
- Wetting and Drying Status
- Boundary Condition Inputs
- Number of Equations
- Buffer Blocks
- Front Width
- Constituent Mass and Mass Flux Checks
- Automatic Diffusion Parameter Statistics
- Negative Concentration Report
- Table of concentration results

Velocity File Status

When a RMA2 hydrodynamic file is used as the velocity field for the mesh, it is imperative that the appropriate hydrodynamic time step is used for the current transport time step. The controlling time parameters for the hydrodynamic file are provided on the TH card. This section provided a status of the RMA2 time read and the *corrected* time used for the RMA4 simulation.

Example of echo reflecting timing control for hydrodynamic file

```
RMA2 Model Flag/Version/Node/Elem=   120   435    22557    7362

--> RMA2 Velocity File read at time= 0.0  CORRECTED TIME= -25.0
--> RMA2 Velocity File read at time= 0.5  CORRECTED TIME= -24.5
--> RMA2 Velocity File read at time= 1.0  CORRECTED TIME= -24.0
     . . . skip . . .
--> RMA2 Velocity File read at time= 25.0 CORRECTED TIME=   0.0
--> RMA2 Velocity File read at time= 25.5 CORRECTED TIME=   0.5
--> Scaled the RMA2 hydro. RMA4 time= 0.50
    Scales=  0.3048  0.3048  0.3048
```

To find this section in the full results listing file, search for the text "**VELOCITY FILE**".

Wetting and Drying Status

This section of the full results listing will summarize any wetting and drying which is present on the hydrodynamic input for the current time step.

To find this section in the full results listing file, search for the text "**THE FOLLOWING ELEMENTS HAVE BEEN ELEMINATED**".

Note: When an element or node is removed from or added to the mesh, the total number of **SYSTEM EQUATIONS** will be changed from that of the previous time step.

Boundary Condition Inputs

This section of the full results listing file lists the boundary conditions that were specified for the current time step of the RMA4 simulation. Boundary conditions may be defined as a specified boundary concentration (BC card) or as boundary mass loading (BL card). The type of boundary condition is stated, then the associated parameters defining the boundary condition.

If you are simulating rainfall or evaporation, the concentration specification values will be automatically converted to be in the units appropriate for the RMA4 calculations. Verify that the input rainfall/evaporation rate was converted with the consistency of the unit designation of the SI-card. For example if the SI card was one, then the rainfall/evaporation units of cm/hr will be internally converted to m/sec.

This section may also contain information on any flow control structures that are defined for this simulation. Each control structure material type ($\geq$ 904) must have concentration mixing instructions provided in the run control file on the CS card.

Note: The "ACTIVE" descriptor for a node boundary condition means that the velocity vectors are flowing into the model domain and the specified BC will be applied. Conversely, the specified BC will not be applied if flow is out of the model domain. The boundary mixing option on the BC Card is a notable exception.

To find this section in the full results listing file, search for the key words "**`BOUNDARY CONDITION`**" and "**`RAINFALL`**".

Number Of Equations

The number of equations necessary to obtain a solution is reported here in the full results listing file. This number may change when wetting and drying are occurring. Remember that the maximum number of equations allowed is stated as **MAX NO. OF EQUATIONS** as described in "Program Dimensions" on page 48.

To find this section in the full results listing file, search for the key words "**`SYSTEM EQUATIONS`**".

Buffer Blocks

Before RMA4 can begin to solve the equations, it must build the solution matrix. If the value of **MAX BUFFER SIZE**, as described in "Program Dimensions" on page 48, is not large enough to allow RMA4 to store the matrix in memory, RMA4 must store the remainder of the matrix on disk as temporary files, or *buffer blocks*. The number of buffer blocks written is indicated at the beginning of each iteration.

To find this section in the full results listing file, search for the key words "**`BUFFER BLOCKS`**".

Tip: The dimension labeled **MAX BUFFER SIZE** influences the number of temporary files which are written at each iteration. You may be able to improve the performance of RMA4 by increasing this dimension in the RMA4 source code. For details, see "Setting the Optimal File Buffer Size " on page 126.

Front Width

The **MAXIMUM FRONT WIDTH** is listed immediately under the **BUFFER BLOCKS WRITTEN** section in the full results listing. The front width is indicated at the beginning of each iteration. Larger values of front width will increase computation time and may be indicative of a failure to properly reorder the mesh.

The RMA4 model running with one constituent will have a front width smaller than the RMA2 front width.

To find this section in the full results listing file, search for the key words "**`MAXIMUM FRONT WIDTH`**".

Example:

```
TOTAL NUMBER OF ACTIVE SYSTEM EQUATIONS =    426
BUFFER BLOCKS WRITTEN=          0
FINAL LQ SIZE        =       6860
MAXIMUM FRONT WIDTH  =         20
```

Constituent Mass and Mass Flux Checks

In this section you will find (1) the total mass of a constituent within the model domain, and (2) the amount of flow across each geometry continuity check line you have defined. The total mass flux for the active constituent is listed as **TOTAL (Mass)**, **X-FLUX**, **Y-FLUX**, and **PERCENT**. The percent field is always with respect to the *first* continuity check line.

To find this section in the full results listing file, search for the text "**`MASS IN NETWORK..`**".

Example: typical representation

TOTAL CONSTITUENT MASS IN NETWORK, TIME= 19.500, Nstep= 39
CONSTITUENT : 1
MASS (*units*) : 592284.64E+02

CONTINUITY CHECKS...TIME = 19.500 NSTEP= 39 CONSTITUENT #= 1
(Total MASS FLUX for the active constituents)

++	LINE	TOTAL(Mass)	X-Flux	Y-Flux	% of 1st Line
++	1	59999.99	59999.99	0.00	100.00
++	2	60026.36	60026.36	0.00	100.04

Tip: The "++" symbol on the continuity check lines were intentionally placed here to permit the "grep" unix utility to be used to extract information. For instance, if the information for the second check line was desired for the entire simulation, then the following command could be used:

grep "++ 2" full_results_lisiting > check_line2.text

Concentration Statistics

The statistics for minimum and maximum concentration computed during a given time step are provided in the full results listing file. They help to inform you as to how well the model is performing at any given time. These statistics provide locations where the minimum and maximum concentrations were determined. In situations where the model is unstable, these sections can help locate the problem.

```
ACTIVE NODAL CONCENTRATION STATUS FOR CONSTITUTENT#1:
(active nodes= 43) NSTEP=   39
     MIN Concentration = 29.2020 found at node= 1  Time=19.50
     MAX Concentration = 30.0000 found at node=40  Time=19.50
```

Tip: The "found at" keyword were intentionally placed to permit the "grep" unix utility to be used to extract information. For instance, to check for the extreme values of concentration for the entire simulation, then the following command could be used:

grep "found at"   full_results_lisiting   >   extreme_conc.text

Automatic Diffusion Statistics

The statistics for calculating automatic diffusion are written to the full results listing file, and to the terminal display, after every time step. They inform you as to how well the model is performing at any given time. These statistics provide locations where the minimum and maximum diffusion coefficients were determined. In situations where the model is unstable, these sections can help provide you with the location of the problem.

To find this section in the full results listing file, search for the text "**`STATISTICS`**".

Example: typical representation

```
AUTOMATIC PARAMETER STATISTICS FOR TIME STEP =   39
    ELEM  DIFF-MAX     ELEM  DIFF-MIN
       1    16.245        8    15.794
```

Negative Concentration Status

This diagnostic printout was added for the convenience of scanning the printout to determine if the simulation is having numerical difficulties. If a negative concentration was computed, (concentration values less than -1e-6), then a negative concentration warning message will occur.

Example: typical representation

```
--> Number of Negative Concentrations=  0 for Hr= 19.50 Step=39
```

To find this section in the full results listing file, search for the key words "**`Negative Concentration`**".

Tip: The unix uitility grep may be used to scan the print file for the negative concentration diagnostic.

grep –in "negative conc" full_results_lisiting   > negcon.text

Dynamic Simulation Nodal Results

This section of the full results listing file contains the latest nodal results of the simulation run. RMA4 will write the nodal results at the requested time step interval as specified on the TP card. There are two choices, the long form and the short form of the nodal results.

*A "Column of Data" in this context consists of the headings **NODE**, **U-VEL**, **V-VEL**, **DEEP**, and **C(1) through C(6)**.*

The long form nodal results are listed as ten columns of data, each containing node numbers, along with each node's *x* and *y* velocity components, depth, and concentration for each constituent, one through six. To conserve space, the long form of the table actually repeats the ten columns. The table is interpreted by reading down the first column of data to the end, then starting at the top of the second column of data and reading down to the end.

Long Form Example of the nodal results:

RMA4 dynamic rain test PECLET=10 // RAIN 12 in/hr

RESULTS AFTER 19.500 HRS OF SIMULATION...(T-STEP= 39 RMA2 Hr= 19.5000)

NODE	U-VEL	V-VEL	DEEP	C(1) … C(6)	NODE	U-VEL	V-VEL	DEEP	C(1) … C(6)
	(L/s)	(L/s)	(L)	(mass/cu-L)		(L/s)	(L/s)	(L)	(mass/cu-L)
1	1.03	0.00	10.0	29.202 … 0.00	23	1.01	0.00	10.0	29.539 … 0.00
2	1.03	0.00	10.0	29.202 … 0.00	24	1.01	0.00	10.0	29.640 … 0.00
3	1.03	0.00	10.0	29.202 … 0.00	25	1.01	0.00	10.0	29.691 … 0.00

etc.

The short form nodal results, eliminate all hydrodynamic information and report only concentrations the constituent(s) modeled. In the following example, the last value listed (highlighted) is the concentration for node number 43 at simulation time 19.5 hrs.

Short Form Example of the nodal results:

RMA4 dynamic rain test PECLET=10 // RAIN 12 in/hr

RESULTS AFTER 19.500 HRS OF SIMULATION...(T-STEP= 39 RMA2 Hr= 19.5000)

CONCENTRATIONS FOR CONSTITUENT 1

	0	10	20	30	40
	--------	--------	--------	--------	--------
1	29.202	29.389	29.589	29.793	30.000
2	29.202	29.389	29.589	29.793	30.000
3	29.202	29.338	29.539	29.742	29.948
4	29.237	29.438	29.640	29.845	
5	29.291	29.488	29.691	29.896	
6	29.291	29.488	29.691	29.896	
7	29.291	29.488	29.691	29.896	
8	29.237	29.438	29.640	29.845	
9	29.338	29.539	29.742	29.948	
10	29.389	29.589	29.793	30.000	

To find this section in the full results listing file, search for the key words "**RESULTS AFTER**".

Understanding The Summary Listing File

The summary listing is simply a brief synopsis of information for any nodes which you specify on TRN cards. For each node, the summary listing file will provide a table with simulation time step hour and computed concentration for each requested active constituent. At the bottom of this file is a summary of the velocity field at these nodes.

The nodes for which data is requested are provided on TRN cards. A TRN card and the corresponding results for a sample problem are shown below.

Summary Listing Example:

Given a run control file with a summary print request TRN card of the form:
TRN 4 34 63 123 151 -1

```
RMA4 FE TABS-MD MODEL FOR QUALITY IN THE HORIZ PLANE
Madora - RMA4 run control - HOTSTART thru hour 24.0

RESULTS AFTER  24.0 HRS OF SIMULATION...(T-STEP= 24)

  CONCENTRATION SUMMARIES FOR CONSTITUENT       1
    TIME-NODE          4        34        63      123      151
      1.00      100.000    65.485     0.218    0.000    0.000
      2.00      100.000    84.232     1.085    0.001    0.000
      3.00      100.000    93.909     3.984    0.006    0.000
      4.00      100.000    97.921    10.760    0.026    0.001
      5.00      100.000    99.310    22.521    0.101    0.006
     . . .
     10.00      100.000    99.886    88.593   10.247    1.696
     . . .
     15.00      100.000   100.341    99.645   56.504   28.269
     . . .
     20.00      100.000   100.438   101.977   85.263   71.803
     21.00      100.000   100.438   102.264   88.453   77.330
     22.00      100.000   100.431   102.517   91.155   81.870
     23.00      100.000   100.415   102.732   93.440   85.629
     24.00      100.000   100.387   102.900   95.361   88.766

  VELOCITY SUMMARIES FOR NODE       4
           TIME      X-VEL      Y-VEL      DEPTH
           1.00     0.6700    -0.0762    37.1307
           etc
```

The summary file can be imported into a spread sheet windows application program.

Interpreting The Solution

It is much easier to interpret the solution file with a graphical user interface, such as SMS, than to analyze a printout or stare at x-y plots at individual nodes. Since the solution file is written in binary form and cannot be readily moved across computer platforms, it is very convenient to have both RMA4 and the graphical user interface resident on the same platform. However utility programs, such as BIN2ASC and ASC2BIN, are available to assist when you need to move solution files across computer platforms. For details, see "Moving RMA4 Binary Solution Across Platforms" on page 145.

The primary solution file contains the computed concentrations for each node in the mesh for all active constituents.

It is advisable to examine concentration contours. It is also helpful to have the hydrodynamic file available to check plume patterns with the flow patterns.

The following checklist of items are suggested.

- Are there signs of irregular or oscillating concentration magnitudes?
- Are there any two delta-X instabilities present?
- Are the contours reasonable for the velocity vector patterns?
- Are there signs of irregular concentrations due to wetting and drying?

- Are the inflow boundary conditions holding the specifications?
- Are boundary buffering adjustments needed along reversal boundary conditions?
- Are the results influenced by the initial conditions?
- Have the results reached equilibrium?

Verifying The Simulation

Verification: A Process

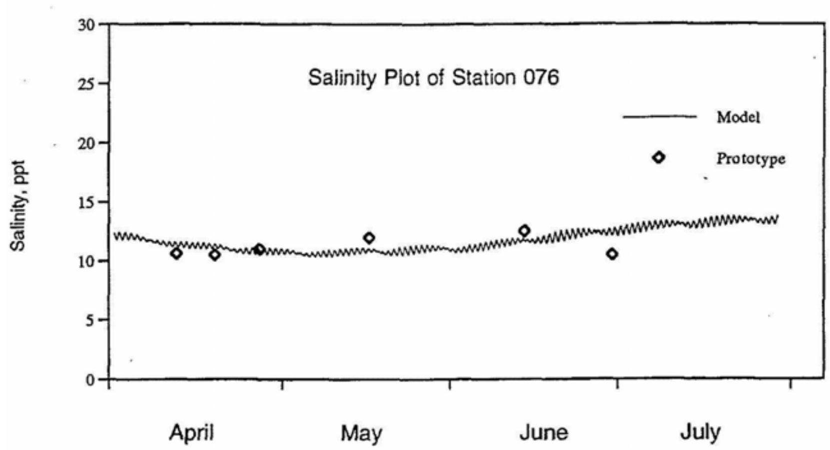
Verification of any numerical model is a process of run, inspect, compare to known field data, document the outcome, make adjustments, and re-run.

Compare to Field Data

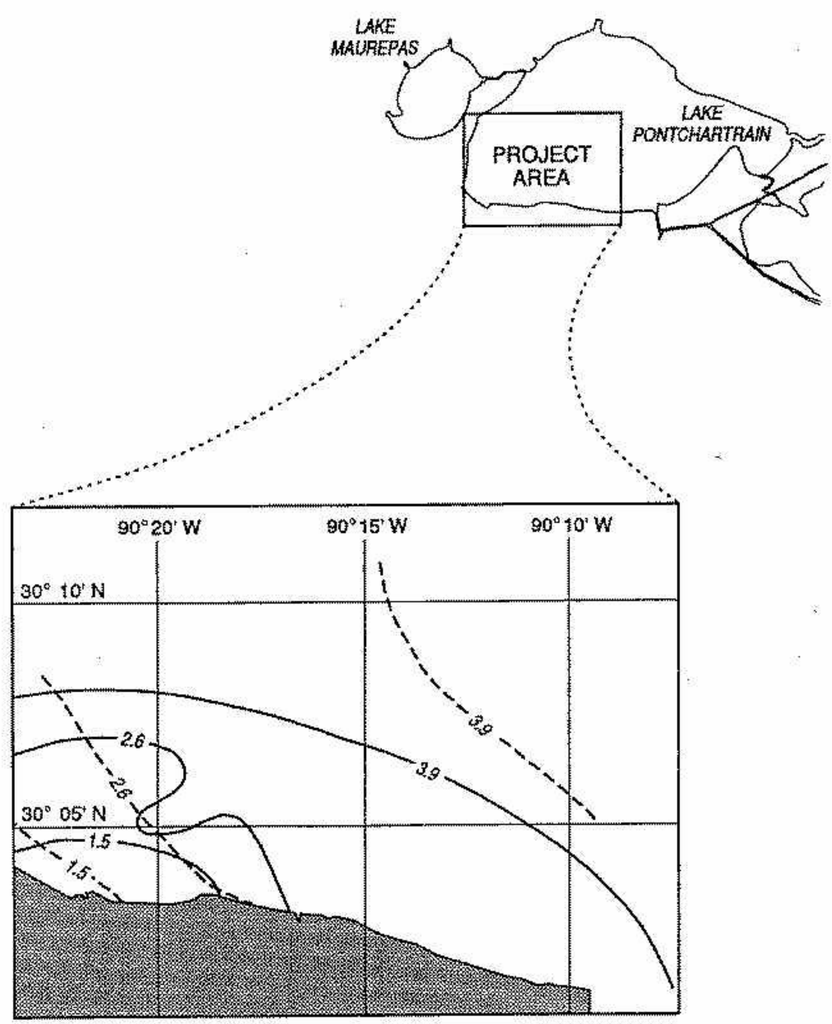
Direct Approach

The direct approach to verification is to have collected a synoptic time series of concentration data at multiple field stations and compare each station to the model simulations results. It is imperative to have the RMA2 and RMA4 simulations mimicking the same event that was present in the field during the data collection period.

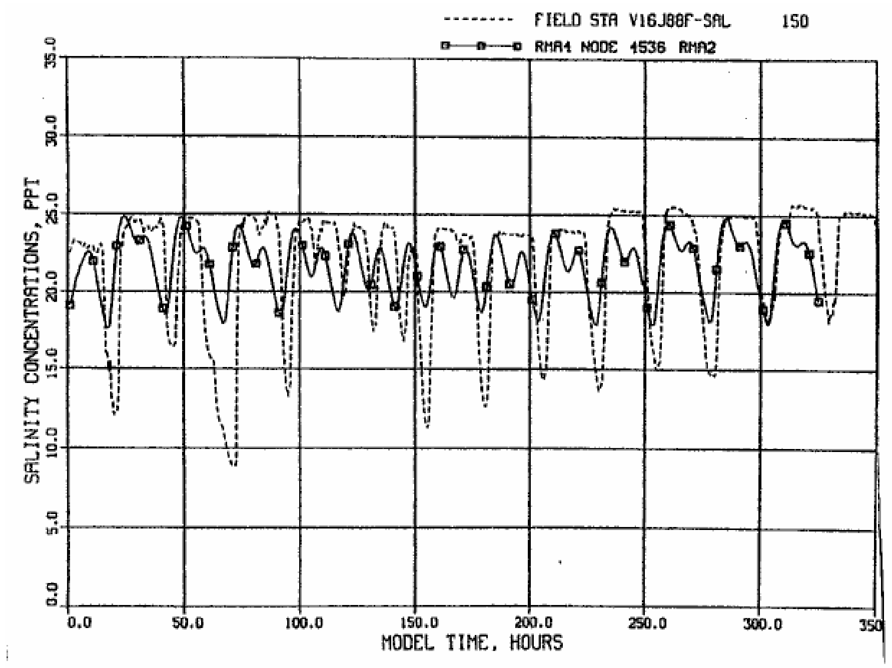
Several examples of comparison plots of RMA4 model results with field data were extracted from the “Salinity Changes in Pontchartrain Basin Estuary Resulting from Bonnet Carre’ Freshwater Diversion”, Technical Report CHL-97-2, by McAnally and Berger. Salinity grab samples were compared to RMA4 results for the Biloxi Marshes region.



The goal of the Bonnet Carre' study was to use RMA4 to predict monthly average salinity. The statistic, mean absolute error (MAE) , was calculated and found to be less than 2.5 ppt between model versus prototype. The figure shows model versus prototype isohalines of salinity.



Another example comparison plot of the RMA4 model results with field data was generated in 1989 for the Grand and White Lakes project.



Present day salinity studies typically use the TABS-MDS (previously referred to as WES RMA10), three dimensional model, because that model couples the hydrodynamics and salinity transport. The TABS-MDS model provides a more accurate simulation of the density affects associated with salinity.

Indirect Approach

Need examples

Advanced Techniques

Long Term Simulations

There are several issues and obstacles to bear when a long term simulation is considered. Some of these issues are:

- Residence Time
- Computational Requirements

Residence Time

One of the primary issues to be addressed when considering performing a long term simulation is "*how long is long enough*". In other words, you must know the residence time ($T_{Residence}$). Residence time is the time required for a constituent particle to be removed from a given water body. This removal can be a consequence of hydraulic flushing and/or decay (for non-conservative constituents). Furthermore the residence time should be defined relative to the study objective. For example, typically a complex system would have a residence time of a month or more, but if the study objective was concerned about a local pier slip, then the residence time would be in units of days or weeks.

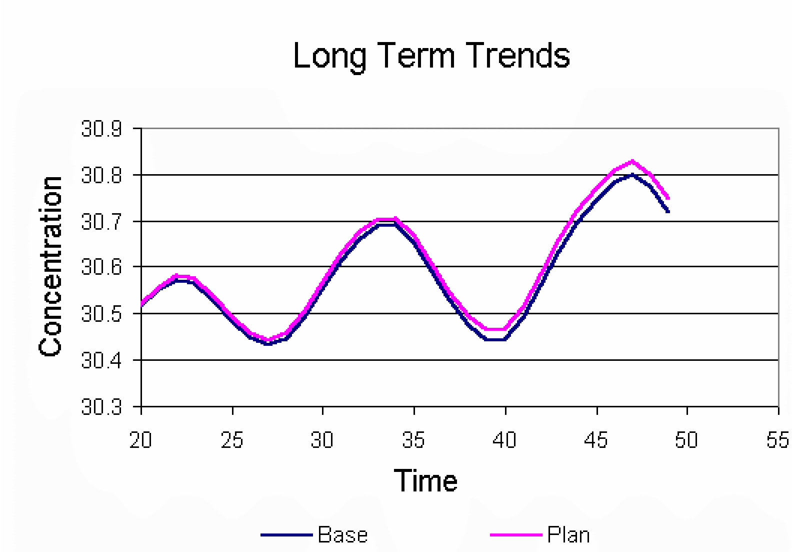
Once the residence time has been estimated, it is advisable to run the RMA4 model for a multiple (N) of the residence time matching the study objective.

Equation 12

$$\text{RMA4 Simulation Time} = N * T_{Residence}$$

We know that N should never be less than one, but the actual value is debatable. The appropriate value of 'N' probably lies between a 2 and 6. For instance, the residence time multiplier of N=3 for the Newark Bay, New Jersey, project was insufficient.

One way to tell if N is sufficient is to plot the long term concentrations between the base condition and the channel deepening plan condition. It is particularly helpful to examine the differences to determine if they are still growing at the end of the simulation. This non stabilized difference suggests a larger residence time multiplier is necessary.



This example plot illustrates two problems. First there is no sign of equilibrium. Secondly the differences between base and plan get larger with time.

Computational Requirements

The obvious issues related to long-term simulations are computer related. Estimating the amount of memory, disk space, and archival space required not only to run the model long term, but also to process the data and analysis it satisfactorily is essential.

Ideally it is desirable for the problem to fit entirely in memory so that no auxiliary buffering to disk will cause excessive input/output (I/O) and slow the computational speed. For small to medium projects, the memory requirements to accomplish this are very affordable.

For more discussion on this topic, see the section entitled "Performance Enhancements" on page 125.

Speeding Up The Computation

Recently, the computational hardware speed has been approximately doubling every few years. It is obvious to say that purchasing an up-to-date computer to meet your needs is a given. But with the new hardware come dreams and aspirations that grow with the technology of the times. Consequently there is an ongoing challenge to speed up the computational turn around time. There are several ways to help speed up the RMA4 computation:

- Plan for Optimal Performance
- Eliminate Unnecessary I/O
- Solve the Problem in Memory
- Save the Global Matrix and Re-Solve

Plan for Optimal Performance

By the time the long-term modeling phase has begun, there should have been some experimentation and testing conducted to make sure that you have planned for optimal performance of the model. A check list would include:

- The computer equipment is sufficient
- The geometry input follows good modeling rules and the exterior boundary is slope fixed
- The hydrodynamic input field was satisfactorily verified and is free of irregularities
- The sensitivity of RMA4 input parameters have been determined for your problem
- The output file size has been projected.

Eliminate Unnecessary I/O

All forms of unnecessary input/output (I/O) should be avoided. I/O is typically associated with low band width technology and will drastically effect the computational speed. The types of I/O which are associated with RMA4 that may be reduced or avoided will be discussed.

Once you have eliminated the diagnostic phase of modeling and begin the production phase of modeling, you can eliminate or drastically reduce unnecessary I/O. The most obvious choices are the ASCII writes to the full results listing file, and debug diagnostic traces to the "screen" file ($L card and TP card).

Another consideration is to not save the binary solution for the spin-up phase of the simulation. (TO card). One suggestion is to use the summary print feature to monitor key locations during the spin-up phase.

Other I/O issues involve solving the problem in memory, optimally choosing the size of the buffer dimension, and re-solve. These are complex issues and will be discussed in individual sections.

Solve the Problem in Memory

If the global matrix being solved fits entirely in internal memory, then no unnecessary input/output (I/O) will be required that will cause a slow down in the computational speed.

See the section entitled "Setting the Optimal File Buffer Size" on page 126.

Save the Global Matrix and Re-Solve

The re-solve technique is completely optional. The re-solve capability was added to permit speed up of long term RMA4 simulations. The re-solve capability is of value whenever the hydrodynamics are either steady state or have a repeatable cycle for the duration of the RMA4 simulation. In addition, the repeating tidal cycle must be uniformly divided into an integer number of time steps for proper hydrodynamic forcing.

The basic idea behind re-solve is to save the left hand side of the system of equations to a file for all time steps that define a cycle. Typically these files are named "Q001.00**n**", where **n** is an integer starting with 1 and sequentially increments to the total number of time steps that define the repeatable hydrodynamic cycle. (The specific file name is dependent on the $M card, machine identifier.) When the hydrodynamic cycle begins to repeat, the appropriate saved file for this time step is read. During the re-solution process the right hand side of the system of equations is formed then solved. For long simulations, where *many* cycles are repeated, the computational speedup is significant. The longer you run, the greater the speedup.

It is worthy to note that the re-solve technique has significant I/O overhead while saving the global matrix for an entire cycle. In addition, there are some "scratch" velocity files saved during the re-solve technique. Therefore, unless many cycles, key word *many*, are simulated, then the re-solve technique will not produce a noticeable speed up and may indeed be slower than the traditional approach.

There are some disadvantages to the re-solve. Extreme care and bookkeeping are required to set up the run control file to properly use re-solve. And there must be plenty of spare disk space allocation to save the large re-solve files.

The size of one re-solve file is equal to the number of buffer writes per time step times the size of one buffer file. There will be one re-solve file for each time step within a cycle. It is important to note that the re-solve files are not automatically deleted if the run prematurely terminates. The user may have to remove these files after a simulation to free up storage.

It is recommended to save the summary results file for at least several nodes to double check for reasonable concentration and that the velocity/hydrodynamic field cycled correctly during the re-solution. An error in the re-solve run control will cause an abrupt oscillation, negative concentration, and/or excessively unreasonable results. Such an error is usually very obvious and may be quickly found by examining the summary results file.

The following tables are provided to show the re-solve input control required in the RMA4 run control file and the post checking for repeatability that is recommended.

Example of a RMA4 Run Control using Re-Solve

	T1 CRANET ISL EFFLUENT MIXING ZONE ANALYSIS
	T2 RMA4 RUN OF UNIT CONCENTRATION DILUTION
Titles	T3 RMA4 Cranet Isl. RMA4v450 Resolve 7-10-2000.
	PE
I/O	$L 10 20 0 0 31 32 33 34 1
Machine ID	$M 4
Geom Scales	GS 0.3048 0.3048
Hydro Scales	HS 0.3048 0.3048 0.3048
Check Lines	GCL 1 1 7 12 17 22 27 32 37 42 -1
	GCL 2 22203 -1
	GCL 3 22127 -1
There are a	GCL 4 21833 21830 21827 21824 21822 -1
total of	GCL 5 18843 18840 18837 18835 -1
11 Check	GCL 6 12360 12357 12354 12351 12349 -1
Line	GCL 7 11880 11877 11874 11871 11869 -1
	GCL 8 11866 11863 11860 11857 11855 -1
	GCL 9 11527 11524 11521 11518 11516 -1
	GCL 10 11317 11314 11311 11308 11306 -1
	GCL 11 8256 8253 8250 8247 8245 -1
Summary Prnt	TRN 12 5571 5846 5552 6409 -1
Fluid	FQ 1 0
Quality	FQC 0.
	TP 500 0 25 1 -1 0
Trace Print	TC 0.0 0.5 101 50. 1
Time Control	COM RMA2 OUTFALL ON AT HR 25., OFF AT HR 50.
	TH 25. 50.0
RMA2 Time	FT 17.0
Temperature	IC 1 0.
Init Cond	DF 1 20.00 20.00
	DF 2 15.00 15.00
Diffussion	DF 3 10.00 10.00
Coef	DF 4 10.00 10.00
Direct	DF 5 20.00 20.00
Assignment	DF 6 5.00 5.00
imats 1-8	DF 7 10.00 10.00
	DF 8 20.00 20.00
Activate	PE 1 12 0.05 1.0 1.0
Peclet	PET 1 10 0.05 1.0 1.0
	PET 2 12 0.05 1.0 1.0
Assign	PET 3 10 0.05 1.0 1.0
Peclet	PET 4 10 0.05 1.0 1.0
by IMAT #	PET 5 7 0.05 1.0 1.0
	PET 6 4 0.05 1.0 1.0
	PET 7 12 0.05 1.0 1.0
	BCL 5 0.00 1 0.25
Define BC by	BCL 4 0.00 1 0.25
Line #,	BCL 2 0.00 1 0.25
using:	BCL 3 0.00 1 0.25
BC Mixing	BCL 10 1.00 1 0.25
Chamber	BCL 11 1.00 1 0.25
Technique	BCL 1 0.00 1 0.25
(BCFCT=.25)	<<< continued on next page >>>

	<<< RMA4 re-solve example continued >>>
Start Saving Global Matrix For Re-Solve	RE 1 0 0.50 END simulation hr= 0.0 ts=0 RE 1 0 0.50 END simulation hr= 0.5 ts=1 RE 1 0 0.50
Continue to Save for all 50 time steps that compose the repeating tidal cycle	END simulation hr= 1.0 ts=2 <<<skip for readability >>> RE 1 0 0.50 END simulation hr= 24.5 ts=49 RE 1 0 0.50 END simulation hr= 25.0 ts=50 This is the last save RE 0 1 END simulation hr= 25.5 use=1 ts=51
Start Re- Using. Requesting the correct Re- Solve file #	RE 0 2 END simulation hr= 26.0 use=2 ts=52 RE 0 3 END simulation hr= 26.5 use=3 ts=53 <<< skip for readability >>> END simulation hr= 49.0 use=48 ts=98 RE 0 49 END simulation hr= 49.5 use=49 ts=99 RE 0 50
Continue Re-Solving	END simulation hr= 50.0 use=50 ts=100 STOP

Example of a using a RMA4 summary file to check the Re-Solve strategy

TIME	Node=###	X-VEL	Y-VEL	DEPTH
0.5	0.000	-0.037	0.070	18.394
1	0.003	-0.040	0.076	18.494
1.5	0.008	-0.041	0.078	18.580
2	0.006	-0.039	0.075	18.648
2.5	0.003	-0.036	0.068	18.694
3	0.002	-0.030	0.058	18.714
23	0.089	0.006	-0.014	17.928
23.5	0.102	-0.004	0.005	17.996
24	0.117	-0.014	0.023	18.082
24.5	0.127	-0.023	0.041	18.182
25	0.080	-0.031	0.057	18.288
25.5	0.035	-0.037	0.070	18.394
26	0.023	-0.040	0.076	18.494
26.5	0.021	-0.041	0.078	18.580
27	0.014	-0.039	0.075	18.648
27.5	0.009	-0.036	0.068	18.694
28	0.007	-0.030	0.058	18.714
49	0.121	-0.014	0.023	18.082
49.5	0.131	-0.023	0.041	18.182
50	0.083	-0.031	0.057	18.288

The shaded rows show that the hydrodynamic file is correctly being reused every 25 hours.

Check for Conservation of Mass

RMA4 (version 4.5 or higher) provides a mass check summary. This option is activated with the MC card. One or more continuity check lines must be defined to make this option a viable tool. The traditional constituent mass flux checks, previously discussed in "Constituent Mass and Mass Flux Checks" on page 57 provide the mass flux for constituent 1 across each check line.

The mass conservation summary defines the flux check as the total inflow and total outflow as the contribution of a sum of one or more check lines. For instance, the MCI card may be used to list all check lines which define inflow into the model, and likewise the MCO card defines the outflow check lines. In addition, the total mass in versus the total mass out, and percent of gain/loss are reported. If the mass check option is active, the summary table is found in the full results listing after all simulation time steps have been completed.

Mass Conservation Summary Table

In the first example, there was only 1 check line defining the inflow, and 1 check line defining outflow. A mass check was accomplished by initializing all concentrations to zero, providing a steady state flow field, and introducing a contaminant pulse into the system.

The mass conservation summary table for example 1 shows the RMA4 results for a representative number of time steps. The first four columns of the table can be thought of as specified by the user. The last two columns are calculated within RMA4. At hour 4, the peak of the contaminant pulse, the total flux across continuity check line 1, is 500,000 mass-L^3/sec. This is calculated with the knowledge that check line #1 is the location of a 50,000 cfs inflow specification and a boundary concentration specification of 10 mg/l. By hour 4, non of the contaminant has passed the outflow check line, so the total flux out is zero. Therefore the fouth column of mass *in* minus mass *out* is simply the flux in (500,000) times the number of seconds representative of a time step (1800 sec), 9.0e+8 mass-L^3. The sum of the computed elemental mass contained within all elements in the mesh for a given time step are presented in column 5. The last column simply shows the mass change from the present and previous time step. Contaminant begins to cross the outflow check line at hour 15. All but a trace of the contaminant has exited the outflow check line by the end of hour 100. The mass check shows that there is a 0.12% error in mass conservation.

Mass Example 1: MASS CONSERVATION CHECK FOR CONSTITUENT= 1
Using 1 line(s) as the INFLOW and
1 line(s) as the OUTFLOW.

Time Step	Total Flux In (mass-L^3/s)	Total Flux Out (mass-L^3/s)	Mass (in–out) (mass-L^3)	Total Mass $\sum Elements$	Mass Diff (n)-(n-1)
0.5	390.0	0	7.0200E+05	1.02847E+06	0.0000E+00
1.0	780.0	0	1.4040E+06	2.49284E+06	1.4644E+07
1.5	1560.0	0	2.8080E+06	5.44742E+06	2.9546E+07
2.0	3125.0	0	5.6250E+06	1.13744E+07	5.9269E+07
2.5	6250.0	0	1.1250E+07	2.32270E+07	1.1853E+08
3.0	12500.0	0	2.2500E+07	4.69324E+07	2.3705E+08
3.5	250000.0	0	4.5000E+08	5.61955E+08	5.1502E+09
4.0	499999.9	0	9.0000E+08	1.49415E+09	9.3219E+09
4.5	250000.0	0	4.5000E+07	1.82750E+09	3.3335E+09
5.0	12500.0	0	2.2500E+07	1.81555E+09	-1.1949E+08
5.5	6250.0	O	1.1250E+07	1.87358E+09	5.8032E+08
6.0	3125.0	0	5.6250E+06	1.89013E+09	1.6547E+08
6.5	1560.0	0	2.8080E+06	1.89279E+09	2.6643E+07
7.0	780.0	0	1.40400E+06	1.89185E+09	-9.4157E+06
7.5	390.0	0	7.02000E+05	1.89005E+09	-1.7984E+07
15.0	0	0.3	-4.78127E+02	1.89009E+09	-2.2694E+06
30.0	0	54969.7	-9.89455E+07	7.70073E+08	-1.0063E+09

40.0	0	2211.2	-3.98015E+06	1.39227E+07	-4.3500E+07
50.0	0	3.1	-5.57832E+03	-1.19299E+04	-6.4337E+04
80.0	0	0	-2.36836E+00	8.83725E+00	-2.5800E+01
100.	0	0	1.46405E-03	-6.40707E-03	0

Summary	Total Mass In	Total Mass Out	Diff (in-out)	% Gain(+)/Loss(-)
	1.04921E+06	1.04934E+06	-1.29875E+02	+0.12%

Mass Conservation Sensitivity Test

Using the same simulation tactics as described above, sensitivity of the shape of the input contaminant pulse, diffusion, and time step size were tested. The shorthand notation (rise+max+fall) describing the input contaminant pulse represents the number of points for the rise, max, and fall of the inflow concentration boundary condition specifications. For instance, a 5+1+5 means that 5 time steps of increasingly larger (* 2) input concentration values were specified; ie: 0.3125, 0.625, 1.25, 2.5, 5.0. The peak value of 10 was assigned for one time step, followed by 5 time steps of decreasing values (* ½) of 5.0, 2.5, 1.25, 0.625, and 0.3125. The results indicate that mass conservation improves if the input pulse is gradual. The most benificial is a gradual rise, as apposed to a gradual fall. There was no sensitivity of either the diffusion coefficient or decreased time step size.

Mass Example 1 Continued: MASS CONSERVATION CHECK,SENSITIVITY TEST

TEST	Discription Input Contaminant Rise+Max+Fall	Max – Min Diffusion Coefficient	Worst Negative Concentration	Time Step Size (seconds)	Mass Gain(+) or Loss(-)
1	0+1+0 Point	70 – 13	-0.4685E+00	1800	333.33%
2	0+2+0 Point	70 – 13	-0.4685E+00	1800	166.67%
3	0+8+0 Point	70 – 13	-0.4685e+00	1800	41.67%
4a	1+1+1 Point	70 - 13	-0.3438E+00	1800	83.33%
4b	1+1+1 Point	703 – 126	-0.2589E-08	1800	83.33%
4c	1+1+1 Point	70 - 13	-0.3438E+00	900	83.33%
5	2+1+2 Point	70 – 13	-0.1719E+00	1800	33.33%
6	3+1+3 Point	70 – 13	-0.8597E-01	1800	15.15%
7	4+1+4 Point	70 – 13	-0.4290E-01	1800	7.25%
8	5+1+5 Point	70 - 13	-0.2149E-01	1800	3.55%
9	6+1+6 Point	70 – 13	-0.1074E-01	1800	1.75%
10	7+1+7 Point	70 - 13	-0.5373E-02	1800	0.87%
11	7+1+0 Point	70 - 13	-0.9195E-02	1800	1.31%
12	0+1+7 Point	70 – 13	-0.4685E+00	1800	167.32%

Run Control

Format for Run Control Data

RMA4 obtains its run control data from a set of input data cards provided in run control files.

The RMA4 data cards conform to what is known as HEC format. Each card has 80 columns of characters in which to hold its data. The first three columns are reserved for the card name, so there are actually 77 columns in which to hold data.

A data card occupies a single line in the input file. The card line is divided into data fields, of which the first contains the card name, and is designated as field 0. Field 1 begins the actual data for the card. RMA4 uses a free field format for data card input. Each item of data constitutes a field. There can be as many fields on a card as there is room within the 80 columns of characters.

Note: If more than one card modifies a variable, the last card rules.

Summary Of Run Control Data Cards

Although most of the data cards are independent of order, there are indeed some dependencies.

The following table lists all the data cards available in RMA4, and whether or not the card is required. The order of the list is the suggested order in which the cards should appear in the RMA4 run control file.

Card	Descriptive Content	Required
T1-T2	Title cards	No
T3	Title card	Yes
$L	Input/output file numbers	Yes
$M	Machine type identifier	Yes
SI	System International unit indicator	No
CA	Special Calculation Variable (ALPHA)	No

Card	Descriptive Content	Required
CO	Comments (anywhere except the first line)	No
CS	Control Structure Mixing	No
DF	Full Diffusion Coefficient (Dx and Dy)	+
DX	Diffusion Coefficient in X-Plane	+
DY	Diffusion Coefficient in Y-Plane	+
FT	Fluid temperature	Yes
FQ	Fluid Qualities Concentrations	Yes
FQC	Fluid Qualities, Decay Control	Yes
~~GC~~	Geometry,continuity check line (use GCL)	Replaced
GCL	Geometry,continuity check line (new)	No
GE	Geometry,element connection table	No
GNN	Geometry,nodal coordinates,bottom elevations	No
GS	Geometry, scale factors	No
GT	Geometry, element type (IMAT)	No
GV	Geometry, eddy viscosity tensor	No
GW	Geometry, one-dimensional cross sectional	No
HD	Hydrodynamic water depth	No
HS	Hydrodynamic Scale factors	No
HU	Hydrodynamic Fluid velocity in x-direction	No
HV	Hydrodynamic Fluid velocity in y-direction	No
MC	Mass Check	No
IC	Initial conditions for quality concentrations	Yes
PE	Peclet control of diffusion coefficients	+
RA	Rain/Evaporation	&
RE	Re-Solve Solution Technique	No
TC	Time control	Yes
TH	Timing for the hydrodynamics	Yes
TO	Timing for RMA4 binary solution write	No
TP	Trace by constituent for all nodes	No
TQ	Trace print summary by Quality constituent #	No
TR	Trace print control for all nodes	No
BC	Boundary condition specification	*
BCC	Boundary condition control parameters	*
BL	Boundary mass loading	*
END	End of boundary condition specifications for the time step	Yes
STO	Stop the simulation	Yes

** At least one of these specifications of boundary conditions is required.*

+ At least one of these specifications for diffusion coefficients is required for each material type $\leq$ 900.

& Rainfall/evaporation definition is required if the RMA2 used this specification.

Input Variables

The following is a table of input variables which are the parameters found on the RMA4 data cards (except those in *italics*, which are not actually input on the card, but are related to the card). The table lists the variable name, a description, and the card or cards to which it is associated.

Variable	Description	Content
ALCOF	Locking frequency per hour	CS
ALPHA_CA	Special calculation variable (alpha)	CA
ALFA1	Angle of control structure in radians	CS
ANG	Element orientation for diffusion tensor	GE, GV
BEGINT	Decimal hour to begin saving RMA4 solution to file	TO
BCFCT	Factor applied to bc concentrations	BCL,BCN
ENDT	Decimal hour to stop saving RMA4 solution to file	TO
WD	The bottom elevation of each node	GNN, GWN
CORD(J,1)	The x-coordinate of node J	GNN
CORD(J,2)	The y-coordinate of node J	GNN
DELT	Length of computation time step	RE, TC
DELTN	Delta time step	BCC
ELEV	The average initial water-surface elevation over the mesh	IC
FLD	Any alphanumeric user comment	CO, END, REV, STOP
BCFCT	Boundary condition ebb/flood buffering factor	BCN
FLD	Alphanumeric comments	CO,END
		STOP,REV
GAMLK	Mixing exchange factor for a locking event	CS
GPEC	Peclet number	PE
IALTBC	Logical unit number for alternate boundary condition file	$L
IBCFCT	Boundary condition ebb/flood buffer control switch	BCN,BCL
IC1	Card group identifier, all cards	ALL
IC3	Data type identifier on some cards	SOME
	Constituent numbers for trace by quality	TQ
ICON	Continuity line number	BRA
IDOS	Switch for dissolved oxygen and BOD	FQ
IECHO	Switch to control echo printing of input control data	TP
IFINO	Logical unit number for RMA4 solution output	$L
IGCL_IN	Array containing the list of check lines representing "inflow" for the mass conservation check	MCI
IGCL_OT	Array containing the list of check lines representing "outflow" for the mass conservation check	MCO
IGEON	Logical unit number for geometric input	$L

Variable	Description	Content
	(GFGEN)	
IHOTN	Logical unit number for initial quality conditions	$L
IHOTO	Logical unit number for RMA4 to write-restart file	$L
IMAT	The element type (n-value and eddy coefficients)	GE, GT, …
IOT	Logical unit number for writing full results listing	$L
IPEC	On/off switch to control eddy viscosity by Peclet number	PE
IPRT	Switch to print element input data, initial conditions and n-values	TP
IREPRT	Report input parameter option	TP
IRESL	Re-solve restore switch	RE
ISAV	Re-solve save switch	RE
ISKIP_END	Number of time steps to skip off the bc input file	BCC
ISPRT	Logical units number for RMA4 summary print results	$L
ITRACE	Trace subroutine calls and controllers (debug)	TP
IVELN	Logical unit number for hydrodynamic input(RMA2)	$L
IVRSID	Computer type identifier	$M
IWPAR	Automatic diffusion parameter diagnostic file switch	$L
JSPRT	List of node numbers for summary print	TRN,TQ
LINE(J,K)	Corner node numbers for continuity check line J	GCL
LMT	Total number of corner nodes on a given continuity line	GC
MC_FLAG	On/off switch for conducting a mass conservation check	MC
METRIC	System International (flag)	SI
NJN	Flow control identifier(>=904)	CS
NJT	Flow control type	CS
NOP(J,I)	List of nodes comprising the connection for element J	GE
NQAL	Number of quality constituents transported	FQ
NQAL_CHECK	The constituent number used for mass conservation check	MC
NTIME	Total number of time steps	TC
NTSEG	Print interval by modulo function	TP
ORT(J,1)	X-direction diffusion coefficient for element J	DX,DF,PE
ORT(J,2)	Y-direction diffusion coefficient for element J	DY,DF,PE
OHGOSH	Diagnostic internal debug print (much)	TP
PBCX	Mass loading by constituent (globally or at node, element, or material type)	BL,BLN, BLT,BLE
SIDF	Elemental inflow rate	RA
SS1, SS2	Left and right channel side slope, respectively	GN,GW
TTSWITCH	Steady state flag switch	TC
TBC	Array containing the BC for each quality consituent	BCN
TCORR	Time correction for input velocity file (RMA2)	TH
TEND	Time of last step to be used on input velocity file (RMA2)	TH
TITLE	Character identifier for the run and all output files	T1-T3

Variable	Description	Content
TMAX	Maximum simulation time	TC
TSTART	Starting hour for the simulation	TC
TOLD(K,ICON)	Initial quality concentration (ICON) for node (K)	IC
VEL	Array containing x-velocity, y-velocity, depth by node	HU,HV,HD
	VEL(1,LOC1) is the x-velocity at node LOC1	HU
	VEL(2,LOC1) is the y-velocity at node LOC1	HV
	VEL(3,LOC1) is the depth at node LOC1	HD
VPEC	Initial guess (Peclet) when using automatic Peclet	PE
WD	Bottom elevation	GN
WIDTH	Channel width	GNN
VOLLK	Volume of the lock	CS
WIDS	Storage width associated with zero depth	GN,GW
WIDTH	Channel width at zero depth	GN,GW
WTEMP	Average initial fluid (water) temperature	FT
XKCOEF(i)	Decay coefficient for each constituent i	FQC
USCALE	Scale factor for x-velocity component	HS
VOLLK	Volumn of the lock	CS
VSCALE	Scale factor for y-velocity component	HS
WTEMP	Water Temperature, Celcius	FT
WSCALE	Scale factor for depth	HS
XSCALE	Scale factor for x-coordinates	GS
YSCALE	Scale factor for y-coordinates	GS
ZSCALE	Scale factor for z-coordinates	GS

RMA4 Execution Job Sheet

JOB EXECUTED ___________________ DATE OF RUN ____/____/____ TIME OF RUN _________________

JOB PRINTED __________________ SUBMITTED BY _____________________ CPUs ____ PRIORITY_____

PURPOSE:

SIMULATION TIME: Start ____________ Finish ____________

FILES:

RMA4 Primary Run Control File ____________________
Geometry file from GFGEN ____________________
Hydro Solution from RMA2 ____________________
RMA4 Hotstart input ____________________

RMA4 Full results listing ____________________
RMA4 summary results listing ____________________
RMA4 Hotstart output ____________________
RMA4 final results file ____________________
RMA4 auto parameter listing ____________________
RMA4 diagnostic files ____________________

GRAPHICAL ANALYSIS:

Concentration plots ____________________
Time Series plots ____________________

COMMENTS:

The RMA4 Data Cards

This section describes all of the data cards used by RMA4. Every effort has been made to describe each card in a clear and complete manner. However, the appropriate sections in the manual text should be consulted when further explanation is desired.

Note: At the time of this writing, SMS does not save/visualize the RMA4 run control file. However, the RMA4 model solution may be visualized in SMS.

T1-T2 Cards: Job Title

Optional

Card Description: Used to provide descriptive information about the data file.

Field	Variable	Value	Description
0, C 1	IC1	T	Card group identifier.
0, C 2	IC3	1, 2	Sequence.
2 - 10	TITLE	Any text	Any alpha-numeric data, up to 77 characters.

Note: A 'T' card must be the first user input card in the primary RMA4 run control file. Any number of T1 and T2 cards may be used and sequence is not significant. The title card section *must* be ended with a T3 card.

Example

```
T1     3700 Elements x 9115 Nodes x 20 Material types
T1
T2     Created by - John Doe
T3     Mississippi River Gulf Outlet -- mean tide
  .
```

See also: "Using Titles" on page 30, T3 card.

T3 Card: Job Title

Required

Card Description: Used to tell RMA4 that there are no more ‘T’ cards to be processed, and to provide descriptive information about the data file.

Field	Variable	Value	Description
0, C 1	IC1	T	Card group identifier.
0, C 2	IC3	3	Sequence.
2 - 10	TITLE	Any text	Any alpha-numeric data, up to 77 characters.

Note: A ‘T’ card must be the first user input card in the primary RMA4 run control file. Any number of T1 and T2 cards may be used and sequence is not significant. However, only one T3 card can be used and it *must* be the last title card in the set. RMA4 reads the ‘3’ to mean END of ‘T’ cards.

Tip: The alpha-numeric information on the T3 card is incorporated into the header of the RMA4 binary solution file. You should use this card to provide information that will later allow the solution file to be identified.

Example

```
T1    3700 Elements x 9115 Nodes x 20 Material types
T1
T2    Created by - John Doe
T3    Mississippi River Gulf Outlet -- mean tide
   .
   .
```

See also: “Using Titles” on page 30, T1-T2 cards.

$L Card: Input/Output File Control

Optional, but recommended

Card Description: Used to specify what types of files RMA4 will read and write.

Field	Variable	Value	Description
0, C 1-2	IC1	$L	Card group identifier.
1	IGEON		GFGEN geometric data, input for RMA4
		0	All geometry will be coded via RMA4 input data
		+	GFGEN's geometry data opened on unit =10
2	IVELN		RMA2 hydrodynamic data, input for RMA4
		0	All hydraulics will be coded via RMA4 input data
		+	RMA2 solution data will be opened on unit =20.
3	IHOTN		Hotstart input.
		0	Initial conditions for RMA4 will be coded in the run control file containing this $L card.
		+	Initial conditions will be read from logical unit =30 (The Hotstart results from the previous run).
4	IALTBC		Alternate dynamic boundary conditions (input).
		0	No alternate boundary conditions file.
		+	Alternate boundary file read from unit =4.
5	IFINO		RMA4 solution output (binary final results of hydraulic calculations).
		0	No RMA4 solution output is saved
		+	RMA4 solution output will be created on unit =31.
6	IHOTO		Hotstart output.
		0	No Hotstart file will be written.
		+	RMA4 will write a Hotstart file on unit =32.
7	IOT		Full (standard) results listing file
		0	No full listing will be created.
		+	Full results listing will be created on unit =33.
8	ISPRT		Summary results listing by node option.
		0	No special list of nodes written.
		+	Summary results listing created on logical unit =34 (TRN or TQ card(s) required).
9	IWPAR		Auto parameter (diffusion) diagnostic file
		0	No special list of nodes written.
		+	Table of automatically calculated diffusion parameters on logical unit=40

Note: Any default logical unit number can be overruled by coding a negative number in the data field for that unit number. The logical unit number will then be the absolute value of the negative number specified.

$M Card: Machine Identifier

Required*

Card Description: Used to specify the type of computer on which RMA4 will be running. The machine type determines the word size used for the matrix buffer temporary files.

Field	Variable	Value	Description
0, C 1-2	IC1	$M	Card group identifier
1	IVRSID		Controller for record length and word size for front solver buffering. Choose the value from one of the following based on the type of computer system on which the model will be running.
		1	**Intel x86 and Pentium Microprocessors (PC).** Direct access record length is unlimited and is defined in terms of bytes.
		2	**Prime Mini-Computer.** Direct access record length is unlimited and is defined in terms of small words (ie. 2 bytes).
		3	**DEC VAX.** Direct access record length, limited to 32K bytes and defined in terms of long words (4 bytes).
		4	**HP or ALPHA workstations, Apple MAC II using ABSOFT FORTRAN, Definicon 020 Board, or DEC VAX to avoid short record limit.** Direct access defined using multiple sequential access files that are opened as required. **Note:** Many files may be left on disk.
		5	**Cray or Cyber-205.** Direct access defined for systems using 64 bit or 8 byte words and where record lengths are defined in bytes.
		6	Same as option 4 above, except the names of the files that are opened will not contain a ‘.’ (dot).
		8	Same as option 4 above, except PAUSE statements in the program are processed for interactive sessions. (Recommended for Apple MAC)

Note*: If no $M card is supplied, IVRSID = 4 by default. The machine identifier is necessary if temporary files known as “buffer blocks” are written or “resolve” is activated.

BCC Card: Boundary Condition Control Parameters

Optional

Card Description: Used if you wish to revise a boundary condition and update parameters between dynamic time steps.

Field	Variable	Value	Description
0, C 1-2	IC1	BC	Card group identifier.
0, C 3	IC3	C	Card type identifier.
1	DELTN	+	The delta time step length in decimal hours. **Note:** The *actual* delta time step, DELT, is revised only if DELTN is greater than zero).
2	ISKIP_END	0, +	The number of END cards to skip in the RMA4 run control file before the simulation begins..

BC Card: Nodal Boundary Conditions

Optional

Card Description: Used to assign constituent concentrations at the specified node(s).

Field	Variable	Value	Description
0, C 1-2	IC1	BC	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** The constituent concentration will be used for all boundary nodes equal to or greater than J. **Warning:** Beware of over-specification.
		N	Specifies **Option 2:** The constituent concentration boundary condition is coded at node J.
		L	Specifies **Option 3:** The constituent concentration boundary condition is coded by check line number J.
1	J		The node or check line number as specified by the value of IC3 above in data field 0, column 3.
		+	**Option 1:** The *starting node* number.
		+	**Option 2:** The *node* number.
		+	**Option 3:** The *geometry check line* number. BCL with negative check line will hold specified concentration regardless of flow direction.
2	TBC(J,1)	+	Constituent concentration (mg/l) for boundary condition quality 1 at node/continuity line, then constituent concentration for boundary condition quality 2 at node/continuity line, ……., then constituent concentration for boundary condition quality NQAL at node/continuity line.
3	IBCFCT(J)		On/Off switch for Boundary Mixing for constituent=J. (Default=0, Off)
		0	OFF. Boundary condition is exactly equal to the constituent's concentration when flow is into the model.
		1	ON. A factor will be applied to the boundary condition concentration to allow a gradual change in the inflow boundary condition after a flow reversal.
4	BCFCT(J)	0<x<1	If the boundary mixing is ON, a factor between zero and one is applied to the boundary condition. A factor close to one is the most gradual re-instatement of concentration after a flow reversal.

Units: Use concentration in the appropriate units of the simulation. Example, use (mg/l) or ppt for English units, and kg for metric concentration.

See also: "Boundary Mixing " on page 42.

BL Card: Boundary Loading (Mass)

Optional

Card Description: Used to assign mass loading on the boundary.

Field	Variable	Value	Description
0, C 1-2	IC1	BL	Card group identifier.
0, C 3	IC3		Card type identifier.
		b̸	Specifies **Option 1:** The mass boundary loading in data field 2 and 3 of this card will be used for all values (J) equal to or greater than ISTART.
		E	Specifies **Option 2:** The mass boundary loading are coded by *element* number.
		T	Specifies **Option 3:** The mass boundary loading are coded by *material type* number (IMAT).
1	ISTART		Code the node number, element number or material type as specified by IC3 above in data field 0, column 3.
		-,+	**Option 1:** The *starting node or starting element* number. A negative ISTART will load all the nodes that are ≥ ABS (ISTART). A positive ISTART will load all the elements that are ≥ ABS (ISTART).
		+	**Option 2:** The *element* number.
		-,+	**Option 3:** The element *material type* number.
			A negative ISTART will load all the nodes that satisfy the material type. A positive ISTART will load all the elements which satisfy the material type.
2	PBCX(J,1)	-,0, +	Mass loading for constituent 1 at ISTART=J.
			See below for units.
3	PBCX(J,2)	-, 0, +	Mass loading for constituent 2 at ISTART=J
4	PBCX(J,3)	-, 0, +	Mass loading for constituent 3 at ISTART=J
5	PBCX(J,4)	-, 0, +	Mass loading for constituent 4 at ISTART=J
6	PBCX(J,5)	-, 0, +	Mass loading for constituent 5 at ISTART=J
7	PBCX(J,6)	-, 0, +	Mass loading for constituent 6 at ISTART=J

Units: Total mass/sec in the appropriate units of the simulation. Example, use ppt for English units, and kg/sec for metric mass.

The BLN option is no longer valid.

CA Card: Special Calculation Variables

Optional

Card Description: Used to provide special calculation instruction for the simulation

Field	Variable	Value	Description
0, C 1-2	IC1	CA	Card group identifier
1	ALPHA_CA	-,0	The default of 1.5 is used if ALPHA_CA $\leq$ 0 or if no CA card is supplied.
		+	Code values between 1.0 and 1.6 **Note**: A value of 1.0 will dictate a fully implicit simulation. A value of 1.5 will specify a second order Taylor Series expansion. **Tip:** Generally, values which are closer to 1.0 will provide more model stability with less accuracy, while values closer to 1.6 provide more accuracy with less stability. With small time steps, the value has little or no effect. With larger time steps, try values closer to 1.0 if model stability is a problem.

Note: Comments may be supplied on this card anywhere within the run control file except as the first or last card types.

CO Card: Comments

Optional

Card Description: Used to provide comments in the run control input file

Field	Variable	Value	Description
0, C 1-2	IC1	CO	Card group identifier
1 - 10	FLD	Text	Any alpha-numeric data

Note: Comments may be supplied on this card anywhere within the run control file except as the first or last card types.

CS Card: Concentrations at a Flow Control Structures

Optional (see note)

Card Description: The concentration mixing exchange parameters for a given flow control structure.

Field	Variable	Value	Description
0, C 1-2	IC1	CS	Card group identifier.
1	NJN	+	Flow controller identifier (904 or larger). Applies these parameters to IMAT = NJN.
2	NJT	+	RMA4 flow controller type: **1** = match concentrations **2** = lock operation; specify fields 3 through 6.
If NJT = 2, Code Field 3-6			
3*	ALFA1	+	Angle of control structure in radians counterclockwise from the positive *x* axis.
4*	VOLLK	+	Volume of the lock (units length^3, either ft^3 or m^3)
5*	GAMLK	+	Mixing exchange factor for a locking event (0.0 ≤ (GAMLK ≤ 1.0).
6*	ALOCF	+	Locking frequency per hour.

* *Use if NJT = 2.*

Note: The CS-Card, Concentration at flow control structures, is required in the RMA4 run control if the RMA2 hydrodynamics had flow control structures (FC card) in the flow simulation.

DF Card: Full Diffusion Coefficient

Required if the PE card or the DF-, DX-, DY- card sets are not used.

Card Description: A combination of the DX card and the DY card for specifying the diffusion coefficients. Also allows for automatic scaling of the *x* and *y* diffusion coefficients.

Field	Variable	Value	Description
0, C 1-2	IC1	DF	Card group identifier.
1	J	+	Element material type (IMAT).
2	ORT(J,1)	-	Invokes Ian King's method of *automatic* scaling of *x* and *y* diffusion of the element type J (unit length/sec). **Note:** Scaling is a function of both velocity and grid size. **Example:** -20 will multiply the *x* diffusion auto-scale by 20. See field 3 below.
		+	The *x* direction diffusion coefficient of element type J (unit length^2/sec).
3	ORT(J,2)	+	The *y* direction diffusion coefficient of element type J (unit length^2/sec). **Note:** If *automatic* scaling is invoked, the *y* diffusion will be multiplied by the value specified here for ORT(J,2).

DM Card: Wet/Dry by Marsh Porosity

Not Assigned in RMA4

Card Description: The Marsh Porosity method makes a more realistic and *gradual* transition when wetting and drying. The marsh porosity option must have been employed during the RMA2 simulation. It cannot be added, changed, or modified in the RMA4 run control.

DX Card: Diffusion Coefficient In The X Plane

Required if the PE or DF card is not used

Card Description: Used to specify a diffusion coefficient in the *x* plane.

Field	Variable	Value	Description
0, C 1-2	IC1	DX	Card group identifier.
1	J	+	Element material type (IMAT).
2	ORT(J,1)	+	The *x* direction diffusion coefficient of element type J (units of length^2/sec, either ft^2/sec, or m^2/sec).

DY Card: Diffusion Coefficient In The Y Plane

Required if the PE or DF card is not used

Card Description: Used to specify a diffusion coefficient in the *y* plane.

Field	Variable	Value	Description
0, C 1-2	IC1	DY	Card group identifier.
1	J	+	Element material type (IMAT).
2	ORT(J,2)	+	The *y* direction diffusion coefficient of element type J. (units of $length^2$/sec, either ft^2/sec, or m^2/sec)

END Card: Mark The End Of Time Step

Required

Card Description: Used to signal the end of all run control instructions for a given time step. For a steady state simulation, only one END card is required. For a dynamic simulation, there will be as many END cards as there are time steps.

Field	Variable	Value	Description
0, C 1-2	IC1	EN	Card group identifier.
0, C 3	IC3	D	Card type identifier.
1-10	FLD	Any	May be used for comments.

FQ Card: Fluid Qualities - Constituent Concentrations

Optional

Card Description: Used to specify the number of quality constituents to simulate. It also assigns special environmental qualities to constituents 1 and 2 for a BOD type of simulation..

Field	Variable	Value	Description
0, C 1-2	IC1	FQ	Card group identifier.
1	NQAL	1-6	Number of quality constituents to transport (program is dimensioned to handle 1 to 6 constituents)
2	IDOS	0,1	On/Off toggle for BOD calculations. IF IDOS=1, Dissolved oxygen (DO) and Biological oxygen demand (BOD) are constituents 1 and 2, respectively

Note: If no FQ card is present, one constituent will be simulated, and BOD is off.

FQC Card: Fluid Qualities - Decay Control

Optional

Card Description: Used to specify the decay coefficient to each constituent.

Field	Variable	Value	Description
0, C 1-2	IC1	FQ	Card group identifier.
0, C 3	IC3	C	Card type identifier.
1-6	XKCOEF	-,0,+	Decay coefficient for each constituent (day^{-1}) Where XKCOEF=0, indicates no decay (default) and a value approaching infinity is rapid decay. Provide a decay coefficient for each constituent specified by NQAL on the FQ Card

Note: The default is no decay. The user supplies a positive decay coefficient in units of day^{-1}, and the RMA4 converts that coefficient to a negative number.

See Also: Decay coefficients are discussed on page 39 and presented as Equation 5

For example, a small decay coefficient equivalent to 50-years or 18250 days, would equal 5.479e-5/days and result in gradual decay. A larger decay coefficient of 1/day would cause rapid decay.

FT Card: Water Temperature

Optional

Card Description: Used to supply the average initial water temperature for the entire mesh.

Field	Variable	Value	Description
0, C 1-2	IC1	FT	Card group identifier.
1	TEMP	+	Average initial water temperature (degrees Celsius).

Note: If no FT card is present, 15 degrees Celsius is used.

GCL Card: Geometry, Continuity Check Line

Optional

This GCL card replaces the GC card.

Card Description: The GCL card is used to specify a line within the grid where the flow rate is of interest. GCL lines may be used to specify the location of boundary conditions.

Field	Variable	Value	Description
0, C 1-2	IC1	GC	Card group identifier.
0, C 3	IC3	L	Card type identifier.
1	J	+	Continuity check line number described by this GCL card.
2-n	LINE(J,K)	+	List the nodes that define this continuity check line. **Note:** If there are more node numbers specifying a continuity check line than will fit in the data fields remaining on the current GCL card, continue coding the remaining node numbers starting in data field 1 of the next GCL card.
n	End of List	-1	A node number of -1 is required to mark the end of the list of nodes that specify this continuity check line.

Example:

```
CO  Line#    Node numbers...Negative number marks the end
GCL    1     10   11   12   13   14   15
GCL          16   17   18   19   20   21
GCL          22   23   -1
GCL    2     100  101   102  109  107  99  -1
```

In general, code the corner nodes making up the continuity check line from right to left when facing downstream. Code only corner nodes to define the line.

Note: The maximum number of flow continuity check lines that can be calculated is determined by the value of MCC, a parameter variable in the source code (generally about 100). The maximum number of nodes allowed for one check line is determined by the value of MCCN.

Code all lines in the same direction (i.e., right to left across the flow), otherwise, mass flux results may be mis-interpreted since some values may be reported in the printout as positive and some as negative.

Tip: Continuity check lines can be used to help verify that the model is providing acceptable results by comparing the mass flux results at the points specified with the corresponding mass flux from field data. The mass flux across the line will be reported in the full results listing file

See also: “Constituent Mass and Mass Flux Checks” on page 57.

GE Card: Geometry, Element Connection Table

Optional

Card Description: The GE card is used to create or modify elements in the mesh using nodes specified on GN cards with the N option, or nodes available in the mesh with which an element can be attached or constructed.

Field	Variable	Value	Description
0, C 1-2	IC1	GE	Card group identifier.
1	J	+	Element number.
2-9	NOP(J,I)	+	Code up to 8 node numbers for element J, listed counterclockwise around the element, starting from any *corner* node.
10	IMAT(J)	+	Material type number for element J (optional, may be specified on GT card.
11	ANG(J)	-, 0, +	Direction of eddy viscosity tensor in **radians**, counterclockwise from the positive x-axis (optional, may also be specified on GV card **Note:** For 1-D elements, the direction is automatically aligned with the orientation of the 1-D element

Note: The element connection table will usually be provided by the GFGEN pre-processor and will reside on the logical unit for GFGEN Geometric Data ($L card). If so, the GE card should be omitted from the RMA4 run control file unless small mesh revisions are required. Otherwise, code the Nodal Point-Element Connection Table.

Tip: To effectively remove an element from the computational mesh, set the material type (IMAT) for the element to a value of zero (the element will appear to be land). This technique may be useful when troubleshooting problems in your mesh.

Tip: Another way the GE card can be useful is when you need to test changes to the mesh but you do not want to commit to modifying the geometry file until you know the results of the changes. You may use the GE card, along with other geometry cards, in the RMA4 run control file to make these types of changes.

See also: GFGEN User's manual.

GN Card: Geometry, Nodal Point Coordinates

Optional

Card Description: Used to add a node to the mesh or to modify the (*x*, *y*, *z*) coordinates of an existing node.

Field	Variable	Value	Description
0, C 1-2	IC1	GN	Card group identifier.
0, C 3	ISI	N	Card type identifier.
1	J	+	Node number.
2	CORD(J,1)	-, 0, +	*x*-coordinate at node J. Units of length
3	CORD(J,2)	-, 0, +	*y*-coordinate at node J. Units of length
4	AO(J)	+	Bottom elevation. Units of length.
5 (1D)*	WIDTH	+	Channel width at zero depth for node J. Units of Length
6 (1D)*	SS1	-, +	Left side slope at node J.
7 (1D)*	SS2	-, +	Right side slope at node J.
8	WIDS	+	Off Channel Storage width associated with node J at zero depth. Units of length **Note:** This feature is primarily used for model verification purposes.
9	WSCRIT	-,+	Water surface elevation to activate Off-Channel storage. Units of length
10	SSS	-,0,+	Side slope for off-channel storage

** Used for 1D nodes only. If you use SMS (or FastTABS version 3.02 or later), it is preferred that data for fields 5 through 10 be supplied on GW cards with the N option.*

Note: The coordinate values read (CORD(J,1), CORD(J,2), and AO(J)) are multiplied by the appropriate scale factors, XSCALE, YSCALE, and ZSCALE from the GS card. Be sure to specify the coordinates in the appropriate length units (feet or meters) as specified on the SI card.

It is possible to use the GN card without specifying data field 4 through 10 if only nodal position is to be modified.

All slopes are with respect to one unit of rise.

SMS 8.0

SMS Note: As of this writing, SMS does not support the parameters 6-10 on the GN card.

See also: GFGEN User's manual

GS Card: Geometry, Scale Factors

Optional

Card Description: This card applies scale factors for the *x* and/or *y* coordinates.

Field	Variable	Value	Description
0, C 1-2	IC1	GS	Card group identifier.
1	XSCALE	+	Scale factor for X-coordinates.
2	YSCALE	+	Scale factor for Y-coordinates.

Note: If no GS card is present then all scale factors are 1.0. To convert feet to meters, the scale factor should be 0.3048.

See also: "Specifying Units" on page 31.

GT Card: Geometry, Element Material Types

Optional

Card Description: Used to specify or modify element material types.

Field	Variable	Value	Description
0, C 1-2	IC1	GT	Card group identifier.
1	J	+	Element number.
2	IMAT(J)	0, +	Element material type number.
3-10		+	You may provide (J, IMAT(J)) sets of values.

Tip: To effectively remove an element from the computational mesh, set the material type (IMAT) for the element to a value of zero (the element will appear to be land).

See also: GFGEN User's manual

GV Card: Geometry, Eddy Viscosity Tensor

Optional

Card Description: Used to specify or modify an element's eddy viscosity tensor.

Field	Variable	Value	Description
0, C 1-2	IC1	GV	Card group identifier.
1	J	+	Element number.
2	ANG(J)	-, 0, +	Direction of eddy viscosity tensor (Radians, counterclockwise from the positive *x*-axis)

If desired, you may fill the GV card in complete element/direction sets, or continue on another GV card.

Note: RMA4 requires an eddy viscosity tensor for every element in the mesh. Normally, this information is specified with GE cards in the GFGEN geometry file where the default value is zero degrees. The eddy viscosity tensor can also be specified in the RMA2 run control file on GE cards.

See also: GFGEN User's manual

GW Card: Geometry, One-dimensional Channel Width Attributes

Required for one-dimensional nodes if GN cards with the N option are not used

Card Description: Used to only specify or modify *one-dimensional* trapezoidal channel attributes at the node specified.

Field	Variable	Value	Description
0, C 1-2	IC1	GW	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Universal assignment for all nodes ≥ NODE.
		N	Specifies **Option 2:** Individual node assignment.
1	NODE		The *one-dimensional* starting node or node number as specified by IC3 above. **Note:** Enter one-dimensional corner nodes and transition nodes.
		+	**Option 1:** The *starting* node number.
		+	**Option 2:** The node number.
2	WIDTH	+	Channel surface width at zero depth for NODE. Units of length.
3	SS1	-, +	Left side slope at NODE.
4	SS2	-, +	Right side slope at NODE. **Note:** Off-Channel Variables Follow.
5	WIDS	+	Off-Channel Storage width associated with NODE at zero depth. Units of length.
6	WSCRIT	-,+	Water surface elevation to activate Off-Channel storage. Units of length.
7	SSS	-,0,+	Side slope for off-channel storage

Note: Code only *one* corner node per GW card. All slopes are with respect to one unit of rise.

If you are using SMS, the GW card is the preferred method for defining one-dimensional channel width attributes as opposed to the GN card.

SMS 8.0 **SMS Note:** SMS-8 will read and interpret, but will not update GW card data. Be aware that if the grid is renumbered by SMS, the node numbers on GW cards may no longer agree with the new element connection table. If this is the case, you will have to update the GW cards manually.

See also: GFGEN User's manual

HD Card: Hydrodynamic, Water Depth

Optional

Card Description: This card assigns the water depth when no RMA2 hydrodynamics are specified.

Field	Variable	Value	Description
0, C 1-2	IC1	HD	Card group identifier.
1	J	+	Starting node number at which this global assignment will be made.
2	VEL(3,J)	+	Depth of water at node J. Units of length.

Note: If no HD card is read, the input RMA2 hydrodynamic solution should be used to specify depth at each individual node. The HD card input water depth units of length will be scaled by the HS card water depth scale factor.

HU Card: Hydrodynamic, X-Velocity

Optional

Card Description: This card assigns the water depth when no RMA2 hydrodynamics are specified.

Field	Variable	Value	Description
0, C 1-2	IC1	HU	Card group identifier.
1	J	+	Starting node number at which this global assignment will be made.
2	VEL(1,J)	+	X component of velocity at node J (length/sec)

Note: If no HU card is read, the input RMA2 hydrodynamic solution should be used to specify velocity components at each individual node. The HU card input x-velocity units of length/sec will be scaled by the HS card x-velocity scale factor.

HV Card: Hydrodynamic, Y-Velocity

Optional

Card Description: This card assigns the water depth when no RMA2 hydrodynamics are specified.

Field	Variable	Value	Description
0, C 1-2	IC1	HV	Card group identifier.
1	J	+	Starting node number at which this global assignment will be made.
2	VEL(2,J)	+	Y component of velocity at node J (length/sec)

Note: If no HU card is read, the input RMA2 hydrodynamic solution should be used to specify velocity components at each individual node. The HV card input y-velocity units of length/sec will be scaled by the HS card y-velocity scale factor.

HS Card: Hydrodynamic, Scale Factors

Optional

Card Description: This card assigns the scale factors to be applied to the input RMA2 hydrodynamics.

Field	Variable	Value	Description
0, C 1-2	IC1	HD	Card group identifier.
1	USCALE	+	X-velocity scale factor to be applied to the hydrodynamics.
2	VSCALE	+	Y-velocity scale factor to be applied to the hydrodynamics.
3	WSCALE	+	Depth scale factor to be applied to the hydrodynamics.

Note: If no HS card is read, all hydrodynamic scale factors are set to one. Set all three of the scale factors to 0.3048 to convert from feet to meters.

See also: “Specifying Units” on page 31.

IC Card: Initial Quality Concentration

Required for Coldstart

Card Description: This card assigns the initial concentration of constituent particles for the simulation.

Field	Variable	Value	Description
0, C 1-2	IC1	IC	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Universal assignment of concentration for all nodes ≥ ISTART.
		T	Specifies **Option 2:** Concentration assigned by element material type.
		E	Specifies **Option 3:** Concentration assigned by element number.
		N	Specifies **Option 4:** Concentration assigned by node number.
1	ISTART		The starting node number, or specific material type, element, or node number, as defined by IC3 above.
		+	**Option 1:** The starting node number for universal assignment.
		+	**Option 2:** The material type number.
		+	**Option 3:** The starting element number.
		+	**Option 4:** The node number.
2-7	TOLD	+	Initial quality concentration for each constituent. Enter from 1 to the maximum number of quality constituents (NQAL) indicated on the FQ card. (mg/l or ppt for English, kg for metric units)

MC Card: Mass Check

Version 4.50 or higher

Optional

Card Description: This card assigns the initial concentration of constituent particles for the simulation.

Field	Variable	Value	Description
0, C 1-2	IC1	MC	Card group identifier.
0, C 3	IC3		Card type identifier.
		b̸ (blank)	**Option 1:** Request a Conservation of Mass Check.
		I	**Option 2:** Specify a list of inflow check lines.
		O	**Option 3:** Specify a list of outflow check lines.
1	MC_FLAG	0,+	**Option 1:** On/Off switch to request a check for mass conservation upon completion of the simulation. <u>Default is no checking</u>
2	NQAL_CHECK	+	**Option 1:** The quality constituent number to use for the mass conservation check. <u>Default</u> = constituent #1
1-10	IGCL_IN	+	**Option 2:** List of continuity check lines representing the total inflow for purposes of performing a mass continuity check. <u>Default</u>: line #1 is inflow (IGCL_IN),
1-10	IGCL_OT	+	**Option 3:** List of continuity check lines representing the total outflow for purposes of performing a mass continuity check. <u>Default</u>: the last continuity check line defined will be the outflow (IGCL_OT).

Example: Mass Check request, where lines 4 and 5 are the inflow, and line 6 is the outflow.

```
MC    1    1
MCI   4    5
MCO   6
```

PE Card: Automatic Peclet Number Control of Diffusion

Required if no DF-cards or (DX-,DY-Card) sets are used.

Card Description: Used to provide for *real time* adjustment of the diffusion coefficient based upon the computed velocity and individual size of each element. Larger elements and elements with higher velocities will have larger diffusion coefficient values. Also, smaller Peclet numbers will result in larger values of diffusion coefficient.

Field	Variable	Value	Description
0, C 1-2	IC1	PE	Card group identifier.
0, C 3	IC3	b̸ (blank)	Specifies **Option 1:** Set Peclet parameters for the entire mesh.
		T	Specifies **Option 2:** Set Peclet parameters for the element *material type* (IMAT).
1	IPEC	0, 1	**Option 1:** IPEC is an On/Off switch for automatic diffusion coefficient by Peclet number (Zero turns the option off).
	IMAT	+	**Option 2:** *Material type* for the Peclet parameter assignment.
2	GPEC	+	Peclet number assigned for every element of material type IMAT. **Note:** A smaller Peclet number will result in a larger value for diffusion, and visa-versa.
3	VPEC	+	**Coldstart:** For every element of material type IMAT, VPEC is the *initial guess* used for the average elemental velocity. **Hotstart:** The *minimum* velocity (*length/sec*) to be used to compute the automatic diffusion coefficient equation. **Note:** Remember that the velocity must be input in length/sec units after scaling has been applied.
4	ORT(J,1,)	+	Ratios for computed diffusion in the *xx* and xy direction for all IMATs from the Peclet calculation (default = 1).
5	ORT(J,2)	+	Ratios for computed diffusion in the *yy and yx* direction for all IMATs from the Peclet calculation (default = 1).

Example:

PE	1	20	0.5	1.0	1.0
PET	9	15	0.5	1.0	1.0

This example issues a global Peclet number of 20 as a global assignment everywhere, except a Peclet number of 15 will be used for material type 9 (IMAT=9).

Recall the formula for Peclet numbers (P) introduced in the RMA2 User's Guide, where P is recommended to be between 15 and 40.

$$P = \frac{\rho u dx}{E}$$

However, for RMA4, the Peclet number (P) takes the form

$$P = \frac{udx}{D}$$

As can be easily seen, D can be approximated as

$$D_x = \frac{E_{xx}}{\rho}$$

where:

Coefficient			English Units	Metric Units
ρ	=	fluid density	1.94 slugs/ft^3	998.46 kg/m^3
u	=	average elemental velocity calculated at the gauss points	ft/sec	m/sec
dx	=	length of element in streamwise direction	ft	m
E	=	Eddy Viscosity for RMA2	lb-sec/ft	Pascal-sec
D	=	Diffusion Coefficient for RMA4	ft^2/sec	m^2/sec

See also: "Specifying The Diffusion Coefficients" on page 34.

RA Card: Rainfall And Evaporation

Version 4.32 or higher

Required if rainfall/evaporation was used in RMA2

Card Description: Used to assign concentration to the rainfall or evaporation condition that was specified during the RMA2 hydrodynamic simulation.

Field	Variable	Value	Description
0, C 1-2	IC1	RA	Card group identifier.
1	IC3		Card type identifier.
		ƀ	Specifies **Option 1**: Use the unit discharge in data field 2 for all elements equal to or greater than J
		E	Specifies **Option 2**: The *element* number for element inflow
		T	Specifies **Option 3**: The *element material type* number for element inflow.
2	J		
		+	**Option 1**: The starting *element* number
		+	**Option 2**: The *element* number
		+	**Option 3**: The *element material type* number
3	SIDF	-,0,+	The elemental inflow rate,
			Specified in *inches/hour* if the SI card is set to English units (or *cm/hr* if set to Metric). Positive values represent rainfall (inflow), negative values represent evaporation (outflow).

See also "Rainfall / Evaporation Concentration Boundary Condition" on page 45

Note: No RA card is necessary unless the RMA2 hydrodynamic flow field also had rain/evaporation.

RE Card: Re-Solve Solution Technique

Optional

Card Description: This card provides the means of saving the matrix for re-solution.

Field	Variable	Value	Description
0, C 1-2	IC1	RE	Card group identifier.
1	ISAV		Re-solve save switch.
		-1	Do not save global matrix for re-solution.
		0	Action based on the value of IRESL.
		1	Save the global matrix for re-solution.
2	IRESL		Re-solve restore switch.
		0	Action based on the value of ISAV.
		+	Use the re-solve file saved during time step IRESL.
3	DELTH	0,+	Time step for this solution step (decimal hours). This value is used only when ISAV is active. Note that the delta time step from the resolve file is used if IRESL is active.

Note: No RE card is necessary if the delta time step is zero, ie steady state.. This feature has the potential of greatly increasing the computation run time.

SI Card: System International Units

Version 4.32 or higher

Optional

Card Description: Used to control the type of units RMA4 will use. Units are English or Metric.

Field	Variable	Value	Description
0, C 1-2	IC1	SI	Card group identifier.
1	METRIC	0	English units were used for both the GFGEN geometry (ft) and RMA2 velocity field (ft/sec) that were input for this simulation.
		1	Metric units are expected as input and used for output (default).

Note: The SI card must be placed early in the card line-up in the run control file (immediately after the $L card). If no SI card is present, metric units are assumed.

See also: "Specifying Units" on page 31.

STO(P) Card: Stop the RMA4 Simulation

Required

Card Description: This card signals the end of *all* computation after the current time step has been completed.

Field	Variable	Value	Description
0, C 1-2	IC1	ST	Card group identifier.
1	IC3	O	Card type identifier.
2-10	FLD	Any	May be used for comments.

This card will override any previous control concerning the length of the RMA4 simulation.

See also: "END Card: Mark The End Of Time Step" on page 96

TC Card: Timing Control for RMA4 Simulation

Required

Card Description: This card specifies the timing controls for the RMA4 simulation.

Field	Variable	Value	Description
0, C 1-2	IC1	TC	Card group identifier.
1	TSTART	0,+	Starting time for the RMA4 simulation (decimal hours).
2	DELT	0,+	Delta time step (decimal hours).
3	NTIME	+	Total number of time steps.
4	TMAX	0,+	Maximum simulation time (decimal hours).
5	TTSWITCH		Time term switch. *Previously known as a "steady state flag"SSF switch.*
		-1	Time terms are on. With ALPHA=1.0 the calculations are completely implicit, which is less accurate than explicit. The explicit calculations are more stable and generally produces more diffusion in the model.
		0	Time terms are off. Time terms are omitted from the computations. **Note:** Used only for initialization.
		1	Time terms are on (default). With ALPHA = 1.5, a second order Taylor Series Expansion.

TH Card: Timing Control for Hydrodynamics

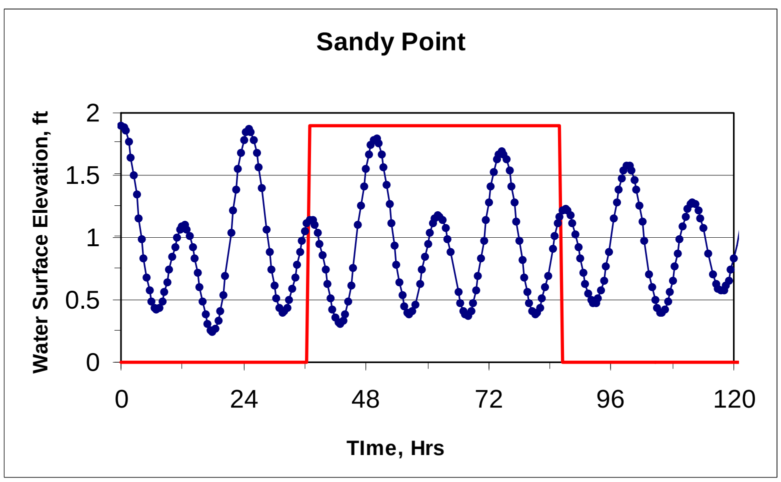
Required

Card Description: This card specifies what part of the RMA2 hydrodynamic solution will be used for the RMA4 simulation.

Field	Variable	Value	Description
0, C 1-2	IC1	TH	Card group identifier.
1	TCORR	0,+	The decimal hour in the RMA2 velocity solution at which to start. Hours to be subtracted from the input RMA2 input velocity file.
2	TEND	0,+	The decimal hour in the RMA2 velocity solution at which to end. Time of the last step on the input RMA2 velocity file.

Note: Upon reaching "TEND', the RMA2 velocity file will be rewound and used again.

In the RMA2 simulation example shown below, Sandy Point was the tidal water surface boundary condition. For the RMA4 simulation, TCORR=37.0 hours and TEND = 86.0 hours.



TO Card: Timing for RMA4 Solution Output

Optional

Card Description: This card specifies what part of the RMA4 solution file will be saved to disk.

Field	Variable	Value	Description
0, C 1-2	IC1	TO	Card group identifier.
1	BEGINT	0,+	Decimal hours to begin saving RMA4 solution.
2	ENDT	0,+	Decimal hour to stop saving the RMA4 solution.

Note: If no TO card is present, the numerical results for all time steps are recorded in the RMA4 solution file, otherwise, only the hours between BEGINT and ENDT will be saved to the solution file.

```
Example:  TO  100  200
```

will save all computed time steps between hours 100 and 200, inclusively.

TP Card: Trace Print by Constituent for All Nodes

Required

Card Description: Listing control used to specify the type of information to be written, as well as the write frequency.

Field	Variable	Value	Description
0, C 1-2	IC1	TP	Card group identifier.
1	IREPRT		Control for the reporting of input run control parameters or initial conditions to the full results file
		0	A report is generated, except initial nodal information and element connections are not reported.
		+	Activate the full report. If IREPRT>1, then this variable serves as an incremental counter on all nodal and elemental listings.
2	IPRT		Full results listing print option.
		0	Print all input data except suppress initial conditions and detailed geometry information.
		1	Print all input data, and use the expanded form for results.
		2	Print all input data except detailed geometry information, and use the short form for results.
3	NTSEG		Full results listing print interval.
		0	Turn off print for results.
		+	Use modulo function to determine print interval for results, such that if NTSEG is an even multiple of the time step, then write to the listing file. **Example:** If NTSEG = 3, write to the file every 3rd time step.
4	IECHO		Data card echo to standard output *.
		0	Do not echo input cards as they are read.
		1	Print input cards as they are read (default).
5	ITRACE		Subroutine trace print to standard output *.
		-1	No subroutine trace, but permits diagnostic "negative depth" file to be created.
		0	No subroutine trace (default).
		1	Trace program logic as each subroutine is called.
6	IOHGOSH		In-house debug to standard output *.
		0	No detail internal print trace (default).
		1 - 4	Diagnostic debug print traces.

** Standard output is normally sent to the terminal screen. A log file is the standard output when running in Batch mode.*

TQ Card: Trace By Constituent Quality

Optional

Card Description: Summary results listing control used to specify which nodal concentrations are listed for the given constituent number.

Field	Variable	Value	Description
0, C 1-2	IC1	TQ	Card group identifier.
0, C 3	IC3		Trace constituent.
		1	Constituent 1
		2	Constituent 2
		3	Constituent 3
		4	Constituent 4
		5	Constituent 5
		6	Constituent 6
1	JSPRT(J,1)	+	Node number(s) to trace. End the list with **-1**.

TR Card: Trace All Active Constituents by Node

Optional

Card Description: Summary results listing control used to specify which nodal concentrations are listed for all active constituent numbers.

Field	Variable	Value	Description
0, C 1-2	IC1	TR	Card group identifier.
0, C 3	IC3	N	Trace all active constituents at node JSPRT.
1	JSPRT(J,1)	+	Node number(s). End the list with **-1**.

Note: To save the nodal information to a separate file, activate the variable, ISPRT, on the $L-card and specify a output file name with the R4_SRL keyword.

Performance Enhancements

Why Is RMA4 Running Slow?

RMA4 is a complex numerical model, which performs many calculations per time step to obtain a solution. To run efficiently, it requires a system with ample resources (disk space, memory, CPU power). There are several factors which may contribute to a reduction in run performance.

Computer Processor Speed

Obviously, the faster your CPU, the faster RMA4 will be able to solve your problem. Unless you do not need results in a hurry, it is not recommended to run an RMA4 simulation of any significant size on a computer with less power than a 486 PC. In fact, a workstation environment is the preferred setting.

Computer Physical Memory (RAM)

Each of the TABS-MD models, including RMA4 will perform optimally if the the global matrix being solved fits entirely in internal physical memory.

Temporary Files

RMA4 writes temporary buffer files when the size of the solution matrix exceeds the dimensions of the matrix array. The temporary files are used as additional space to store the solution matrix. The number and size of these files depends upon the dimensions of the solution matrix array as defined in the source code, and upon the computer system on which RMA4 is running.

The process of writing to and reading from temporary files places a significant strain on the performance of RMA4, and is further exaggerated when the access time of the disk is slow. For information on reducing or eliminating the number of temporary files, see the section entitled “Setting the Optimal File Buffer Size ” on page 126.

Disk Performance

The speed at which your disk can transfer data will have some influence on the time that RMA4 will take to complete your simulation. Especially if RMA4 is writing temporary files, a slow disk will hamper performance.

Disks can become fragmented with use over time. Check your disk and de-fragment if necessary.

Fragmented disks can reduce performance. If your disk is fragmented, new files may be written in pieces and scattered all over the disk. This means the disk drive has to temporarily stop reading or writing the file, move to another location, and continue; thus, the loss of performance.

See section "Eliminate Unnecessary I/O" on page 69

Setting the Optimal File Buffer Size

If you have the RMA4 source code and a FORTRAN 77 compiler, you can customize RMA4 to better conform to your system. With a little experimentation and manipulation of array dimensions, you may be able to realize considerable performance improvements

If your system has a large amount of RAM, say 256 megabytes or more, it is likely you can improve performance by dimensioning RMA4 to decrease the number of temporary files that are written. You can reduce the number of temporary files by increasing the size of the buffer used to store the solution matrix. This is done by changing the variable NBS in the RMA4 include file to a value which is better suited for your system. For most problems, with the number of elements less than 10,000, a value between 100,000 and 200,000 should eliminate the temporary files.

How the Buffer Size Relates to File Size

The estimated size of the buffer in bytes can be determined prior to actually running the model. For instance, for the 64-bit workstations, that have 8 bits/word, 8 bytes/word, and the MSV and NBS variables set to = 3 million, then the size of one buffer file would be = 144,000,016.

Total Words	2 + (3 * MSV) + (3 * NBS)
Total Bytes	Total Words * Number of Bytes per Word for you computer system

Where NBS and MSV are parameter variables, set prior to compilation. NBS is the buffer size and MSV is typically equal to the maximum number of nodes. For more details about parameter variables, see section "Redimensioning RMA4" on page 127.

Calculating the Optimal Buffer Size

Set the NBS parameter variable to an initial guess. Find the string "`BUFFER BLOCKS WRITTEN`" in the full results or screen listing.

If the number of buffer blocks written is zero, then the buffer size is large enough for the problem. The only concern at this point is where the buffer size is too large. See section "Effects Of An Oversized Buffer" on page 127.

If the number of buffer blocks written is non-zero, then make the calculation for NBS*new* as shown. If your computer has enough RAM to accommodate a larger buffer size, then modify the parameter variable NBS in the RMA4 include file and recompile the program to reflect the re-dimensioning. See section "Redimensioning RMA4" on page 127.

```
Given a value of Parameter variable NBS = NBSold
BUFFER BLOCKS WRITTEN=         N
FINAL LQ SIZE        =         S

NBSnew greater than N * NBSold + S
```

Effects of A Buffer Set Too Small

During the simulation there will be numerous buffer blocks written. The machine identifier ($M card) dictates whether this will be one large file with numerous records or whether each block will be written to a separate file. Irregardless this causes the disk to be used in place of memory. It is generally true that I/O speed to/from a hard drive is slower than internal memory, therefore the model will run slower than optimal.

Effects Of An Oversized Buffer

Some systems have the ability to use virtual memory, meaning the disk is used as memory when the available RAM is full. If you make the buffer size too large, RMA4 may run, but be aware that your system may run out of actual RAM and begin using the disk, in which case you are no better off than before, maybe worse.

The symptoms that this is happening in RMA4 would be: still running slow, the disk is constantly in use during the run, and there are no temporary files written. If this is the case, try reducing the buffer size until your system no longer needs to use virtual memory. The idea is to reach a compromise, having the least amount of temporary files or running entirely in RAM.

Redimensioning RMA4

RMA4 is written in standard FORTRAN 77 syntax.

RMA4 is also compatible with FORTRAN-90.

There are several arrays whose size can be modified, and it should be determined by your problem and the system on which you are running RMA4. These array dimensions can be changed in the source code by editing the RMA4 include file and recompiling. The array dimension variables are as listed in the table below.

Variable	Description
MND	Maximum number of nodes in the mesh.
MEL	Maximum number of elements in the mesh.
NBS	Maximum solution matrix buffer size (in RAM). This dimension influences the number of temporary files written.
MFW	Maximum front width. A function of the mesh domain length to width ratio, the number of nodes and elements, and the success of GFGEN's reordering (GO card).

MSV	Buffer write control. Suggestion: set MSV equal to MND
MSTEP	Maximum number of time steps
MPB	Maximum size for the summary-listing buffer. This should be large enough to accommodate the number of nodes specified on the TRN card, multiplied by the number of time steps. Example: To create a summary listing for 8 nodes, for a simulation with 40 time steps, MPB should be at least 8 * 40 = 320.
MBNP	Maximum boundary node points
MBB	Maximum boundary buffer
MCC	Maximum number of continuity check lines.
MCCN	Maximum number of nodes per continuity check line.
MQAL	Maximum number of quality constituents
MFCS	Maximum flow control structures

Example Parameter statement in the rma4v450.inc file:

PARAMETER (MND=33000,MEL=11000,NBS=3000000,MFW=550,MSV=MND,

MSTEP=1500,MPB=24000,MBNP=300, MBB=10)

Example Compile Commands:

Standard FORTRAN F77 unix compile: f77 -o rma4v450.exe rma4v450.f

Machine Precision Issues

In some projects, machine precision may become an issue. Different types of computer systems has different word lengths, and this word length can affect precision. If you suspect RMA4 will encounter very large or very small numbers during a simulation, be aware that machines with smaller word lengths, such as 16-bit or 32-bit PC's, may not be able to make the calculations satisfactorily. For instance there are 8 digits of information stored with a 32 bit computer.

Work Around Tips

Some common tricks of the trade are:

- Reduce the number of significant digits in the (x,y) coordinate scheme applied in the geometry.
- Alter the units in which the computations are being made.
- Comile the model in double precision.

FORTRAN Compiler Issues

There are many compiler options: precision, debug, and optimization switches are commonly available.

Precision compiler options are usually a toggle of computing in either single or double precision. Since the RMA4 model internally computes the shape functions in double precision, it is usually a waste of speed and resources to calculate everything in double precision. There are unusual circumstances, such as those mentioned in section "Machine Precision Issues" on page 128, where this generalized statement is not true.

FORTRAN compilers have options to allow for debug, such as: array bounds checks, exponential over- and under-flow, trace back options, etc. Most all diagnostic compiler options will drastically slow the computational performance. These options usually are not activated except for debugging new-source code developement.

Activating the optimization switches may result in a desirable speedup. However, it is recommended to test any new compiler option not only for performance but for accurancy of computation.

Common Problems

The Simulation Stops Prematurely

If the model is running interactively, an alarm beep or a screen flash will alert the user that a problem has been detected. Most of these errors cause the model to stop execution

Cause 1

File was not saved properly.

Remedy

- Disk capacity or quota has been exceeded

Cause 2

"... record requires too much data"

Remedy

- Improper file assignments (ie., used RMA4 final results binary as a HOTSTART input)
- $M-card has the wrong machine identifier

Cause 3

"IRESL not defined"

Remedy

RE-card was not read during input

Cause 4

"...about to divide by zero ... RMA2 time previous cannot equal present"

Remedy

The user may be trying to run both Steady state and dynamic in one run

1. First run steady state and save a HOTSTART file.

2. The TTSWITCH and DELT variables (TC-card) must be zero.
3. Then read the HOTSTART for initial conditions and run dynamic.
4. Now TTSWITCH=-1 and DELT is a positive number.

Cause 5

The central processor unit has run out of allotted time. The simulation requires more CPU time than the computer allows. This problem can occur on mainframe type computer systems where a user's CPU use is limited.

Remedy

1. If possible, increase the maximum CPU time available to you.
2. Divide the simulation into two or more smaller simulations using the Hotstart capability of RMA4.
3. Use batch queues that offer larger CPU times.

Error Concerning Temporary File

(Logical Unit 9)

Cause 1

The $M card has the wrong machine identifier.

Remedy

The machine ID = 1 for DOS/Windows-95/98/2000/NT PC,

3 for VAX, 4 for UNIX workstations, 5 for Cray Y-MP, and 8 for Macintosh.

Cause 2

All disk space has been exhausted.

Remedy

Delete any temporary files which may have been left from a previous RMA4 run. If there is adequate RAM available, recompile with a larger buffer size.

Hotstart Difficulties

Cause

Improper settings for Hotstarting

Remedy

Double check these hotstart control parameters:

- IHOTN > 0 on the $L card for inputting a hotstart
- IHOTO > 0 on the $L card for creating a hotstart file
- Verify that the proper file names are being used
- Check if the file was contaminated by incorrectly moving the file from computer A to computer B
- Verify the ISKIP_END setting on BCC card

Exhausted All Disk Space

Cause

There is not enough space left on the disk for storing information.

Remedies

1. Purge unnecessary files or obtain more disk space.
2. Reduce any unnecessary diagnostics (ITRACE=0 on TP Card)
3. Only save the solution time steps after the spin-up period (TO card).
4. Save the summary results listing rather than a full results listing (TRN card and $L card).
5. Set the logical unit number to zero for all output files which are not critical ($L card).
6. Check to see if all temporary files and/or core dumps were deleted.
7. If there is enough RAM available, optimize the buffer size for your problem
8. Purchase more disk space.

Oscillations in Concentration

Cause

An oscillation is any unexplained irratic behavior in the numerical solution. When the "high-low-high" concentration solution is observed from node to node on an element edge, it is commonly called “2-Delta-X Oscillation”. It is usually the result of the diffusion assignment not being appropriate for the size of element in the area.

Remedy

Fix one or all, as necessary:

- Refine the resolution of the mesh
- Increase the diffusion parameters
- Reduce the RMA4 time step
- Consider altering the ALPHA_CA variable on the CA Card
- Check the re-solve control parameters

Execution Running Slower Than Expected

Cause

There can be many reasons for RMA4 to run slow. Below are a few suggestions for improving performance.

Remedy

1. If there is ample RAM available, try increasing the dimension of the parameter statement in the RMA4 include file which specifies the

buffer size for temporary files. The idea is to have fewer buffer writes and more in memory.

2. Try to cut back on the number of files saved and the frequency of the write. Input/Output operations can severely decrease speed. Also, try to run the model on a computer with high speed I/O.
3. Check to see if the disk space on the computer is fragmented.
4. When compiling the source code, use compiler optimizations.
5. Run the simulation when the computer is not sharing resources.
6. Consider using the Resolve capability. See RE-Card.
7. Check compiler switches. Deactive debug, array bounds checks, underflow and overflow check.

See also "Speeding Up The Computation" on page 68.

Warning And Error Messages

Array Bounds Underflow

Cause

A node or element number is zero or negative.

Remedy

Examine the GFGEN geometry and/or RMA4 run control file.

Array Bounds Overflow

Cause

Too many nodes, elements, equations, continuity checks, etc. for the program dimensions.

Remedies

1. Decrease the problem size.
2. Increase the dimensions of the model.

Error While Reading Data

Cause 1

Unexpected end-of-file was hit before all data was processed.

Remedy

Check for:

- misspelled file names
- missing END cards and STO card
- wrong filenames assigned
- corrupted data file(s)

Cause 2

An illegal card type.

Remedy

An illegal card type is typically a typographical or case error, or a blank line. Be sure the card names and parameters are typed correctly, and use upper case for card names.

Note: Never use the **TAB** key to separate data in RMA4 input files.

Error Reading An Input Binary File

Cause

The binary file must have been created by the same computer that is attempting to read the file. Transferring a binary file created from on one computer to another computer which is not of the same family will not work! For example, from a Cray-YMP (64-bit) to a UNIX (32-bit) workstation requires a binary conversion.

Remedy

Use a utility to convert the binary file to ASCII, move it to the alternate machine, then convert it back to binary on the new machine. See the section "Moving RMA4 Binary Solution Across Platforms" on page 145 .

RMA4 File Formats

TABS Binary File Header Format

All TABS-MD models operate on and generate binary files, each of which have several header or banner records that describe critical features about the contents of the file. The binary files are written by standard FORTRAN 77 unformatted write statements. The variable types follow standard FORTRAN assignments, where variables are REAL, except for those beginning with the characters 'I' through 'N' which are INTEGER.

Record 1

MFLG, IREC, NP, NE

where

MFLG	Model identifier flag (100-119 for GFGEN results) (120-129 for RMA2 results) (135-139 for RMA2 Hotstart file) (140-149 for RMA4 results) (155-159 for RMA4 hotstart files) (160-169 for SED2D Concentration/DelBed files) (180-189 for SED2D Bed Strata files)
IREC	Version number of the TABS-MD model
NP	Number of nodes in the mesh
NE	Number of elements in the mesh

Record 2

IWRT1, (IBAN(i),i=1,IWRT1)

where

IWRT1	Number of items contained in the banner array
IBAN	Integer interpretation of the banner character strings

Record 3

IWRT2, IWRT3 , (IREC(i),i=1,IWRT2), (FREC(i),i=1,IWRT3)

where

IWRT2	Number of items contained in the IREC array
IWRT3	Number of items contained in the FREC array
IREC	Integer flags which are set during execution
FREC	Floating point flags which are set during execution

Record 4

IWRT4, (ITIT(i),i=1,IWRT4)

where

ITIT	Integer interpretation of the title character string
IWRT	Number of items contained in the ITIT array

Records 5-End are not header records….

The contents of record number 5 through the end varies depending upon the type of file you are dealing with
(u-v-h solution, hotstart,vorticity solution, etc.)

GFGEN Binary Geometry File Format

The contents of each record for the binary geometry file.

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 137.

Record 5

NP, NE, ((CORD(j,k),k=1,2), ALFA(j), AO(j), j=1,N),

((NOP(j,k),k=1,8), IMAT(j), TH(j), NFIXH(j),j=1,M),

Record 6

(WIDTH(j),SS1(j),SS2(j),WIDS(j), j=1,N)

where

NP	Number of nodes in the mesh
NE	Number of elements in the mesh
CORD	Array containing X- and Y-coordinate
ALFA	Array containing
AO	Array containing bottom elevation
NOP	Array containing element connection
IMAT	Array containing each elements material type assignment
TH	Array containing elemental direction for Eddy Viscosity
NFIXH	Array containing elemental direction for Eddy Viscosity
WIDTH	Array containing 1D element, channel bottom width
SS1	Array containing 1D element, left side slope
SS2	Array containing 1D element, right side slope
WIDS	Array containing each element's off-channel storage width
WSCRIT *	Array containing each element's off-channel storage width criticle water surface elevation switch
SSS *	Array containing each elements off-channel storage side slope

Record 7 and 8 (capability added 8 Feb 2002)

IDNOPT, IDM_AUTO

IF (IDNOPT .GT.0) NP, (IDMN(j), AC1(j), AC2(j), AC3(j), AC4(j), j=1,NP)

where

IDNOPT	On/Off switch for marsh porosity
IDM_AUTO	On/Off switch for automatic setting of marsh porosity
NP	Number of nodes in the mesh
IDMN	Flag determines if default or over-ride parameters were used
AC1 – AC4	Marsh porosity parameters for each node.

RMA2 Binary Solution (u,v,h) File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 137.

Record 5

TET, NP, ((VEL(j,k),j=1,3), k=1,NP), (NDRY(k), k=1,NP), NE, (IMAT(k),k=1,NE), (WSEL(k), k=1,NP)

where

TET	Simulation time, in decimal hours
NP	Number of nodes in the mesh
VEL	Array containing --> X-velocity, Y-velocity, and Depth
NDRY	Array containing wet/dry status for each node (1 = wet, 2 = dry, -1 = About to become re-wet)
NE	Number of elements in the mesh
IMAT	Array containing each elements material type assignment
WSEL	Array containing water surface elevation for each node

Records 6 (through the *Last record*)

Continue reading simulation records in a loop until an end-of-file is reached.

RMA4 Binary File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 137.

Record 5

SIMTIM, NQAL, NP, ((TOLD(j,k),j=1,NP), k=1,NQAL),

NE, (IMAT(k),k=1,NE)

where

SIMTIM	Simulation time, in decimal hours
NQAL	Number of quality constituents transported for this simulation
NP	Number of nodes in the mesh
TOLD(j,k)	The concentration (mass/unit-volume) for all nodes *j=1,NP*, for each constituent *k*
IMAT	Array containing each elements material type assignment

Records 6 (through the *Last record*)

Continue reading simulation records in a loop until an end-of-file is reached.

RMA4 Binary Hotstart File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 137.

Record 5

NSTEP, NP, NQAL, ((TOLD(i,j),i=1,NP),j=1,NQAL),

((TDOT(i,j),I=1,NP),j=1,NQAL), (WTEMP(i),I=1,NP)

where

NSTEP	Simulation time step number
NP	Number of nodes in the mesh
NQAL	Number of quality constituents transported for this simulation
TOLD	The concentration (mass/unit-volume) for all nodes *I=1,NP*, for each constituent J
TDOT	
WTEMP	Water temperature for each node

Records 6 (through the *Last record*)

Continue reading simulation records in a loop until an end-of-file is reached.

RMA4 Binary Re-Solve and Buffer File Format

Record 1

LQ, NEC, LCS(MSV), LPS(MSV), PVS(MSV),LHS(NBS), QS(NBS,2)

where

LQ	Number of columns in current matrix
NEC	Number of buffer blocks written
MSV	Parameter variable set during compilation. It is a buffer write control. Always set MSV equal to MND, the maximum number of nodes.
NBS	Parameter variable set during compilation. Maximum solution matrix buffer size
LCS(MSV)	Number of columns
LPS(MSV)	Pivot column
PVS(MSV)	Pivot value
LHS(NBS)	Temporary storage of LHED, the equation number for the current matrix column LQ
QS(NBS,1)	Pivot equation (normalized)
QS(NBS,2)	Pivot column

Record 2 - Last

Continue reading/writing buffer records in a loop until an end-of-file is reached

Recall the technique for estimating the size of (1) buffer write.

Total Words	2 + (3 * MSV) + (3 * NBS)
Total Bytes	Total Words * Number of Bytes per Word

Utilities

FastTABS and SMS

FastTABS is a graphical user interface used (from 1992-1996) for the TABS two-dimensional numerical models. It provides a convenient means of building a mesh and viewing simulation results. The new generation graphical user interface for one- and two- dimensional numerical models is called SMS. SMS began beta testing in 1994 and has been on the market since 1996 and continues to be enhanced for multi dimensions. The SMS version 7x was released in 1999.

FastTABS is a proprietary product of Brigham Young University.

SMS4-6 is a proprietary product of Brigham Young University

SMS7 is a proprietary product of Brigham Young University

Moving RMA4 Binary Solution Across Platforms

Many TABS users work on multiple computer systems; PC's ,UNIX workstations, and mainframe computer systems. These different systems may manipulate binary files with a different word size. This makes it difficult to transfer a TABS binary solution file between these systems.

The solution to this dilemma is to convert the solution file to ASCII format, which can be transferred to any ASCII compliant computer. The new file is then copied to the second computer, and then converted again to binary on that system.

The utility programs **BIN2ASC** and **ASC2BIN** are available for this purpose. To be successful, you will need to compile both programs on both systems involved in the transfer.

Steps For Moving An RMA4 Binary Solution File

1. On the system where the solution file resides, run **BIN2ASC** to obtain an ASCII form of the solution.
2. Copy the ASCII file to the other computer system.

3. Run **ASC2BIN** on the new system to return to solution to a binary form compatible with the new system.

Merging RMA4 Solution Files

The utility program **MERGAVG** using the “merge” option, reads two or more RMA4 solutions created from the same geometry and merges the requested hours of simulation number 1 with simulation number 2, etc. The newly created RMA4 “look-alike” binary solution file can then be conveniently used for post-processing. This utility is useful when multiple dynamic Hotstarts were necessary causing the solution to be spread across multiple files.

This utility may also be used to exclude specified hours from a solution file by merging only one file and requesting only the hours you wish to save.

Obtaining Estimates for Diffusion Coefficients

The utility program **MAKE_EV_DF** provides a means for a modeler to obtain an approximation for the viscosity and diffusion coefficients. MAKE_EV_DF program will examine elemental sizes and velocity values to make initial estimates for the RMA2 eddy viscosity assignments and the RMA4 diffusion coefficient assignements. The output is in a tabular form that can be cut/paste into the run control file(s).

The MAKE_EV_DF program reads either an ASCII or GFGEN binary mesh.. It will optionally assigne a uniform velocity or use a provided RMA2 velocity field to perform the Peclet calculation.

Upon execution of this utility, all input requests are queried and the user supplies the appropriate file names and other requested information.

The output is categorized by material type to provide the appropriate range of values for the input eddy viscosity for RMA2 and/or diffusion coefficients for RMA4.

Example: Input and output sample from MAKE_EV_DF program

```
 *** PROGRAM SETEV - Last Mod: 25 July 2000 ***
            COMPUTES EDDY VISCOSITIES
           and DIFFUSION COEFFICIENTS
   Based on a given goemetry and velocity field

  Is the geometry input ASCII or BINARY?
B
  Enter the BINARY geometry input filename
craney.geobin
 What is the reference elevation for mean water?
0.0
 Enter the filename for the output EDDY VISCOSITIES:
make_ev_df.out
  HOW WILL THE CURRENT VELOCITIES BE ESTIMATED?
      1)  Read from an existing RMA2 solution
      2)  Assumed uniform over the domain
   ENTER EITHER 1 OR 2
1
  Enter hydro file (binary) name
craney.r2sol
  Enter the number of time steps to PROCESS, or
  Enter a negative number to specify a time window
-1
  Enter the minimum and maximum time values
50.0    53.0
```

```
 Check if input geom and hydro units match
 METRIC Flag     =       0
 GFGEN Unit Flag =ENGLISH
 RMA2 Unit Flag  =ENGLISH

 EDDY VISCOSITIES   English(lb-sec/sq-ft)   Max Friction (non-dim)

      IMAT=     1      680.4300     Friction Range=  0.0208 -  0.0340
      IMAT=     2      279.1921     Friction Range=  0.0208 -  0.0795
      skip
      IMAT=     8       23.1908     Friction Range=  0.0208 -  0.1277

 DIFFUSION COEFFICIENTS   English (sq-ft/sec)
      IMAT=     1      350.7371
      IMAT=     2      143.9135
      skip
      IMAT=     8       11.9540

 DIFFUSION COEFFICIENTS   Metric (sq-m/sec)
      IMAT=     1       32.6024
      IMAT=     2       13.3773
      skip
      IMAT=     8        1.1112

 --> Program finished
```

Exterior Curving and Eliminating Bad Boundary Break Angles

The utility program **SLOPEFIX** will aid in curving all exterior element edges. However, inorder for this to be effective, you must curve the mesh geometry prior to running RMA2. See the RMA2 User's Guide.

Obtaining a Summary From a RMA4 Solution File

The utility program **R4_2_SUM**, reads a RMA4 binary solution file and extracts summary information for a list of nodes or for a point nearest a given (x,y) coordinate.

The program requires an input file containing a list of requested locations from which to retrieve concentration information. An example of the two alternatives follows:

TRNXY	1818	-1	-1
TRNXY	-1	4523300.0	873560.

The identifier "TRNXY" must start in column 1 and by completely capitalized. The remaining (data, node, x-coordinate, y-coordinate) on each line is in free field format. The program will extract nodal information if the node value is positive. Or if the node number is unknown, and the coordinates are supplied, the program will extract summary information at the closest node satisfying the supplied coordinates.

Upon execution of this utility, all input requests are queried and the user supplies the appropriate file names and other requested information.

The corresponding RMA2 solution file may be optionally supplied to R4_2_SUM so that the concentration can be analyzed with the corresponding hydrodynamic field.

Technical Support

On-line Support

Technical information is available on the Internet 24 hours a day, 7 days a week under normal operational conditions.

World Wide Web

Check out our list of Frequently Asked Questions on the Internet at URL **http://chl.wes.army.mil/software/tabs/faq**.

E-Mail

If you cannot find a solution to your problem yourself, you can send us e-mail at ***SMS@erdc.usace.army.mil*.** We will try to obtain a solution to your problem and reply with our findings.

TABS Hotline

There are several technical support options available to TABS-MD users. We are continually striving to improve your support options to provide you with the answers you need as simply and as quickly as possible.

Currently there is one person to handle Hotline support phone calls for TABS, so we request that you try all possible avenues to find a solution to your problem before calling for support. If you must call, please have the following pertinent data available:

- Version number and last modification date of all models
- The GFGEN Geometry file (*.geo and *.geobin)
- The RMA2 run control file (*.bc) and binary solution output
- The RMA4 run control file (*.r4bc) and binary solution output
- Computer information: type, memory capacity, operating system, etc.
- A written account of any error messages
- Your phone number, e-mail and FTP addresses if applicable

The Hotline phone number is (601) 634-2730, (601)634-3249. FAX (601) 634-4208.

Glossary of Terms

Advection

The mode of constituent transport that corresponds to the movement of the constituent along with the motion of individual water particles.

Alternate boundary condition file

(Supplemental run control file)

A formatted input file which contains the boundary specifications for an RMA2 run.

ASCII

The **A**merican **S**tandard **C**ode for **I**nformation **I**nterchange 8-bit character set. ASCII values represent letters, digits, special symbols, and other characters.

An ASCII file, or text file, is a file which contains only ASCII characters in the range from 0 to 127.

Aspect ratio

An element's length to width ratio. Long, slender elements with an aspect ratio greater then 15 may cause stability problems.

Banner

An alphanumeric set of information included in all TABS binary output files which describes the flow of data between GFGEN, RMA2, RMA4, and SED2D.

Base to Plan comparisons

The process of identifying differences in numerical model results between existing conditions and revised conditions, usually a change in geometry.

Basin

The entire parcel of land drained by a river and its tributaries.

Batch mode

The opposite of interactive. A job is typically submitted via proclv and is put in a job queue and will execute as CPU time and memory become available on the mainframe computer.

Bathymetry

The submerged bed elevation corresponding to a water body.

Binary

A numbering system consisting of only the numerals 0 and 1.

The TABS system uses some binary files. A binary file permits an efficient means to store numerical results.

Binary files are dependent on the word length of the computer from which they were created. They cannot be directly moved across computer platforms.

A TABS binary file is created by an un-formatted FORTRAN WRITE statement.

See Also:

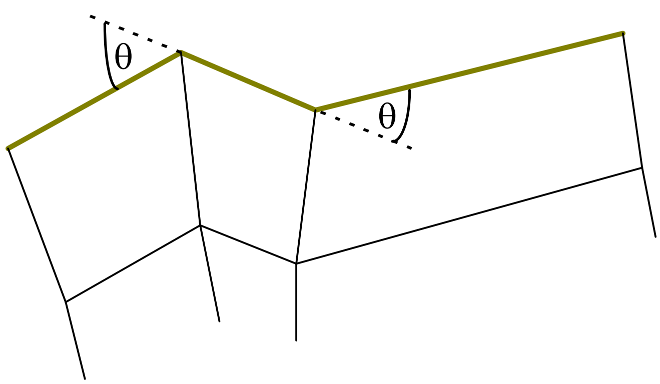
"What Kind Of Computer Do You Have?" on page 31.

Biological Oxygen Demand (BOD)

Sometimes referred to as biological chemical demand. A measure of organic pollution.

Boundary break angle

A boundary break angle is the angle at the node on the "land edge" boundary connecting two elements together. For RMA2 and RMA4, boundary break angles are recommended not to exceed approximately 10 degrees.



Boundary conditions

Water levels, flows, concentrations, stage/discharge relationships, etc., that are specified at the boundaries of the area being modeled. A specified concentration for inflow are typical RMA4 boundary conditions.

See Also:

Boundary effect

A consequence of dissimilarities between the model boundary conditions and the conditions occurring in the prototype at the location of the model boundaries. This effect may be minimized if the model's boundaries are far from the area of interest.

Boundary node

Any node which lies along an *exterior element* edge, or demarcates the wet/dry interface.

Buffer blocks

Temporary files created by RMA2, RMA4, or SED2D, used as virtual memory to store the solution matrix when the BUFFER SIZE dimension value for the matrix arrays is not large enough to hold the matrix. If the model runs to a normal completion, these files are automatically deleted.

Card

A term which comes from the 1960-1980's when computers received data on punched cards. Each card supplied the computer with a line of data.

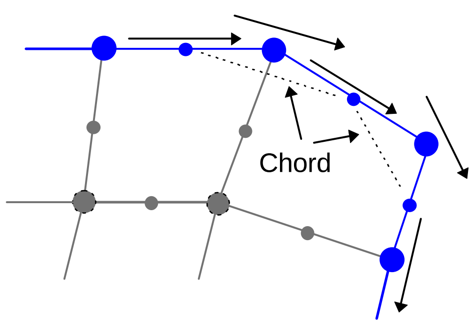
The TABS programs use cards in the same way. The difference is that the card data are stored in a disk file and not in a filing cabinet.

Card image

An ASCII line of data for the computer to read.

Chord

An imaginary line segment connecting two adjacent midside nodes. On a boundary, the flow direction for slip flow (parallel flow) is parallel to this chord.



Coastal and Hydraulics Laboratory (CHL)

The US Army Engineer Research and Development Center, at the Waterways Experiment Station, Coastal Hydraulics Laboratory, Vicksburg, Mississippi, is the principal Corps agency for engineering research and experimentation in hydraulics and hydrodynamics and is one of the largest hydraulics laboratories in the world.

The Coastal Hydraulics Laboratory provides TABS Numerical Model maintenance and support for Army Corps installations. To find out about availability and support for the TABS modeling system, contact the Coastal and Hydraulics Laboratory via e-mail at ***SMS@erdc.usace.army.mil*** or call (601) 634-3339.

Cold start

A model run using initial conditions that are not expected to be close to conditions as solved by the model, i.e., a level water surface elevation and velocity values of zero.

Collocated

Two or more objects placed together at the same location.

Compiler

A special computer program which converts a higher level language (such as FORTRAN) to a coded set of machine dependent instructions (fetch the contents of REGISTER 1). All TABS programs are written for the FORTRAN-77 compiler.

Conservative pollutant

A conservative pollutant does not experience the generation or decay of mass (no gain or loss). For example salt and temperature are treated as conservative constituents in RMA4.

Constituents

Any of several substances whose movement in water can be simulated with an advection/diffusion model. Refers to the pollutants in the flow field (i.e., salt, dye, dissolved oxygen, etc.) that you are attempting to simulate with RMA4.

Continuation card

Used in a run control file. An additional card of the same type as the previous one, used to continue specifying input parameters.

Continuity

The term continuity refers to conserving mass within the model. The continuity check lines are typically used to estimate the flow rates and serve as an error indicator. The RMA2 model satisfied mass conservation in a weighted residual manner. The continuity lines can be used to check mass conservation a different way, by direct integration. Large discrepancies between the two methods indicate probable oscillations and a need to improve model resolution and/or to correct large boundary break angles.

Although continuity checks are optional, they are a valuable tool for diagnosing a converged steady state solution. For steady state, the continuity check lines should represent *total flow in equals total flow out.* However, if the continuity checks indicate a mass conservation discrepancy of ± 3%, you may want to address the resolution in the geometry. Large mass conservation discrepancies can lead to difficulty when the hydrodynamics are used for transport models, RMA4 and/or SED2D.

Continuity line

A string of corner nodes across which the total flow (or constituent if running RMA4) can be measured. The use of continuity lines also provides a convenient way to specify boundary conditions.

Control structure element

A special element with an IMAT ≥ 904. These elements may have 1D or 2D formulation.

See RMA2 User's Guide for more information.

Corner node

Defines a vertex of an element. A point within the mesh that has an (x, y) coordinate and z depth.

If an element has three sides, then it has three corner nodes.

Curved boundary

An optional aesthetic means to outline key landmarks within the computational domain. A quadratic curved side is created by assigning (x, y) coordinates to the mid-side node of an element. Curving the mesh for both the hydrodynamic and transport simulations can help conserve mass. See SLOPEFIX utility.

SMS 8.0

SMS Note: Curved boundaries may be created within the SMS program by unlocking the nodes and moving the mid-side node. Some automatic curving features are available as well.

Curved element side

An optional aesthetic means to outline key landmarks within the computational domain. A quadratic curved side is created by assigning (x, y) coordinates to the mid-side node of an element.

Data Field

A specific location on a record (card in TABS programs) in a data file where a data value occurs.

Datum

The horizontal plane to which soundings, ground elevations, or water surface elevations are referenced.

Some examples of a datum are:

Mean high water
Mean higher high water
Mean low water
Mean lower low water
Mean sea level
Mean tide level
National Geodetic Vertical Datum

Delta time step

The increment of prototype time between two time steps.

Diffusion

The spreading of constituent due to turbulent mixing. A three dimensional spreading action produced by the turbulent flow components.

Dissolved Oxygen (DO)

A measure of the amount of oxygen dissolved in water.

Diurnal tide

A tide with one high water and one low water in a tidal day.

Diverge

The inability of the numerical model to achieve convergence.

Ebb tide

The period of tide between high water and the succeeding low water; a falling tide. Flow exits the estuary during ebb tide.

Eddy

A current of fluid moving contrary to the direction of the main current, especially in a circular motion. A whirlpool motion. A vortex.

Eddy viscosity tensor

The eddy viscosity tensor defines the orientation (relative to the +x axis) of the primary longitudinal eddy viscosity, which is normally aligned with the primary flow direction. In other words, the E_{xx} (x momentum turbulent exchange in the x direction) is applied to the dominant or longitudinal direction of flow. For riverine cases, this angle may be set to the local river axis. This will then allow you to relax the transverse eddy viscosities. For estuarine conditions where it is difficult to define a primary flow direction, this variable is normally ignored and the transverse and longitudinal eddy viscosities are set the same.

Element

A segment, triangle, or quadrilateral shape composed of corner nodes and mid-side nodes. An element must be ‘connected’ to a neighboring element.

An element is composed of a list of nodes in a counterclockwise fashion and may define a 1, 2, or 3 dimensional problem. A line segment defines a one-dimensional area, a triangle or quadrilateral defines a two-dimensional area, while the three dimensional area is defined by adding layers to an element.

RMA2/RMA4 can handle one-dimensional and two-dimensional elements.

Element connection table

The set of GE cards which define the nodes contained in each element

Exit boundary

A boundary condition location at which flow exits the mesh.

Far-field problems

Calculations at the boundaries are not considered important to the RMA2 simulation and are not to be used as accurate data for bankline information. Accuracy in the velocity field increases as you move away from the boundaries. For simulations where bankline conditions are of importance, a three dimensional model which addresses near-field problems should be employed.

FastTABS

A computer program that provides a graphical, point and click means for performing pre- and post-processing for surface water numerical models.

Developed at the Waterways Experiment Station and Brigham Young University. FastTABS was replaced by SMS in 1996.

Fetch length

The distance across which the wind can blow without a land obstruction.

Field data

Data which has been collected at an existing, physical site, used when verifying the simulation.

Finite element

A method of solving the basic governing equations of a numerical model by dividing the spatial domain into elements in each of which the solution of the governing equations is approximated by some continuous function. This method lends itself well to the river/estuarine environments because of its diversity in computational mesh (element size, shape, orientation), flexibility of boundary conditions, and continuity of the solution over the area.

Flood Tide

The period of tide between low water and the succeeding high water; a rising tide. Flow enters the estuary during a flood tide.

Flow fields

The domain in which the water flows.

Free field

A data format where spaces or commas separate the data items on the input line. There is no fixed position in which the data is required to be located. Only the order of the items is important.

Free-surface flow

A fluctuating water surface elevation. A numerical model which can calculate a changing water surface elevation is a free-surface model. Models designated as “rigid lid” do not permit free-surface calculations.

Front width

The number of equations in the numerical model's solution matrix that are assembled simultaneously.

Frontal passages

As in weather; refers to a meteorological storm front.

Froude number

A unitless mathematical expression used to describe a flow field. Froude numbers greater than or equal to 1 are supercritical, less than 1 are subcritical. RMA2 may become numerically unstable with Froude numbers greater than about .6.

The equation for Froude number is

$$F = \frac{V}{\sqrt{gh}}$$

where

F = the Froude number
V = the fluid velocity
g = gravity
h = fluid depth

Galerkin method of weighted residuals

The Galerkin method of weighted residuals is a finite element method which requires that the integral of the residual times the weighting functions should equal zero.

Gate

A movable barrier, such as a tide gate, in a river or stream.

Gauss point

Sampling location within the element used for numerical integration. There are 9 Gauss points for a triangle and 16 Gauss points for a quadrilateral.

GFGEN

Geometry **F**ile **GEN**eration program used to create the computational mesh for all TABS applications.

Gradient

The difference between the bottom elevation of any two corner nodes of an element. Large streamwise depth gradients along short distances are unstable.

Harmonic constituents

The component of tides are usually referred to as harmonic constituents. The principal harmonic constituents of the tide are

Name of the component	Symbol	Period (hrs)

Principal lunar semidiurnal	M_2	12.42
Principal solar semidiurnal	S_2	12.00
Larger lunar elliptic semidiurnal	N_2	12.66
Luni-solar diurnal	K_1	23.93
Lunar diurnal	O_1	25.82

The routine prediction of tides is based upon a simple principle which asserts that for any linear system whose forcing can be decomposed into a sum of harmonic terms of known frequency (or period), the response can also be represented by a sum of harmonics having the same frequencies (or periods) but with different amplitudes and phases from the forcing. The tides are basically such a system.

For the open coastal regions, the tidal prediction capability requires only that you have prior observations of the tides at the station over a suitable period of time from which the amplitudes and phases of the major harmonic constituents can be ascertained. There are typically less than 40 amplitudes and phases for important periods required to reconstitute a tidal signal. Fortunately there are computer programs available which can provide a predicted tidal signal at most every published USGS station location.

Head of tide

The location up river where the tidal signal has been damped so that it is insignificant.

HEC format

A naming convention for the style of run control input derived by Hydrologic Engineering Center (HEC) in which each line of input is defined by a ‘Card Type in data field 0’ and the data follows in fields 1 through n.

Example, GN card with the N option:

Field 0	Field 1	Field 2	Field 3	Field 4
GNN	**node**	**x coordinate**	**y coordinate**	**bottom elevation**

High tide

The maximum elevation reached by each rising tide.

Homogeneous fluid

A fluid which has uniform properties.

Hotstart

The process of supplying the numerical model a set of initial conditions which were obtained from the results of a previous simulation.

Hydraulics Laboratory

See Coastal and Hydraulic Laboratory.

The US Army Corps of Engineers, Waterways Experiment Station, Hydraulics Laboratory, Vicksburg, Mississippi, merged with the Coastal Engineering Research Laboratory in 1996 to form the Coastal and Hydraulics Laboratory (WES-CHL).

The CHL provides TABS Numerical Model maintenance and support for Army Corps installations.

Hydrodynamic

Relates to the specific scientific principles that deal with the motion of fluids and the forces acting on solid bodies immersed in fluids, and in motion relative to them.

Hydrograph

A time series recording of the measurement of flow across a river or stream.

Hydrostatic assumption

The assumption that there is no significant vertical momentum in a given flow field, i.e., vertical pressure is balanced by gravitational forces.

$$\frac{\partial P}{\partial z} + \rho g = 0$$

where

P = pressure

z = water depth

ρ = density

g = gravity

ill-formed elements

Elements which do not conform to the rules for constructing a finite element mesh. See the GFGEN Users Guide for descriptions of ill-formed elements.

IMAT

Material Type. A variable name used in TABS programs to specify a number representing the type of material within an element. The IMAT is located on the GE card and is used to aid in assigning modeling coefficients.

Note: IMAT=0 is equivalent to assigning a land boundary around the element.

Inflow boundary

A boundary condition location at which flow enters the mesh.

Interactive mode

Opposite of Batch mode. The program requires the user to respond to questions.

If the program is running on a mainframe computer, the program is time sharing the CPU with other jobs, which can cause delays in some cases.

Iteration

Repeating a sequence of instructions a specific number of times, changing parameters and obtaining a new solution each time, until a predetermined condition is met.

Junction element

A special 3 to 8 node element which defines the intersection of 3 to 8 one-dimensional elements.

Leaking

A description of the inability of a mesh to properly hold water. Some modelers refer to a 'leak test' as a means to check out a mesh.

Leaks, or "oozes", are a result of poor element shapes, large boundary break angles, and/or erroneous boundary condition specifications.

Logical unit

Computer lingo used to associate a device number with a data file. In this FORTRAN statement, **10** is the logical unit number:

```
READ(10,*) DATA
```

Low tide

The minimum elevation reached by each falling tide.

Magnitude of velocity

A scalar value; the magnitude, *M*, of the resultant velocity vector. It is the square root of the velocity *x* component squared plus the *y* component squared.

$$M = \sqrt{x^2 + y^2}$$

Material type

A number representing the type of material within an element. The associated variable is IMAT.

Mean Absolute Error

A statistical method that represents the magnitude of the difference between two data sets, not the positive or negative directionality of the error.

$$MAE = \frac{1}{n}\sum_1^n \left|h_c - h_o\right|_{time=i}$$

where h_c is calculated and h_o is the observed for a given time=i

Mean high water

(MHW) The average height of the high waters over a 19 year period.

Mean higher high water

(MHHW) The average height of the higher high waters over a 19 year period.

Mean low water

(MLW) The average height of the low waters over a 19 year period.

Mean lower low water

(MLLW) The average height of the lower low waters over a 19 year period.

Mean sea level

(MSL) The average height of the sea surface for all stages of the tide over a 19 year period, usually determined from hourly height readings.

Mean tide level

(MTL) A plane midway between mean high water and mean low water. Not necessarily equal to mean sea level.

Mesh

A collection of nodes and elements which defines the domain of the study area.

Mid-side node

A node between two corner nodes in an element. TABS models require mid-side nodes.

Mild Slope Assumption

In the derivation of the governing equations, the mild slope assumption is applied, which assumes that the normal force at the bottom of the bed is essentially vertical. This also results in the assumption that there is no horizontal component of that normal force to be accounted for in the horizontal momentum equations.

Mixed tide

A type of tide intermediate to those predominantly a semidiurnal tide and those predominantly a diurnal tide.

National Geodetic Vertical Datum

(NGVD) A fixed reference adopted as a standard geodetic datum for elevations determined by leveling. Established in 1929. Also referred to as National Geodetic Vertical Datum of 1929 and Sea Level Datum of 1929. The NGVD is usually preferred as the primary datum for engineering design.

Neap tide

A tide occurring approximately midway between the time of new and full moon. The neap tidal range is usually 10 to 30 percent less than the mean tidal range.

Node

A point containing an x, y, and z coordinate which defines a location in space. Mid-side nodes (x, y, z) are linearly interpolated from adjacent corner nodes, unless the element side is curved.

Non-conservative pollutant

A non-conservative pollutant experiences the generation or decay of mass (gain or loss). It is subject to either chemical or biological change. For example a nitrogen based pollutant is non-conservative.

Null point

A location in a network where there is no net fluid transport (no flow).

Off-channel storage

A 1D and 2D element feature. The storage width and side slopes associated with the node, as specified on GW cards using the N option.

One-dimensional element

A line segment composed of two corner nodes and one mid-side node. The geometry is defined by cross section (a straight bottom line between corner nodes) and reach length. The calculated velocity is averaged over the cross section.

Outfalls

An outfall may be an outlet of a body of water (as a river or lake), or the mouth of a drain or sewer.

Parameter revision

The process of modifying a run control input parameter during a simulation in the middle of a time step. The REV card is used for this purpose.

Peclet number in RMA4

Defines the relationship between element properties, velocity, and diffusion.

$$P = \frac{udx}{D}$$

Plume

An elongated and dynamically changing water quality constituent within a water body.

Pollutants

Anything by intrusion or contact with an outside source that makes the water body impure, inferior or poisoned. For example, water contaminated (polluted) by industrial waste.

PREQUAL

The subroutine within RMA4 that reads the TABS formatted run control file..

Prototype

Field data or physical model data.

The original, or basis for the new study.

Record length

In this context, a FORTRAN specific term dealing with the size of a data type in a binary file.

Residence Time

The time required for a constituent particle to be removed from a given water body. This removal can be a consequence of hydraulic flushing and/or decay (for non-conservative constituents).

Resource Management Associates

The TABS numerical models were initially developed by Dr. Ian King at Resource Management Associates, (RMA), in Lafayette, California. An RMA representative can be reached at (707) 864-2950. Dr. King now resides in Australia.

Rip-rap

Stones, chunks of concrete, or other debris on an embankment slope, or stream side slope, generally used to prevent erosion.

RMA2

The one-dimensional/two-dimensional depth averaged hydrodynamic Finite Element numerical model within TABS.

RMA4

The one-dimensional/two-dimensional depth averaged water constituent transport Finite Element numerical model within TABS.

RMA10

RMA10 was renamed in 2001 to TABS-MDS to reflect the fact that it can model coupled, 3d hydrodynamics, salinity, and/or sediment transport.

RMA10 is a multi-dimensional (combining 1-D, 2-D either depth or laterally averaged, and 3-D elements) finite element numerical model written in FORTRAN-77. It is capable of steady or dynamic simulation of 3-dimensional hydrodynamics, salinity, and sediment transport. It utilizes an unstructured grid and uses a Galerkin-based finite element numerical scheme. The WES-CHL version is based upon the work of Dr. Ian King of Resource Management Associates.

Roughness

In a river or stream bed, the material on the side slopes or the bottom, such as stones, etc., which inhibit the flow.

Run control file

An input data file which provides parameters that control the model simulation run.

SED2D

Originally known as STUDH, SED2D is a two-dimensional depth averaged sediment transport Finite Element numerical model within TABS.

Semidiurnal tide

A tide with two high and two low waters in a tidal day with comparatively little diurnal tide inequality.

SMS

A computer program that provides a graphical, point and click means for performing pre- and post-processing for surface water numerical models. SMS officially replaced FastTABS in 1996.

Developed at the Waterways Experiment Station and Brigham Young University. Corps of Engineers employees may contact the Waterways Experiment Station for more information via e-mail at **SMS@erdc.usace.army.mil**,or call (601) 634-3339

Source code

The US Army Corps of Engineers, Engineering Research and Development Center, at the Waterways Experiment Station, Coastal and Hydraulics Laboratory (CHL), Vicksburg, Mississippi, provides TABS Numerical Model maintenance and support for Army Corps installations. To find out about availability and support for the TABS modeling system, contact the CHL via e-mail at **SMS@erdc.usace.army.mil**, or call (601) 634-3339.

Special elements

Junction element, transition element, or control structure element.

Spin-up

The process by which a model moves from an unrealistic set of initial conditions to more realistic results that represent a solution that is not strongly influenced by the initial conditions.

To estimate spin-up, calculate the time for a wave to travel the length of the mesh and return.

The speed of the wave is calculated as

$$C = \sqrt{gh}$$

where

C = wave speed (celerity)

g = gravity
h = depth

Spring tide

A tide that occurs at or near the time of new or full moon and which rises highest and falls lowest from the mean sea level.

Steady state

A simulation in which the boundary conditions are static. The variables being investigated do not change with time. RMA2 considers the steady state simulation time as hour zero.

Storm passages

A significant meteorological event that passes over the domain of the model.

STREMR

A depth averaged finite difference hydrodynamic numerical model developed by Bernard and Schneider of WES.

Subcritical

Froude number < 1

Supercritical

Froude number ≥ 1

System International

(SI) Formally named in 1960 by an international general conference on weights and measures. This system provides exact definitions of the metric system units for the fields of science and industry.

TABS

The TABS-MD Modeling System is comprised of four main programs: GFGEN, RMA2, RMA4, and SED2D.

TABS-MDS

RMA10 was renamed in 2001 to TABS-MDS to reflect the fact that it can model coupled, 3d hydrodynamics, salinity, and/or sediment transport. It is written in FORTRAN-90.

Tailwater

The water surface elevation at the exit boundary.

Thalweg

The line representing the deepest part of a river or channel.

Tidal day

The time of the rotation of the earth with respect to the moon, approximately 24 hours and 50 minutes.

Tidal range

The difference in height between consecutive high and low (or higher high and lower low) waters.

Tide

The periodic variation in the surface level of the oceans and of bays, gulfs, inlets, and estuaries, caused by gravitational attraction and relative motions of the moon and sun.

The types of tides are:

- Diurnal tide
- Mixed tide
- Neap tide
- Semidiurnal tide
- Spring tide

Transition element

A special 'T' shaped 5 node element which makes the transition between a one-dimensional element and a two-dimensional element.

Turbulence

In a turbulent motion, the true velocity and pressure vary in a disorderly manner. A turbulent motion is always unsteady, since at a given point the velocity changes continuously in a very irregular way.

Turbulent exchange coefficient

A frequently applied approximation is derived from the assumption that the Reynolds stress varies linearly with the gradient of the time-averaged velocities. Under this theory, the stresses caused by random turbulent motions are analogous to Newton's law of viscosity for viscous stresses arising from molecular motions. This approximation gives rise to the turbulent coefficient of viscosity, also called eddy viscosity.

Two delta-X

A numerical instability which presents itself as a high value followed by a low value followed by a high value at the corners of the elements. When contoured, a two delta-X oscillation looks like a case of mesh measles.

Two-dimensional element

A triangle (3 corners and 3 mid-side nodes) or quadrilateral (4 corners and 4 mid-side nodes) shape which defines the geometry in two space coordinates and averages over the third space coordinate. In a two-dimensional *Horizontal* model, the averaging occurs over depth. In a two-dimensional *Vertical* model, the averaging occurs over width. Several two-dimensional horizontal elements aligned side by side may accurately define the bottom elevation of a navigation channel.

Velocity field

In this context, the term velocity field represents the nonal values of velocity and water depth used for the various time steps of the transport model. Typically the velocity field is the hydrodynamic solution created by the RMA2 model.

Verification

The process by which we gain confidence in the ability of our model to predict behavior of the prototype. Field data, like the model results, are only an approximation of reality and must be treated with skepticism. In verifying RMA2, the primary adjustments to be made are to the geometry, boundary conditions, roughness, and eddy viscosity. In verifying RMA4, the primary adjustments are boundary conditions and diffusion coefficients. If geometry revisions are required, then the RMA2 hydrodynamic model must be re-run. These adjustments are made interactively until the model agrees satisfactorily with field (prototype) observations.

Vertically homogeneous fluid

Refers to the similarity of the flow properties within the water column.

Viscosity

A proportionality constant relating fluid stress to the rate of strain. The degree to which a fluid resists flow under an applied force.

Vorticity

A measure of fluid rotation. Fluid flow involving rotation about an axis. A spiral motion of fluid within a limited area, especially a whirling mass of water or air that sucks everything near it toward its center.

Water column

An elemental projection in the *z* direction. The water profile from the surface to the bottom of the water body.

Waterways Experiment Station

The US Engineering Research and Development Center (ERDC), at the Waterways Experiment Station (WES), located in Vicksburg, Mississippi, is the principal research, testing, and development facility of the ERDC. Its mission is to conceive, plan, study, and execute engineering investigations and research and development studies in support of civil and military missions of the Chief of Engineers and other federal agencies.

ERDC at WES is composed of the following laboratories:

Coastal and Hydraulics Laboratory

NOTE: Hydraulics Laboratory and Coastal Engineering Research Center Merged, Aug 1996.

Geotechnical and Structures Laboratory

NOTE: Geotechnical Laboratory and Structures Laboratory Merged, Feb 2000.

Environmental Laboratory

Information Technology Laboratory

Weir

An obstruction placed in a stream, diverting the water through a prepared aperture for measuring the rate of flow.

Well formed element

An element with the proper aspect ratio, shape, angle, plane, and depth variation along an element (gradient).

Word size

A Computer term. A word is made up of a group of bytes. A system's word size is defined by the number of bytes necessary to make a word on that particular type of computer system. For example, a typical PC uses a two byte word (16 bits), where the Cray YM-P uses an eight byte word (64 bits).

Index

E

F

G

H

I

J

L

M

N

O

P

R

S

T

U

V

W

X

Y

Notes

References

1. Brigham Young University, 1994, "FastTABS 3.0 Hydrodynamic Modeling Reference Manual," Engineering Graphics Laboratory, Provo, Utah.

2. Brigham Young University, 1995, "SMS Hydrodynamic Modeling Reference Manual," Engineering Graphics Laboratory, Provo, Utah.

3. Chu, C. R. and Jirka,G. H. (2003) “Wind and Stream Flow Induced Reaeration” ASCE Journal of Hydraulic Engineering Vol. 129, No, 12.

4. Governing Equations for the Mathematical Model RMA4, Appendix H-A, Ian P. King, Richard R. Rachiele, Resource Management Associates, January 1989.

5. Program Documentation: RMA4 - A two Dimensional Finite Element Water Quality Model, Version 3.0, Ian P. King, Richard R. Rachiele,Resource Management Associates, January 1989.

6. SMS 9.0, Tutorials, Environmental Modeling Research Laboratory, Brigham Young University, July 1, 2005.

7. Users Guide to GFGEN Version 4.35, US Army, Engineer Research and Development Center, Waterways Experiment Station, Coastal and Hydraulics Laboratory, May 2001.

8. Users Guide to RMA2 WES Version 4.5, US Army, Engineer Research and Development Center, Waterways Experiment Station, Coastal and Hydraulics Laboratory, July 2005.

9. Users Guide to SED2D WES Version 4.5, US Army, Engineer Research and Development Center, Waterways Experiment Station, Coastal and Hydraulics Laboratory, March 2003, Draft