Users Guide To

RMA2 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.
400 Columbus Avenue
Valhalla, NY 10595
(914)741-9700
1-800-WEXTECH

FastTABS (1990-1994), SMS 5 and 6 (1994-1999), SMS7 (2000-present) is a product of the Engineering Computer Graphics Laboratory, Brigham Young University.

Users Guide To RMA2 WES Version 4.5

Last document modification date: May 22, 2001

Effective RMA2 source code modification date: May 04, 2001

Editor:	*Barbara P. Donnell*
Web Layout Editor:	W. *Clay LaHatte*
Original Author:	*Dr. Ian King*
Contributing Authors:	*Barbara P. Donnell, Joseph V. Letter,Jr., Dr. William. H. McAnally, Jr., and William A. Thomas*
Source Code Authors:	*Joseph V. Letter, Jr., Barbara P. Donnell, Dr. John I. Finnie, Dr. Ian King, and Dr. Lisa C. Roig.*
Technical Assistance:	*Dr. R. C. Berger, and Gary L. Brown.*

Technical Support

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

E-mail: *tabs@hl.wes.army.mil*

Cataloging Information

Donnell, Barbara P., Letter, Joseph V., McAnally, W. H., and others, "User's Guide for RMA2 Version 4.5," [02 Mar]2001, [http://chl.wes.army.mil/software/tabs/docs.htp].

Contents

Overview

What Is RMA2

RMA2 is a two-dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface two-dimensional flow fields.

RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning's or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady (dynamic) problems can be analyzed.

Origin Of The Program

The original RMA2 was developed by Norton, King and Orlob (1973), of Water Resources Engineers, for the Walla Walla District, Corps of Engineers, and delivered in 1973. Further development, particularly of the marsh porosity option, was carried out by King and Roig at the University of California, Davis. Subsequent enhancements have been made by King and Norton, of Resource Management Associates (RMA), and by the USA ERDC at the Waterways Experiment Station (WES) Coastal and Hydraulics Laboratory, culminating in the current version of the code supported in TABS-MD.

Applications For RMA2

The program has been applied to calculate water levels and flow distribution around islands, flow at bridges having one or more relief openings, in contracting and expanding reaches, into and out of off-channel hydropower plants, at river junctions, and into and out of pumping plant channels, circulation and transport in water bodies with wetlands, and general water levels and flow patterns in rivers, reservoirs, and estuaries.

Capabilities Of RMA2

RMA2 is a general-purpose model designed for far-field problems in which vertical accelerations are negligible and velocity vectors generally point in the same direction over the entire depth of the water column at any instant of time. It expects a vertically homogeneous fluid with a free surface.

RMA2 has these capabilities:

- Identify errors in the computational mesh specification.
- Accept either English or standard SI units. (version 4.27 or higher)
- Restart (Hotstart) the simulation from a prior RMA2 run and continue.
- Simulate wetting and drying events.
  - Provide for a more accurate account of off-channel storage
  - Adjust for wetting and drying by element.
  - Account for Marsh Porosity wetting and drying (wetlands).
- Account for effects of the earth's rotation (Coriolis).
- Apply wind stress either
  - Uniformly over the model domain; constant or time-varying.
  - As a storm; front or tropical cyclonic event (time-varying)
- Employ either direct or automatic dynamic assignment of Manning's n-value by depth (version 4.28 or higher).
- Employ user selectable manual or automatically assigned turbulent exchange coefficients
  - Direct assignment methods
  - Peclet method for automatic dynamic assignment of turbulent exchange coefficients (version 4.28 or higher).
  - Smagorinski method for automatic dynamic assignment of turbulent exchange coefficients (version 4.42 or higher)
- Model up to 5 different types of 1D flow control structures (2D structures are permitted in version 4.50 or higher).
- Compute flow across continuity check lines.
- Provides for user defined computational controls; such as:
  - Wet/dry parameters
  - Iteration controls
  - Revisions within a time step (both coefficients and/or boundary conditions)
- Accepts a wide variety of boundary conditions.
  - Angle by node
  - Velocity components by node
  - Water surface elevations by node/or line *
  - Discharge by node/element/ or line
  - Tidal radiation boundary conditions by line
  - Discharge as a function of elevation by line
  - Wind speed and direction by node/element or element material type
  - Read an *"STWAVE"* radiation stress file to incorporate wave induced currents

* By "line" means that over the user defined continuity line (GC type card), the nodes on that line will have that specification applied.

Limitations Of RMA2

RMA2 operates under the hydrostatic assumption; meaning accelerations in the vertical direction are negligible. It is two-dimensional in the horizontal plane. It is not intended to be used for near field problems where vortices, vibrations, or vertical accelerations are of primary interest. Vertically stratified flow effects are beyond the capabilities of RMA2.

RMA2 is a free-surface calculation model for subcritical flow problems. More complex flows where vertical variations of variables are important should be evaluated using a three-dimensional model, such as RMA10.

Governing Equations

The generalized computer program RMA2 solves the depth-integrated equations of fluid mass and momentum conservation in two horizontal directions. The forms of the solved equations are

Equation 1

$$h\frac{\partial u}{\partial t}+hu\frac{\partial u}{\partial x}+hv\frac{\partial u}{\partial y}-\frac{h}{p}\left[\mathrm{E}_{xx}\frac{\partial^2 u}{\partial x^2}+\mathrm{E}_{xy}\frac{\partial^2 u}{\partial y^2}\right]$$
$$+gh\left[\frac{\partial a}{\partial x}+\frac{\partial h}{\partial x}\right]+\frac{gun^2}{\left(1.486h^{1/6}\right)^2}\left(u^2+v^2\right)^{1/2}$$
$$-\zeta V_a^2\cos\psi-2hv\omega\sin\Phi=0$$

Equation 2

$$h\frac{\partial v}{\partial t}+hu\frac{\partial v}{\partial x}+hv\frac{\partial v}{\partial y}-\frac{h}{p}\left[\mathrm{E}_{yx}\frac{\partial^2 v}{\partial x^2}+\mathrm{E}_{yy}\frac{\partial^2 v}{\partial y^2}\right]$$
$$+gh\left[\frac{\partial a}{\partial y}+\frac{\partial h}{\partial y}\right]+\frac{gvn^2}{\left(1.486h^{1/6}\right)^2}\left(u^2+v^2\right)^{1/2}$$
$$-\zeta V_a^2\sin\psi+2hu\omega\sin\Phi=0$$

Equation 3

$$\frac{\partial h}{\partial t}+h\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)+u\frac{\partial u}{\partial x}+v\frac{\partial h}{\partial y}=0$$

where

h = Water depth

u,v = Velocities in the Cartesian directions

x,y,t = Cartesian coordinates and time

ρ = Density of fluid

E = Eddy viscosity coefficient,

for xx = normal direction on x axis surface

for yy = normal direction on y axis surface

for xy and yx = shear direction on each surface

g = Acceleration due to gravity

a = Elevation of bottom

n = Manning's roughness n-value

1.486 = Conversion from SI (metric) to non-SI units

ζ = Empirical wind shear coefficient

V_a = Wind speed

ψ = Wind direction

ω = Rate of earth's angular rotation

Φ = Local latitude

Equations 1, 2, and 3 are solved by the finite element method using the Galerkin Method of weighted residuals. The elements may be one-dimensional channel reaches, or two-dimensional quadrilaterals or triangles, and may have curved (parabolic) sides. The shape (or basis) functions are quadratic for velocity and linear for depth. Integration in space is performed by Gaussian integration. Derivatives in time are replaced by a nonlinear finite difference approximation. Variables are assumed to vary over each time interval in the form

Equation 4

$$f(t) = f(t_0) + at + bt^c \qquad t_0 \le t < t_0 + \Delta t$$

which is differentiated with respect to time, and cast in finite difference form. Letters *a*, *b*, and *c* are constants.

Note: It has been found by experiment that the best value for *c* is 1.5 (Norton and King 1977).

The solution is fully implicit and the set of simultaneous equations is solved by Newton-Raphson non linear iteration scheme. The computer code executes the solution by means of a front-type solver, which assembles a portion of the matrix and solves it before assembling the next portion of the matrix.

The current version of RMA2 is based on the earlier versions (Norton and King 1977) but differs in several ways.

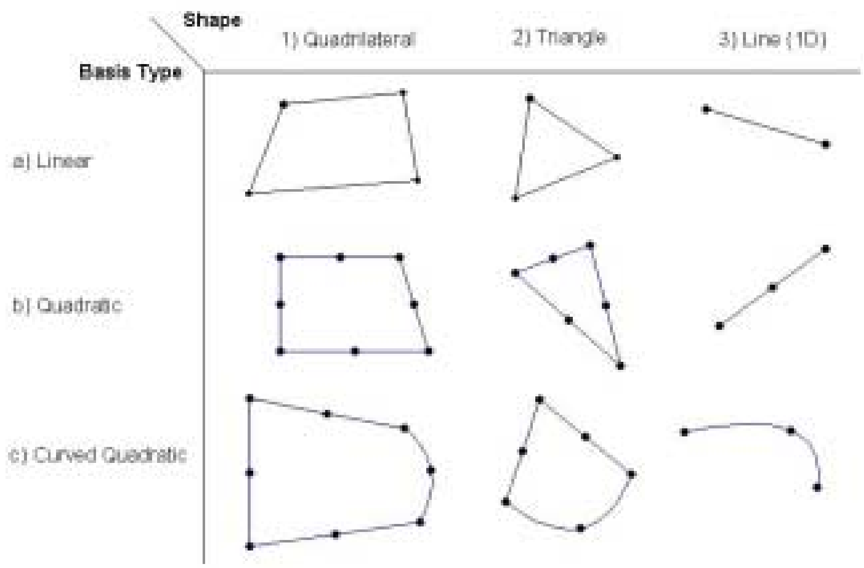
- It is formulated in terms of velocity (**v**) instead of unit discharge (**v**h), which improves some aspects of the code's behavior.
- Employs new numerical solution algorithms.
- Permits wetting and drying of areas within the mesh.
- Permits wetlands to be simulated as either triggered totally wet/dry or as gradually changing between wet/dry states.
- Permits specification of turbulent coefficients in directions other than along the *x* and *y* axes.

- Accommodates the specifications of hydraulic control structures in the network.
- Permits the use of automatic assignment of friction and turbulent coefficients.
- Permits input in either English or System International units.
- Permits a correction term for the effects of secondary currents by solving for the stream-wise vorticity.

Element Types Supported

The figure below is designed to introduce some nomenclature. The contents of columns one and two are all two-dimensional elements (2D) where the variables are depth-averaged. The last column depicts one-dimensional elements (1D) where the computed variables are averaged over the cross-section (depth and width).

Row a) of the figure below are all linear basis elements shaped as a 2D quadralateral, 2D triangle and 1D line. Linear basis elements are composed of corner nodes only and are an expediant way to initially construct a mesh. However, the RMA2 model only supports quadratic basis elements as shown in rows b) and c). These elements have both corner and mid-side nodes. Row b) are all straight-sided quadratic basis elements, in the form of a 2D quadralateral and 2D triangle, and 1D line element. Row c) is the same as row b), except that one side is curved (isoparametric).



RMA2 is capable of supporting different types of quadratic basis elements within the same computational finite element mesh. The types of elements fit into three basic categories:

- Two-Dimensional Elements
- One-Dimensional Elements
- Special Elements

Note: Element edges may be either straight or curved.

Two-Dimensional Elements

Two-dimensional quadratic elements are the primary type used with RMA2 and may be either triangular (3 corners and 3 midside nodes) or rectangular (4 corners and 4 midside nodes) in shape.

A two-dimensional quadratic element possesses a length and a width, determined by the positions of the corner nodes which define the element. The water depth at any location within a two-dimensional element is obtained by interpolating among the depths of the corner nodes which define the element.

Rectangular Element   Triangular Element

One-Dimensional Elements

A one-dimensional quadratic 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. One-dimensional element characteristics are defined using GN and GW cards with the N option. See "Using Special Elements" on page 71 for more information.

SMS Note: As of this writing, in order to allow SMS to recognize one-dimensional element widths and side slopes, you must use a GN card with the N option to specify the *x* and *y* coordinates and the bottom elevation, and a GW card with the N option to specify the surface width, and other 1D specific information. Also note that SMS may not permit a mesh composed entirely of one-dimensional elements because it cannot distinguish between a 3-noded linear triangle and a 3-noded 1D element.

SMS Tip: If the version of SMS does not permit a fully 1D mesh, creating a single quadratic triangle with an IMAT of zero will allow SMS to operate.

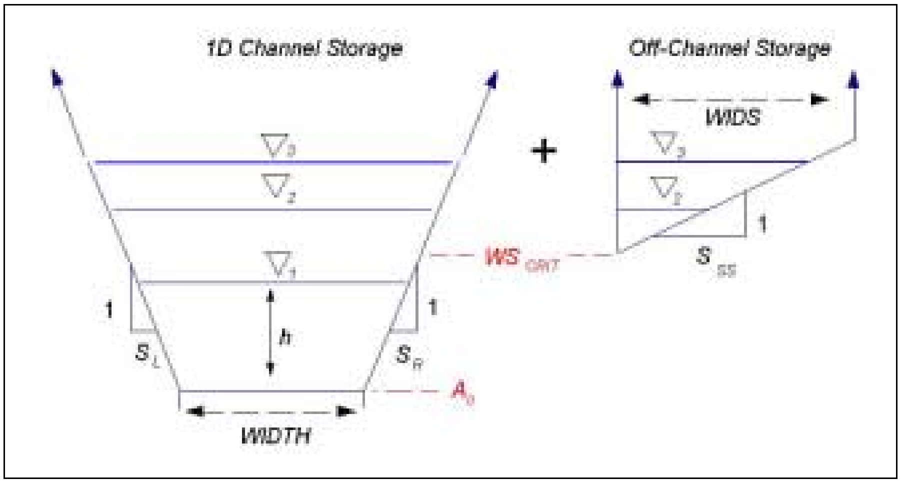
Off-Channel Storage for One-Dimensional Element

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

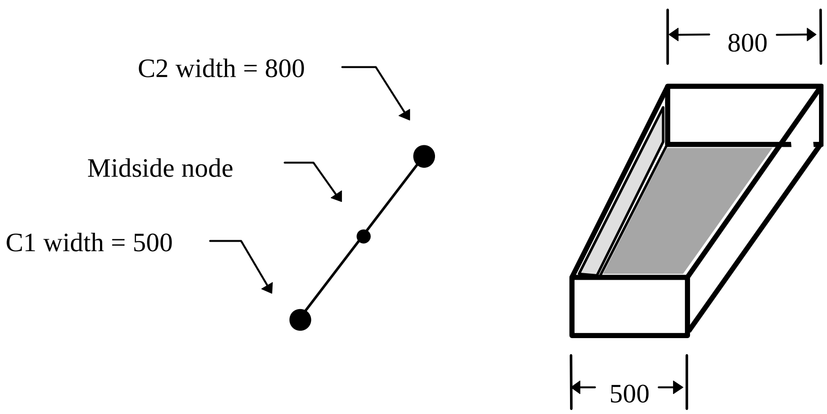
The numerical model's Governing Equations for one-dimensional elements are based on a trapezoidal cross-section with side slopes and off-channel storage. To describe the trapezoidal cross-section, you must assign for each one-dimensional corner node, a bottom elevation, a bottom width (when the depth=0), a left and right bank slope (S_L and S_R). The designation of left and right bank slope is arbitrary. All slopes reference a given distance for one unit of rise. 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 needed, additional off-channel storage contributions are 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). The earlier versions of off-channel storage functioned as if S_{SS}>>WIDS and $WS_{CRIT}<A_o$, where A_o is the bottom elevation.

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

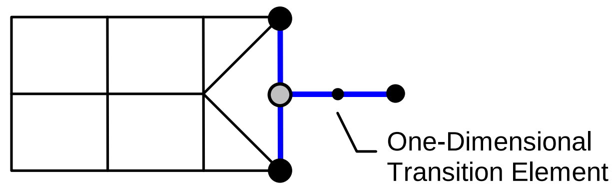
Some one-dimensional elements serve special purposes. These elements fall into three categories:

- Transition Elements
- Junction Elements
- Control Structure Elements

See "Using Special Elements" on page 71 for more information on these special elements.

Transition Elements

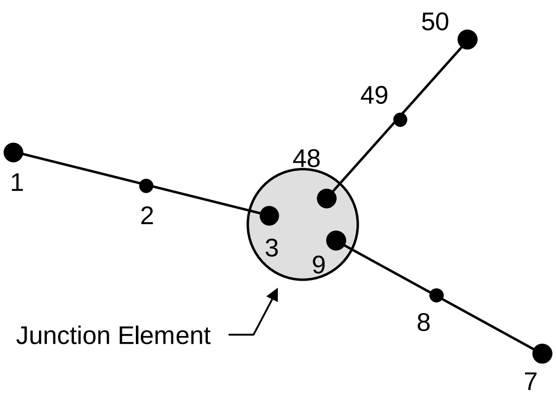
A transition element is required anytime you want to use both one-dimensional and two-dimensional elements in your mesh. This "T-*shaped"* type of one-dimensional element makes the transition between the two-dimensional elements and the one-dimensional elements.



For additional information on Transition elements, see "Using One-dimensional Transition Elements" on page 72, 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 0 901 ≤ imat ≤903

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

Note: The functional limit of the number of elements entering a junction is 8, as dictated by the number of nodes in an element, although it is not advised to use more than 4.

For additional information on Junction elements, see "Using One-dimensional Junction Elements" on page 73, or the GFGEN reference manual.

Types of Junction Elements

At junction elements, the water mass fluxes are balanced and either: all water levels match, all total energy heads match, or momentum is conserved at the junction. The three categories of junction elements are controlled by the material type designation:

- 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. For the first 2 nodes that define the primary channel, it is recommended that the bed elevations and widths be the same.

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 of structure (FC card).

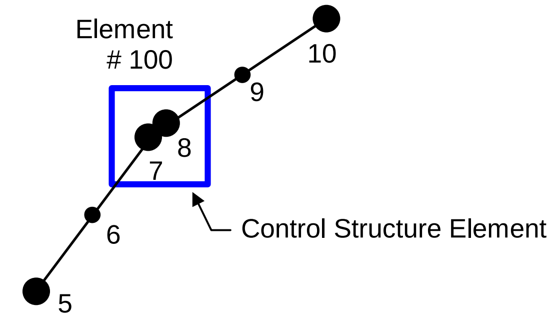
Types Of Control Structure Elements

There are several types of control structures elements:

- Type 1, Point source or sink 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. This order sets the “orientation” of the flow through the structure.



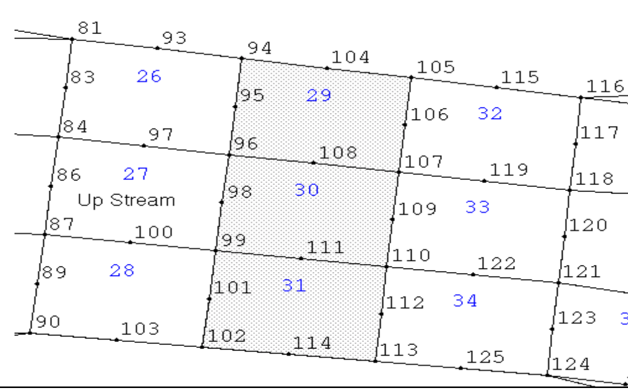
GE 100 7 8 0 0 0 0 0 imat ≥<u>904</u>

<u>2D Control Structure</u>

Two-dimensional Control Structure elements have a unique connection table and has an IMAT value ≥ 904.

At the time of this writing, SMS versions prior to 6.08 could not create the 2D control structure. The 2D control structure modeling technique, followed by an example problem, is presented in the section entitled "2D Control Structure Example" on page 77.

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 downstream edge must fall in positions 5-7 of the element connection table. The user must verify that this was done correctly before proceeding. The midside 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 to bank. To convert these elements to control structure elements, the placeholders for midside nodes 104, 108, 111, and 114, are filled with zeros. In this example, the control structures was 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**

The selection of the type of control structure (FC Card, NJT variable) is important. It may be useful to use a calculator or spreadsheet to check the specification for reasonableness prior to running RMA2. 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 per unit width.

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

Curved Element Edges

Although it is no longer a requirement to prescribe curved (isoperimetric) external boundaries to prevent 'leaking' (releases prior to RMA2 version 4.20), they may be used to achieve aesthetics when viewing the mesh, add length without additional resolution, and to aid in mass conservation when the RMA2 results are to be used for transport applications.

See "Curving Element Edges" on page 89 for more information.

Note: It is not advised to curve interior element edges. SMS will display curved elements if *the curved mesh edges* are activated in the *mesh display options*.

Tip: Curving is particularly useful for one-dimensional elements to achieve the meandering/snaking river effect.

What's New And What's Gone

During the process of writing this manual, the RMA2 source code has been reviewed and revised many times. As a result, many code errors have been eliminated, and some input procedures have been added or changed in an attempt to make the use of RMA2 more straightforward.

Changes since version 4.28

$L Card Changes

The $L card now has ten variables. The last 3 variables are new in this release. They provide input/output (IO) control for diagnostic output when running with automatic features, diagnostics for vorticity transport ("the bend-way correction"), and IO control for the STWAVE wave model results input file.

BQ Card Changes

The BQ card with the L option now provides the capability to specify the flow distribution weighted by water depth across a channel (QXP variable).

GC Card Changes

Use of the GC card will result in a warning stating that the GCL card is the required method to define continuity check lines. The continuity line number is now defined at the start of the GCL card, and not by the card order, as is the case with GC cards. This method is consistent with specification used in RMA4 and SED2D.

GY Card Renamed to GZ Card

The GY card used to specify bottom elevations. It has been renamed as a GZ Card.

TR Card Changes

The on/off switch for the summary results listing file (ISPRT variable) was eliminated from the TR card. Use the $L card for this feature.

Additions

Bendway Correction (Vorticity) Added

An improved method of calculating velocities around curved channels has been incorporated into RMA2 version 4.35 or higher. New data cards were added to accomplish this, and are the BV card, TS card, TV card, and VO card. See "Bendway Correction (Vorticity)" on page 93 for more information.

Multi-Record Hotstart Added

Effective February 1998, with version 4.35 of RMA2, the ability to save/read multiple hotstart records was made available.

GCL Card Added

The GCL card is a replacement for the GC card used to define continuity line numbers. The continuity line number is now defined at the start of the GCL card, and not by the card order, as is the case with GC cards.

GZ Card Added

The GZ card directly replaces the GY card for specifying bottom elevations. The GY card was simply renamed.

RA Card Added - Rainfall And Evaporation

A simple means to include rainfall and evaporation in a simulation was added to RMA2. The RA card is used to supply the parameters for this new method. See "Adding Rainfall And Evaporation" on page 110 for more details.

SM Card Added – Smagorinski Method

An additional turbulence closure methodology, called Smagorinski, was made available to RMA2 versions 4.42 and higher. The Smagorinski option will automatically set and adjust the eddy viscosity during a simulation. The SM card is used to supply the parameters for this new method. See "Smagorinski Method" on page 102 for discussions on the Smagorinski method.

HW Card Added – Wave Radiation Stress

A means to include the effects of wave action on driving circulation in a simulation was added to RMA2 version 4.42 and higher. The HW card is used to supply the parameters to provide control over the wave height and wave period read from the existing STWAVE wave model. See “Applying Wave Radiation Stress” on page 111 for discussions on using a wave field in RMA2.

Off-Channel Storage for 2D Added

Effective April 2000, with version 4.42 of RMA2, the capability to more accurately describe the off-channel storage was added.

New Flow Control Structures Added

Effective January 2000, with version 4.50 of RMA2, the model was enhanced to handle 2D control structures. New flow control structure types are expected to be added.

Deletions

$F Card Deleted

The $F card for fixed data field format is no longer supported.

G1 Card Deleted

The G1 card is no longer supported. Use instead the GS, LA, and BCC card.

Significant Corrections

Marsh Porosity and Wind

Effective 15 Dec 1999, with version 4.45 of RMA2, the model was modifed to turn off the wind stress on shallow active marsh porosity elements.

System Requirements

Because RMA2 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. The ideal situation is to have sufficient RAM to put the entire problem in memory and avoid writing temporary buffer files. For instance, you may be able to squeeze a 15000-element simulation into 32 Mbytes of RAM if you have sufficient disk space to store many large buffer files, but the simulation would execute slowly due to the added I-O burden. A better scenario, with faster turn around, is to have a gigabyte of memory paired with a large disk, to allow the problem to be solved in memory.

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

Note: RMA2 will stop if the problem size is too large and provide a warning. The RMA2 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.

In addition to RAM, the processing power, or speed of your system is also an important consideration when using RMA2. RMA2 can require a large amount of computational power during a simulation, and can take a very long time to complete the run. To be most efficient, try to use a system with a good, powerful processor.

In some cases, machine precision may become an issue. Different types of computer systems can have different word lengths, and this word length can affect precision. If you suspect RMA2 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 199.

Personal Computer Systems

The most commonly used personal computer systems on which RMA2 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 or better 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 RMA2, 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 RMA2 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 RMA2. Also, be aware that RMA2 may run much slower with this configuration than if your system actually had enough physical RAM.

Macintosh

Requires a 68030 or better CPU, 4 Mbytes of RAM, and 20 Mbytes of free disk space.

Mini Computer Systems (Workstations)

Workstations are a good choice for running RMA2. They provide a stable platform without the computational costs associated with many mainframe computer systems. Today's workstations are normally shipped with at least 32 Mbytes of memory, and will usually not have a problem running RMA2. Be sure that you have at least 20 Mbytes of free disk space for installation and minimal execution.

Mainframe/Supercomputer Systems

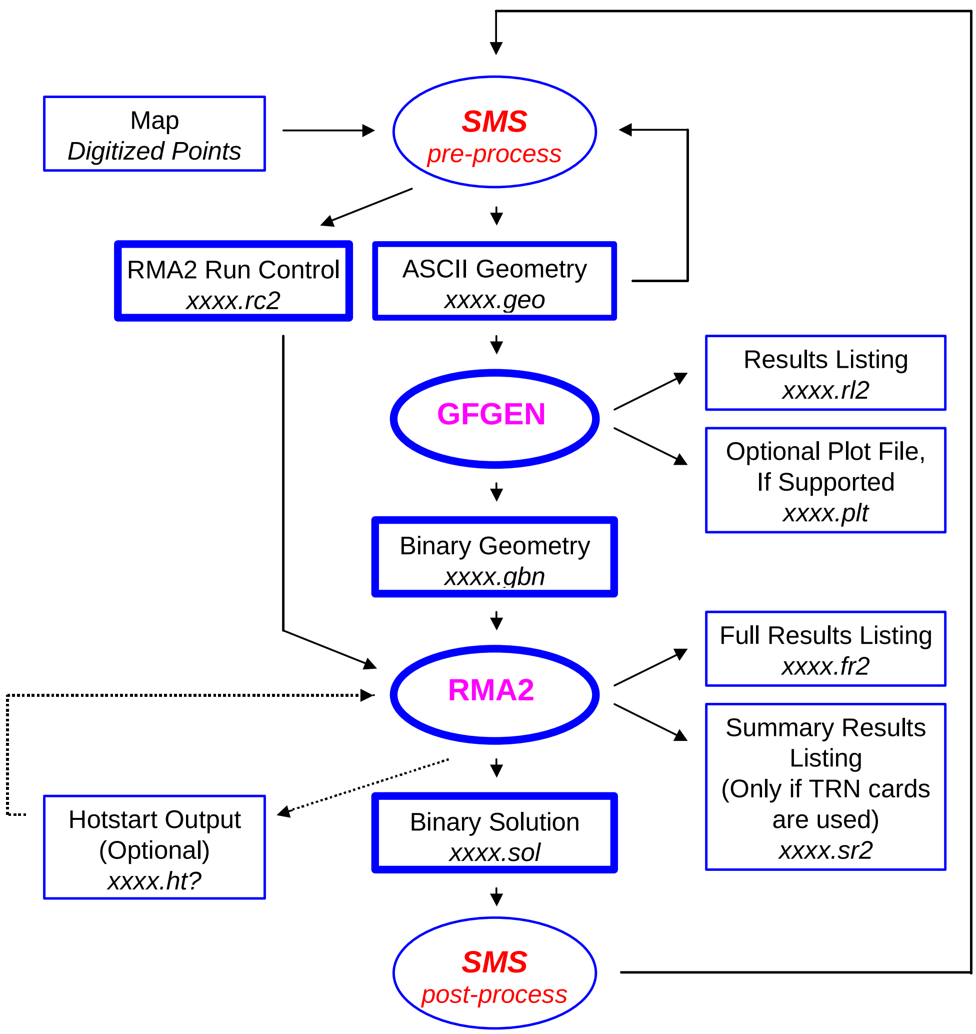
If you have access to one, a mainframe, or supercomputer may be very useful when running RMA2. Many are faster and use more precision than do some workstations.

Be sure, however, to have enough available disk space, at least 20 Mbytes, allotted to you for installation and storing RMA2 run output.

Using RMA2

The RMA2 Modeling Process

The following flow chart illustrates the RMA2 modeling process. Items with bold borders are required, others are optional.



Modes of Operation

At present there are three modes of operation for running RMA2. These modes are:

- Running In Interactive Mode
- Running In Batch Mode
- Running In Auto Mode

Typically RMA2 is set to automatic, simulation super file, mode for compatibility with simulation super file saved by SMS.

The mode of operation 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. Typically RMA2 is set to batch mode for mainframe computer environments and to interactive mode in PC and workstation environments. Automatic mode is designed for SMS simulation super file compatibility.

Running In Interactive Mode

To run in interactive mode, execute RMA2 directly and answer the series of questions referring to input and output file names. For instructions and help, enter a '?'.

The figure below illustrates the startup procedure for RMA2 in interactive mode. Note that the question regarding a particular file will only be asked if that file is active on the $L card. You may choose to null or prevent that file from being opened by entering a '**null**' in response to the file name. The root file name is typically tied to the project and the extension corresponds to the type of file. In this example, "*site*" is used as the project name.

The listing shown is called the *screen* output file. The file name prompts appear on the screen (computer monitor) by default. The screen output file is temporary and may be viewed to the extent of the scroll settings on your computer. If the screen has been inadvertently redirected (>), no prompts will be visible on the screen.

For information on RMA2 input and output file requirements, see the section "Data Files" on page 25.

Some users tire of prompting and prefer to run in *semi-interactive mode* . When running in semi-interactive mode, the responses to the file prompts are included in a directed input file and the screen output file is redirected to a permanent file name. The actual command to run the RMA2 model using the direct (<) and re-direct (>) commands, is shown below.

RMA2v452 < *site.run* > site.screen

where the contents of *site.run* are the file names in the same order as the interactive prompting.

Example: Screen output from an Interactive RMA2 Session.

```
  RMA2 VERSION 4.52   1D & 2D CAPABILITY.      ← Version identifier
  LAST MODIFICATION DATE:  05-02-2000          ← Last Modified

  THIS EXECUTABLE IS DIMENSIONED AS FOLLOWS:
     MAX NO. OF NODES                          1000   ← Dimensions
     MAX NO. OF ELEMENTS                        400
     MAX NO. OF EQUATIONS                      2000
     MAX FRONT WIDTH                            120
     MAX NO. OF CONTINUITY CHECK LINES           15
     MAX NO. OF NODES PER CONTINUITY LINE        40
     MAX BUFFER SIZE                           4000
     MAX PRINT-SUMMARY BUFFER                 25000

  ======  RMA2 VERSION 4.52   1D & 2D CAPABILITY.   ======
  ======  LAST MODIFICATION DATE:  05-02-2000       ======

  ======  TABS-MD FE HYDRODYANMIC MODEL  ======
  //// Original Author: Ian P. King of RMA ////
  ////  Modified and Maintained by WES-HL  ////

  Enter a --> ?  to receive a response menu     ← File Prompting

 ENTER RUN CONTROL INPUT FILE NAME
site.bc                                        ← User Input

 ENTER FULL PRINT OUTPUT FILE NAME
site.fp


 RMA2 VERSION 4.52  READING INPUT DATA ... UNIT=    2

 NO BANNERS ON INPUT CONTROL FILE ... REWIND
 T1  RMA2 4.5 INPUT DYNAMIC-40 HRS    (208 ELEM-629 NODES)
 T2  Cold start with revision to hr 40
 T3  GENERIC ESTUARY- BASE IMAT1-2 (150-.025) IMAT3 (100-.01)
 $M      4
 $L      0    62    60    64     5     3    59

 ENTER ALTERNATE B-C INPUT FILE
null                                              ← How to Null a file
  NOTE: You just turned off Alternate BC file
 ENTER SPECIAL PRINT OUTPUT FILE
site.sp
 ENTER INPUT GEOMETRY FILE (binary)
site.gbn
 ENTER OUTPUT RESTART/HOTSTART FILE (binary)
site.hot
 ENTER FINAL RMA-2 RESULTS FILE (binary)
site.sol

  --> Just Read GFGEN.. nodes & elements=629,  208
```

Running In Batch Mode

Batch mode is typically used on a large mainframe computer system, such as the Cray T3E at the ERDC-Waterways Experiment Station. On the super computers and other mainframes, the batch mode execution of RMA2 requires computer specific job control language. Since the specific syntax changes frequently, it will not be presented in this document. The ERDC-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.

If you have access to the super computers, or another mainframe computer containing the TABS programs, you may wish to execute RMA2 in batch mode - especially if you have a large problem. Generally, you will have access to more CPU time and more system RAM when executing in batch mode. The table below shows a 1996 example of the differences between available system resources when executing in batch mode vs. interactive mode on the super computers.

Execution Mode	Max CPU Time	Max System RAM
Batch Mode	24 Hours	Multiple Gigabytes
Interactive Mode	10 Minutes	2 Mbytes

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: Different mainframe computer systems have different ways of handling the "*screen file*". Some merge it together with the full results listing (standard out). Be advised, and request job control guidance to specifically address this issue.

Running In Auto Mode (Simulation Super File)

Automatic Super Mode in RMA2 (version 4.35 or higher) was designed to be compatible with the simulation super file concept in SMS (version 6.08 or higher). The advantages of both interactive and batch modes were combined into this mode. The graphical user interface, SMS, saves a simulation super file when the "Save RMA2 Simulation" command is issued.   When RMA2 is compiled in automatic super mode, 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, simply enter the word, "null" (in lower case), for the file name, as shown in the R2_SOL_VOR example.

When RMA2 executes, the only interactive question presented to the user is a request for the name of the simulation super file (*site_rma2.run* in this example). The actual command to run the RMA2 model in this mode could use the direct (<) and re-direct (>) commands, as shown.

RMA2v452 < *site_rma2.run* > site.screen where the contents of *site_rma2.run* are the key word and file names listed in the table below. To deactivate or "null" a file name, simply enter the word null (lower case) in place of the file name.

Auto Mode Key-Word	File Name	Description
FILE_PATHWAY	/disk2/proj/	Sets the pwd for these I/O files
R2_BC	site.bc	RMA2 run control BC file
R2_FRL	site.frl	RMA2 full results listing
GEOMETRY_BIN	site.gbn	GFGEN binary geometry, input to RMA2
R2_HOTIN	site.hot1	Initial Condition Hotstart
R2_HOTOUT	site.hot2	RMA2 binary hotstart output
R2_SOL	site.sol	RMA2 binary solution file
R2_SOL_VOR	null	RMA2 debug vorticity solution (set to null)
R2_SRL	site.sum	RMA2 summary results listing via TRN-Cards
R2_ALTBC	site.altbc	RMA2 alternate BC transient file
R2_PARAMETER	site.param	RMA2 auto Roughness and-or Eddy Viscosity echo
R2_WAVE	site.wave	STWAVE binary wave file, input to RMA2

Running in auto mode is convenient for running SMS, GFGEN, and RMA2. The SMS simulation file contains the names of each of the GFGEN and RMA2 simulation super run file names. Consequently by opening the *site.sim* into SMS, all pertinent files are loaded.

Example of a SMS, GFGEN, and RMA2 simulation super file compatible set. This example assumes the project name to be "site".

SMS Simulation File ***site.sim***	**GFGEN Simulation Super Run File** ***site_gfgen.run***	**RMA2 Simulation Super Run File** ***site_rma2.run***
`TABSSIM`	`GEOMETRY_INP site.geo`	`R2_BC site.bc`
`GFGEN site_gf.run`	`GEOMETRY_FRL site.gfp`	`R2_FRL site.fp2`
`RMA2 site_rma2.run`	`GEOMETRY_BIN site.gbn`	`GEOMETRY_BIN site.gbn`
		`R2_SOL site.sol`
		`R2_SRL site.sum`
		`R2_PARAMETER site.auto`

Guidelines For Obtaining a Good Solution

All aspects of the geometry and the numerical model simulation must run in harmony. In addition to the geometry, the RMA2 run control (boundary condition) file must contain the proper information if the simulation is to be successful. A graphical user interface such as FastTABS or SMS will help you build a run control file, but it is still recommended that you examine that file and double check the run control selections.

Note: The primary requirement for a successful numerical model is preparing a good mesh; developing it with the following recommended guidelines in mind. These guidelines are briefly described below and can be found in detail in the GFGEN geometry manual.

Maintain Good Elemental Properties

Some good element properties:

- Maintain a length to width ratio of less than one to ten.
- Restrict element shapes to avoid highly distorted triangles or rectangles.
- Create elements with corner angles greater than 10 degrees.
- Bathymetric elevations should lie almost in a plane.
- Maintain longitudinal element edge depth changes of less than 20%.

Maintain Good Mesh Properties

- A well constructed mesh must first have good element properties.
- The overall bathymetric contours should be smooth.
- Wetting and drying studies work best when the element edges to lie on bathymetric contours.
- Ideally, any *boundary break angle* should not exceed 10 degrees.
- Neighboring elements should not differ in size by more then 50%.
- Use adequate resolution to model the features of the prototype plan.

Note: Most production meshes will violate one or all of these properties, but not excessively. They still are goals to strive for.

Match Resolution With The Situation

Before designing the mesh, consideration should be given to the purpose of the study. There is more to mesh design than having well formed elements and good mesh properties. The resolution requirements for a hydrodynamic study must also consider the end use of the solution. Resolution is not only an issue of distance from the study area, but an issue of supporting post-hydrodynamic purposes, such as the sediment transport model or the ERDC at WES ship simulator model. The table below illustrates the minimum resolution requirements that are necessary to adequately define the hydrodynamics for various circumstances.

Interested in…	Type of Element	Minimal Degree of Resolution
General circulation	2D	Coarse
Eddy patterns	2D	Two elements spanning the prototype diameter of the eddy
Minimum flow exchange	2D	Two elements wide
Well-confined channel away from study area	1D and/or 2D	One element wide
Navigation channel for ship simulator	2D	Four elements laterally across the channel
Turn approaches for ship simulator	2D	One element per ship length in the longitudinal direction of flow
Turning basins for ship simulator	2D	Two elements per ship length
Sedimentation Issues	2D	Fine resolution is required to pick up deposition/erosion patterns
Non-stratified salinity intrusion in a channel	2D	Minimum of three elements wide in the channel

Pre-RMA2 Simulation Check List

After the mesh has been constructed, the following check list can be helpful. Be sure to:

- Confirm that mid-side nodes have been added to the mesh.
- Renumber the mesh with a graphical user interface such as SMS.
- Save the ascii geometry in TABS-GFGEN format.
- Run GFGEN to reorder the mesh (if needed) and save a binary geometry file.
- 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.
- Set the timing and iteration control on the TI card and TZ card.
- If hotstarting, double check parameters on the $L, TR, TI, TZ, and RSC cards

- If the simulation is likely to cause wetting and drying, a DE card and/or DM card will be required.
- Evaluate your selection of roughness and turbulent exchange coefficients.
- Make sure that all nodes in the mesh have been assigned roughness and turbulent exchange values.
- Check that the Coldstart initial condition on the IC card is equal or close to the water surface elevation boundary condition on the BH card.
- Evaluate the location and type of boundary conditions.
- Check the inflow angle of all boundary conditions.
- Save the boundary condition run control input file for RMA2.
- Save the simulation super file(s) for GFGEN and RMA2.
- Make any necessary edits to the run control files, etc.
- Run RMA2

Addressing Persistent Divergence

If convergence problems persists after you have checked the mesh quality characteristics and pre-simulation check list, you should examine the solution to determine where the simulation *first* appears to begin diverging. With a graphical user interface such as SMS, you can view the various time steps, or iterations, from the run to find the location in the mesh, and the time, where the problem begins.

Depending on the circumstances, the model may diverge very slowly or very suddenly. The rate of convergence or divergence may be monitored by examining the maximum change category of the convergence parameters found in the full results listing file (also echoed to the screen or logfile). Slow divergence usually indicates a run parameter problem, while sudden divergence typically indicates boundary condition errors, wetting and drying problems, or super-critical flow.

Note: A momentary transient effect may be mis-interpreted as divergence whenever there has been an update in the run control parameters, modification due to wetting and drying, or a change of boundary conditions (a new time step or REV card). This effect is normal, and a well constructed model should iterate to a converged solution for a given time step. It may be necessary to increase the number of iterations (TI card) to allow the solution to fully converge.

Tip: If you are unable to examine the solution at the time it first begins to diverge, try re-running the simulation with artificially higher values of eddy viscosity, adjust the parameters on the TS-Card to save the solution more often, and/or force the model to prematurely stop (TZ-Card) just prior to divergence.

Basic Operation

Data Files

RMA2 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 RMA2. The types of files RMA2 uses consist of

- RMA2 run control files (normally named with the extension **.rc2** or **.bc**)
- GFGEN binary geometry data (normally named with the extension **.gbn**)
- Hotstart input and output
- Listings of results
- Solution files in binary form containing the RMA2 numerical results.

For a normal run, you should specify as input a run control file and the binary geometry file from GFGEN, and as output a full results listing and a solution file. See the flow chart under the section "The RMA2 Modeling Process" on page 17.

Run Control File

There are many options available to you when running an RMA2 simulation. The run control file (a.k.a. boundary condition file) is what you use to tell RMA2 how to run the simulation. Every action taken by RMA2 is defined or modified in the run control file. The run control file is ascii and its contents are described in this instructional document starting on page 128.

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 again during the simulation.

Note: It is recommended that you use the primary run control file for all RMA2 input. The use of an alternate dynamic boundary condition file is available in RMA2 but is not supported in SMS.

A simple RMA2 steady state run control file is shown below.

```
T1  EXAMPLE: STEADY STATE
T2  RMA2 Run control file
T3  NOYO : PLAN 1
$L  00    0    60   64   0   3    0
SI  0
TR   0   -1   1   0   1
CO … GC   4  631  633  636   639
CO … GC   8    1    7   12    17    22   27    32   37
GCL  1  631  633  636   639   -1
GCL  2    1    7   12    17    22   27    32   37   -1
TZ   0.0   0.0   100    1    0
TI   3    0     0.0001     0.0001
FT  17.0
IC  15.0    0.2    0.25
EV      1   100.00   100.00   100.00    00.00     0.0150
EV      2   100.00   100.00   100.00    00.00     0.0500
EV      3    50.00    50.00    50.00    00.00     0.0300
BQL    1  1500.0   3.14
BHL    2  16.0
END
STOP
```

Geometry File

It is possible to code the entire mesh in the RMA2 run control file. However, this is highly discouraged. The ability to code mesh geometry for RMA2 is intended for making minor geometry modifications when testing a proposed change to the mesh.

The mesh geometry, which RMA2 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.

RMA2 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 RMA2 run control file, you must run GFGEN to obtain the geometry file before running RMA2. The $L card is used to include the geometry file in the simulation run.

The format for the binary geometry file may be found in the section entitled "RMA2 File Formats" on page 221.

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 various points (typically the end) of a simulation in order for the run to be restarted and continued at a later time. Saving a hotstart file, is similar to freezing all pertinent information in computer memory at a point in the simulation to be redefined later.

A Hotstart file is in binary form and can be both an output file, then later used as 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, as well as the time derivatives of all variables.

Prior to version 4.50, only the last iteration of a simulation could be saved to the hotstart output file. This was recognized as a shortcoming and modified. One of the newest features of RMA2 is the ability to save multiple records of information on the hotstart output file. This subsequently allows the user the option to selectively choose a particular saved record by time and iteration number for the restart initial condition. These restarting retrieve parameters are located on the RSC card, page 185.

For more information on Hotstarting, see the sections entitled "Stopping The Simulation" on page 50 and "Resuming A Stopped Simulation" on page 80. The format for the binary hotstart file may be found in the section entitled "RMA2 File Formats" on page 221.

The utility code R2_HOTFIX may be used to interrogate the records that were saved on the RMA2 hotstart. See “Determining the Records Available on a RMA2 HOTSTART File” on page 233

Results Listing File

The listing files can contain a plethora of information pertaining to the simulation results. Upon normal completion of an RMA2 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 creating results listing files, see the sections entitled "Results Listing Control" on page 49 and "Customizing The Full Results Listing File" on page 86.

Full Results Listing

If a full listing of results is desired, use the $L card to specify a full listing file. The TR card is used to specify what types of information will be written in the file. The full results listing file may contain

- Echo of the run control data
- Geometry information
- Wetting and Drying/Marsh Porosity information
- Element information
- Nodal specifications
- Front width size and total equations
- Convergence parameters and active nodal statistics for each iteration
- Nodal velocity, depth, and elevation results
- Continuity check line discharges
- Boundary condition updates

See "Understanding The Full Listing File" on page 59 for details.

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 table of the steady state solutions of *x* and *y* velocity components, depth, etc., for each node listed on the TRN cards
- A dynamic hydrograph of the above information for each node listed on TRN cards

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 tabular listing, see “Understanding The Summary Listing File” page 68.

Solution File

RMA2 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 RMA2 simulation run, contains the results of computations for all time steps defined in the run control file (see "Run Control File" on page 25). The data in the solution file can be graphically analyzed using a compatible post-processor, such as the SMS interface.

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

- *x* component of velocity
- *y* component of velocity
- Water depth
- Water surface elevation
- A flag to mark as wet or dry (NDRY)

The solution file also contains the material type number for each element, which is written as negative if the element is dry.

Vorticity Solution File

The vorticity solution file is a look-alike of an RMA4 solution file, which contains the value of streamwise vorticity at every node. Streamwise vorticity (sec^{-1}) at a node is equal to twice the rotational velocity of the fluid about the axis in the streamwise direction of flow. The vorticity solution file also contains the material type number for each element, which is written as negative if the element is dry.

The existence of the vorticity solution file is controlled on the both $L card and the TS card together. The $L card tells RMA2 to ask you for a vorticity solution file name, and the TS card controls whether the vorticity solution is actually written to the file, and when. For additional information on vorticity, see "Bendway Correction (Vorticity)" on page 93.

Using Titles

The ability to add Titles in the RMA2 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 RMA2 will not recognize the file as valid input. The T3 card actually identifies your solution file.

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. RMA2 reads the '3' to mean the *end* of the Title cards.

Tip: The information on the T3 card is retained by RMA2 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 Should You Care?

Because different computer systems may store and retrieve information in different ways, RMA2 needs to know the type of system on which it is running so it can properly transfer information on the system.

RMA2 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.

Note: The computer identification is an issue only if there is insufficient memory allotted during execution of RMA2 which would require it to write temporary buffer files to solve the large matrix.

What You Need To Do About It?

Tell RMA2 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 your system does not have enough memory available for the simulation. In this case, RMA2 will write temporary buffer files to your disk. See the chapter entitled "Performance Enhancements" on page 199 for additional information on temporary buffer files.

Example: If you are running RMA2 on a DOS or Windows-95/98 based PC, the machine identifier value should be 1.

```
 .
CO   Running on a DOS PC
$M   1
 .
 .
```

Choose The Type of Simulation

There are two options in obtaining a solution to a simulation using RMA2. These are either a steady state or a dynamic simulation.

Steady-State Simulation

The steady-state solution method removes all time derivatives from the governing equations. Therefore, the solution that the Newton-Raphson iterations are driven toward is equivalent to the solution that would be obtained if the boundary conditions were held constant and the model was run dynamically until the solution did not change from one time step to the next.

The basic run control for a "coldstart" steady state simulation is: set either the total cycles to 1 or the total run time as 0.0 hours (variables NCYC and TMAX on the TZ card), and set the number of initial iterations and depth convergence criterion (variable NITI and SSDCRT on the TI card).

By default, the binary solution file for a steady state simulation will have only one results record labeled as hour=0.0. If the TS card IBHO variable was used to request mid-iteration saves of the binary solution file, then the time stamp on the mid-iteration records will be negative.

Dynamic Simulation

The dynamic solution method includes all time derivatives and the solution changes in response to long-wave propagation characteristics. A dynamic solution can be made one of three ways:

a) A dynamic simulation may be initiated as a "coldstart" or "shock" from initial conditions of zero velocity and a flat water surface (as set on the IC card). This method usually is appropriate only when there is no river inflow and the dynamic boundary conditions are ramped from no forcing to full forcing over some period.

b) A dynamic simulation may be initiated using "hotstart" results of a previous steady state simulation. This is used normally when river inflows are present. They dynamic boundary forcings may still then be ramped.

c) A dynamic simulation may be a continuation of a run that has included a steady state simulation. This is usually only done when the problem is small and the steady state solution can be regenerated very quickly. This method is also referred to as a coldstart.

By default, the binary solution file for a coldstart dynamic (time varying) simulation will have an optional results record for the steady state solution (hour=0.0) and additional results records for each computed time step , labeled as hour=DELT, DELT*2, DELT*3…etc.

How to Decide?

The user may choose to obtain an initial steady-state solution that includes the effects of any river inflows and then add dynamic time steps for a time-varying hydrograph or tidal forcing. If there are no river inflows, it may not be necessary to run any steady state iterations prior to starting dynamic time steps.

The table below shows the common methods of handling steady-state/dynamic simulation strategies for combinations of river and tide.

RIVER Inflows	**TIDAL Forcings**	
	None	Yes
None	Why run the model?	Dynamic Only
Minor Constant Flows	Steady-State only	Steady-State + Dynamic
Strong Constant Flows	Steady-State only	Steady-State + Dynamic
Hydrograph of Flows	Steady-State + Dynamic	Steady-State + Dynamic

Tip: Experience has shown that it is not advisable to have wind forcing turned on during steady-state simulations because the "long-term steady-state equilibrium" for wind forcing tends to be both unrealistic and highly unstable.

Specifying Initial Conditions

To begin a simulation, RMA2 must have some values to use as initial guesses for the equations it will be solving. RMA2 must know the conditions for all nodes in the mesh at the instant in time depicted when the simulation begins.

Warning: In the early stages of a simulation, the solution may be contaminated by the inaccuracy of the initial conditions. Typically, for a dynamic estuarine simulation, you would ignore at least the first 24 hours of the solution.

Coldstart Initial Conditions

One-dimensional nodes should have a minimal depth and a nominal velocity assigned with the IC card.

For a Coldstart run, RMA2 begins with a global, flat, water surface elevation as specified on the IC card. Two-dimensional velocities are zero, and one-dimensional velocities are a nominal amount, also specified on the IC card. The one-dimensional nominal velocity is needed to arrive at a starting point for 1D junction flow balancing logic. The depth at each node is calculated as the difference between *initial water surface elevation* specified on the IC card, and the bed elevations from the mesh geometry.

When coldstarting with large bed slopes (such as 1 on 10), it may be helpful from a convergence standpoint to specify the initial condition water surface elevation higher than the highest bed elevation, and the minimum water depth to be greater than the critical depth for wetting and drying (DSET on the DE card); i.e., everything is wet. This will generally allow the model to more easily arrive at a solution. Because this initial water surface elevation may be artificially high, you can then use either the REV card or Hotstart to gradually lower the water surface to the realistic value.

Hotstart Initial Conditions

Data on the IC card is ignored for a Hotstart run.

Any initial conditions supplied by the user are ignored when Hotstarting because they are read from a Hotstart file generated from a previous RMA2 simulation. The $L card is used to indicate the use of Hotstart files.

See "Resuming A Stopped Simulation" on page 80 for more information on Hotstarting RMA2.

Iteration Control

It is impossible to predict the exact number of iterations that will be required to solve a particular problem. Therefore, it is recommended to specify the *maximum* number of iterations to attempt per time step, in combination with a satisfactory depth convergence criterion. This maximum number of iterations attempted for each time step, and the depth convergence criterion are specified using the TI card.

What Is An Iteration?

Webster's dictionary defines *iterative* in the mathematical sense as

> *Of, relating to, or being a computational procedure to produce a desired result by replication of a series of operations that successively better approximates the desired result.*

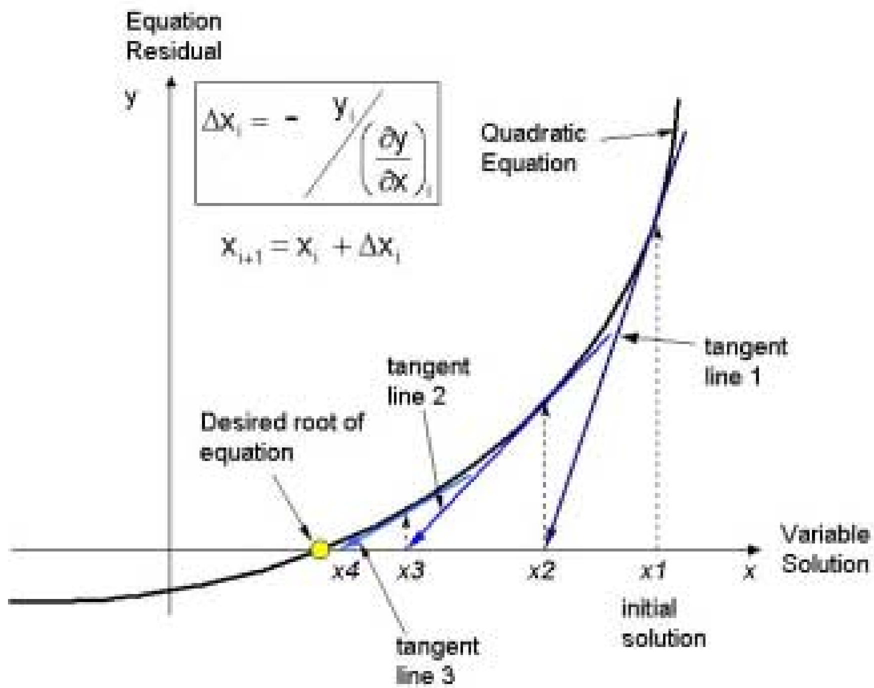
Basically, the process of *iteration* is performed by making an initial guess for the values of the variables in the equation to be solved, computing a solution, replacing that solution back into the equation, and repeating the computation. When the difference between the results from two successive iterations is less than some predetermined amount, the solution is said to have *converged*, and the problem is considered solved.

How RMA2 Finds A Solution

RMA2 uses the Newton-Raphson convergence scheme to obtain a solution. To see how this method works, see the figure below. The simplistic example is presented to illustrate the problem of solving the root of a typical quadratic equation. The idea is to find a solution to the equation as close to the root as possible.

On the *x* axis, *x1* is the initial guess at a solution. The solution with *x1* is the point marked "initial solution". A line (tangent line 1) which is tangent to the curve at this point is computed. The place where this line crosses the *x* axis becomes *x2*; the second guess used to solve the problem.

A new solution is calculated from *x2*, and another tangent line (tangent line 2) is computed. The point where this tangent line crosses the *x* axis becomes the next guess, *x3*. And so on, until the difference in value along the *x* axis, between two successive solutions, becomes less than the a pre-defined convergence criterion. At this point, the solution has converged.



where

x1 = the initial guess at the solution

x2 = the next guess, which is the solution obtained from x1

x_i = the next guess, which is the solution obtained from $x_{(i-1)}$

This example problem is considered solved when the difference between two solutions is less than the predetermined minimum amount. The slope at any point (the tangent) is mathematically the change in the residual error for a unit change in the variable x. For RMA2, the problem is to drive the residual error in the governing equations to near zero.

Note: If the first estimate is very far from the solution, the iterative technique may diverge.

For additional information, see mathematical texts discussing the Newton-Raphson method.

Regulating The Number of Iterations

In RMA2, the Newton-Raphson iterative method of successive approximations is used to solve a system of simultaneous equations. The procedure requires an initial approximation, called *Initial Condition*, to start the computations. Convergence from that first estimate to the final solution usually takes three to five iterations. Of course, the better the first estimate approximates the solution, the fewer the number of iterations required for convergence. If the first estimate is very far from the solution, the iterative technique may diverge.

RMA2 provides two means by which to control the number of iterations that are performed per time step. They are

- Providing A Convergence Criterion
- Directly Specifying the Maximum Number Of Iterations

No more iterations are performed when either of these situations are satisfied.

Tip: Generally, when using automatic coefficient procedures for either eddy viscosity (PE or SM card) or friction (RD card) the convergence rate is slower because the coefficients change as the solution changes.

Providing A Convergence Criterion

The criterion used by RMA2 to test for convergence is the maximum change in calculated depth of all nodes in the mesh from one iteration to the next. The listing below shows a representative sample of a convergence parameter section from the output which RMA2 will produce at the end of each iteration. The Depth is the third *degree of freedom* in the RMA2 equations.

The circled value in the MAX CHG column in the figure is compared to the convergence criterion you supply on the TI card. When this calculated value is less than your convergence criterion, then, unless you are using REV cards, the calculations for this time step are finished and RMA2 advances to the next time step. Otherwise, if you have REV cards, calculations with the current set of data are finished, and RMA2 advances to the next revision data set for the time step.

0.0813 in the MAX CHG column is compared to the Depth Convergence Criterion you specified on the TI card. If it is smaller, RMA2 advances to the next data set or time step.

RESULTS AT THE END OF 1 TIME STEPS...
TOTAL TIME = 0.0 HOURS... ITERATION = 10

CONVERGENCE PARAMETERS

DF	AVG CHG	MAX CHG	LOCATION	
1	0.0175	0.1885	384	X-VEL
2	0.0641	-0.2638	153	Y-VEL
3	0.0190	0.0813	563	DEPTH

The value for the depth convergence criterion is supplied on the TI card. The value typically varies between 0.005 to 0.0001 for steady state, and between 0.05 to 0.001 for dynamic runs. A more stringent criterion is required for true steady state simulations, and/or wetting and drying simulations. RMA2 will continue the Newton-Raphson iteration process until either the maximum number of iterations, or the depth convergence criterion, has been satisfied.

Note: Specifying a depth convergence value of **zero** is certain to cause RMA2 to compute for the maximum number of iterations you specified because a difference in solutions of absolutely zero is highly improbable. However, this is valid, and you may want to do this in some instances.

Directly Specifying the Maximum Number Of Iterations

The maximum number of iterations for both steady state and dynamic runs should be large enough to allow the model to sufficiently converge with the given conditions. The required number of iterations must be larger for complex hydrodynamic studies, such as simulations where wetting and drying occur and the hydrodynamics changes from one iteration to the next. The TI card is used to provide these values to RMA2.

Warning: If the problem at a given time step does not converge to a good solution in the allotted number of iterations, the solutions for future time steps may suffer as a consequence.

Now, What Do I Really Do?

A General Approach to Specifying Iterations

To eliminate extensive use of CPU resources, you may wish to limit the number of iterations performed per time step. You want to use the smallest number of iterations possible, and still solve your problem sufficiently. A trial and error technique may be necessary if you wish to optimize this number.

Tip: Experience has shown that, for *simpler* problems (no wetting and drying, no automatic coefficient calculations, etc.) with no sudden changes in boundary conditions between time steps, the number of iterations per dynamic time step can usually be reduced to two or three, and the dynamic depth convergence criterion can be zero.

A Bit More Precise

When computer resources and time are not an issue, you can choose to practically force RMA2 to meet your convergence criterion, or die trying. By choosing a large number for maximum iterations, such as 50, it is very likely that, if your convergence criterion is not unrealistically strict, RMA2 will converge to a solution before the maximum number of iterations is reached.

Dealing With Convergence Problems

If the solution has not converged satisfactorily, but appears to be getting close, you could increase the maximum number of iterations and try again.

If the solution is *diverging*, check the following situations:

- Erroneous boundary conditions, such as erroneous values in the input signal or an improper flow angle.
- Check the turbulent exchange coefficients, Peclet number, and/or Smagorinski coefficient (see "Modeling Turbulence" on page 46).
- Check for possible wetting and drying problems. When wetting and drying, additional factors are involved (see "Wetting and Drying" on page 102).
- Poor mesh design (ill-formed elements, boundary breaks, large slope gradients, etc.)

If the maxium change after several iterations is at the same node each successive iteration with approximately the same magnitude, alternating in sign, then “*ringing*” is occuring. The solution will not generally converge to any better accuracy and will often untimately diverge. Experience has shown that an adjustment in the eddy viscosity coefficients (usually an increase) will often solve a *ringing* problem.

SMS Note: If you are using SMS or FastTABS version 3.0 or higher, try the **Model Check** option, and the **Mesh Quality** options, to find areas which may lead to problems during the RMA2 simulation.

Time Step Control

Timing is introduced into the simulation when the boundary conditions (head, velocity, discharge, wind) vary in time. This is known as a Dynamic, or Unsteady simulation. The time step you use depends upon several factors, as described in "Selecting A Time Step Interval" below.

Selecting A Time Step Interval

Although RMA2 uses an implicit solution scheme, some experimentation is usually required when establishing the delta time step for dynamic simulations. In general, start with a value appropriate for your type of computations, and increase the delta time step to the largest value that is numerically stable and physically representative of the problem. For example, one hour time steps can be used in some tidal problems, even when amplitude is 6 to 8 feet, with a good reconstitution of observed gage records and current patterns.

Typically, a diurnal tide can use a 1 hour time step, while a semidiurnal tide or mixed tide requires a 0.5 hour time step.

The required computational time interval (time step size) may be dependent upon element sizes, strength of flows, flow patterns, and the rate of change in boundary conditions. Many tidal studies with this type of model employ 30 minute time steps satisfactorily, but experimentation should be used to ensure that the time step you select is appropriate for your problem. One approach to selecting this interval is to run a test case in which the time step size is reduced until the solution does not change, and use this interval for the time step.

For dynamic simulation runs, the computational time interval should be as small as necessary to capture the extremes of the dynamic boundary conditions and maintain numerical stability. Traditionally, this time interval varies between 0.25 hours and 1.0 hour for tidal boundary conditions at the coasts of North America, provided that the hydrodynamics do not include complex features, such as wetting and drying, etc.

To reduce the computational time taken to complete a simulation, the time step interval should be as large as possible, while small enough to still accurately simulate the hydrodynamics of the modeled area. The interval should be small enough to

- Capture the extremes (highest and lowest peaks) of the boundary condition signal. If the interval is too large, the peaks of the signal may be missed.
- Gradually adjust to any wetting and drying conditions.
- Accommodate rapid changes in water surface elevation.

Transport Model Considerations

If the results from the RMA2 run are to be used to drive a transport code (RMA4 or SED2D) then transport model issues may require a shorter time step. Those issues may arise from sharp concentration gradients that advect. Sometimes interpolation of the RMA2 results in time in the transport model may be adequate, other times the RMA2 time step must be shortened and RMA2 rerun.

Setting The Computational Time Step Interval

There is only one time step (solution time) used when running a steady state simulation. The computational time interval (DELT) should be set to zero on the TZ card.

Coding for dynamic (unsteady) flow simulations is only a slight extension of coding for steady state flow. The key is in the timing information. You provide the computational time step interval and total simulation time on the TZ card, and dynamic iteration control on the TI card. END cards are used to mark the end of boundary condition data for a dynamic time step.

Tip: The time step interval can be modified in mid-simulation using the BCC card. This ability is useful for refitting the time step to better approximate certain situations (as listed above), which may occur during the simulation.

Total Simulation Time

The total number of hours you want to simulate is the *simulation time*. You specify the time in hours on the TZ card.

Steady State Simulation Time

There is no time interval used when running a steady state simulation. The total run time (TMAX) should be set to zero on the TZ card.

Dynamic Simulation Time

Dynamic simulations span a predetermined period of time. Specify the number of hours (TMAX) and number of time steps (NCYC) for the simulation on the TZ card.

The RMA2 program will consider the problem finished whenever either TMAX or NCYC are first satisfied.

Note: Remember that total simulation time is not just the time you consider for results, but should also account for the “spin-up” time.

Simulating With Tides

For an estuarine model study, it is necessary to provide one or more boundary locations as a water surface elevation (such as a tidal boundary) which changes with time. For information on providing these boundary conditions, see "Providing Dynamic Boundary Conditions" on page 38.

If you do not have access to good prototype tidal elevations, then one alternative is to access or generate synthesized harmonic tidal data. There are several software packages available, which will generate harmonic tidal data at most USGS station locations. The main harmonic constituent in a real world tidal cycle typically are the principal lunar semidiurnal tide (M_2) component, which has a period of 12.42 hours, and the O1, P1, and K1 diurnal components, which have periods of 25.82, 24.07, and 23.93 hours respectively. The relative importance of each harmonic varies with location.

For additional information on tides, see The Tidal Hydraulics Engineering Manual.

Specifying Boundary Conditions

Boundary conditions are required to drive RMA2 throughout a simulation. They are constraints which are applied along the flow boundaries of the solution domain, and required to eliminate the constants of integration that arise when we numerically integrate the governing equations to solve for **u**, **v**, and *h* in the interior of the solution domain.

External boundary nodes along the downstream end of the network are typically assigned a water-level (head) boundary condition. Also, boundary nodes along the upstream end of the network are typically assigned an exact flow or discharge boundary condition. Each side wall of the network is automatically assigned a parallel flow boundary condition (i.e., slip flow) which allows the program to calculate the velocity adjacent and parallel to the side wall as well as the flow depth there.

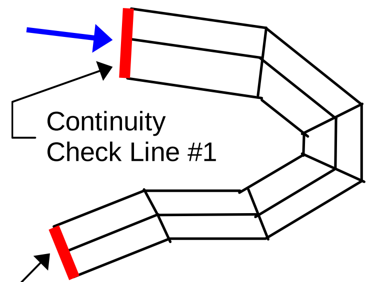
Boundary conditions may be specified on a nodal basis, along the edge of an element, or across a continuity check line. No special equations are required for boundary nodes. The use of a boundary condition specification removes either the depth, or one or both of the velocity components from the computations, and the program expects those values to be entered as boundary input data.

Note: All boundary conditions hold from one time step to the next unless they are specifically modified. RMA2 does not permit a new boundary condition location to be introducted in mid-run, nor does it allow a change in the type of boundary condition at a previously specified boundary location.

Providing Dynamic Boundary Conditions

Dynamic simulations are used to model situations where water levels, flow rates, etc., can change over time, such as an estuary where ocean tides influence the water conditions. Therefore, dynamic simulations can have different boundary conditions specified at each time step.

The boundary conditions are provided in the run control file using the boundary related data cards. For each dynamic time step, a boundary condition should be provided for each inflow, outflow, or water surface condition that has changed from the value supplied in the previous time step. The example below illustrates the use of boundary conditions assigned by continuity check line.



Example:

```
  .
  .
CO   HOUR 1.5
BQL  1  20.0  6.11   Check line #1, angle at 350 degrees
BHL  2  3.0          Check line #2
END of hour 1.5
CO   HOUR 2.0
BQL  1  30.0  6.11   Check line #1, angle at 350 degrees
BHL  2  3.24         Check line #2
END of hour 2.0
  .
  .
```

To signal that all information for a dynamic time step has been entered, an END card is used to mark the end of information for the time step. All text after the "END" characters are comments supplied by the user for ease of reading.

Types of Boundary Conditions

There are several boundary conditions from which to choose to be specified at each node:

- Parallel Flow Boundary Condition (i.e., slip flow)
- Flow Boundary Condition
- Water-Level Boundary Condition (Head)
- Stagnation Point Boundary Condition
- Reflection/Absorption Boundary Condition
- Wind Field Boundary Condition
- Wave Field Boundary Condition

Tip: The most popular set of boundary condition specifications is a downstream water surface elevation, and an upstream discharge (flow). However, the circumstances and field data availability may force you to alter this arrangement. For instance, a non-reflecting boundary condition may prove to be beneficial during a tidal simulation if the mesh domain does not extend to the head of tide.

Note: The use of a boundary condition specification at a point removes either the depth, or one or both of the velocity components from the computations at that point, and the program expects those values to be provided as input data or otherwise prescribed. The type of information removed depends upon the type of boundary condition specification you choose.

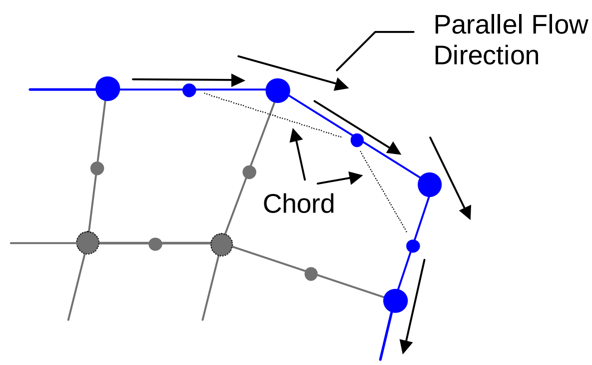
Parallel Flow Boundary Condition

The flow at all boundary nodes defaults to parallel flow (or slip flow). Parallel flow occurs at the nodes which differentiate between the wet and dry areas of the mesh. RMA2 is free to calculate the velocity adjacent and parallel to the side wall, as well as the depth at the side wall. This means that the depth can become infinitely high at the side walls if the conditions of the simulation dictate.

Parallel Flow At Mesh Exterior Edges

RMA2 adjusts the direction of the velocity vectors at boundary corner nodes such that the integral of the water flux across the boundary becomes zero. That flux is locally the velocity normal (perpedicular) to the element edge times the local water depth.

If the depth is constant along the boundary, parallel flow is the flow of fluid parallel to the *chord* connecting the midside node of each of the two adjacent elements. For midside nodes of straight-sided elements, parallel flow is simply parallel to the element edge.



Curving the element edges may alter the parallel flow angle. This allows more realistic vector angles without the computational expense of adding resolution.

Note: The default parallel flow angle can be overruled at a node by using a BA card with the *by node* option, N.

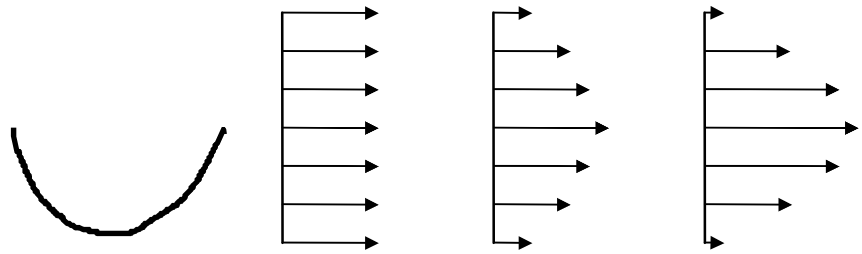
Flow Boundary Condition

A flow boundary condition is specified as either total flow or velocity. They are normally provided at inflow locations, and are used to specify the amount and direction of the fluid entering the mesh.

Total Flow

The most popular and straightforward way to specify a flow boundary condition is to provide the total flow crossing a particular continuity line with the BQ card using the *by line* option, L. The total flow (cubic units per second), flow angle (radians), and *distribution factor* at the line are specified on this card. The *distribution factor*, which can be a real number between 0.0 and 1.0, is used to weight the nodal *x* and *y* velocity components according to the nodal depths and friction across the continuity line.

Lateral Distribution of Nodal Velocities



A factor of 0.0 is the default. A factor of 1.0 will weight the velocities according to local friction and nodal depth across the continuity line. The distribution calculation uses Manning’s equation to determine the conveyance.

Note: The *total* flow crossing the continuity line is the same regardless of the value of the distribution factor.

Another way to get water in and out of elements is by using the BQ card with the *by element* option, E. This method is useful when attempting to include rainfall and evaporation in the simulation. See "Adding Rainfall And Evaporation" on page 110 for more information.

Velocity

The *x* and *y* components of velocity may be provided for each node across the boundary with the BCN card. The flow angle is determined by the resultant of the *x* and *y* components.

Warning: Since the values of the *x* and *y* components of velocity and the angle are supplied, and the model solves for depth, the total flow across the boundary will fluctuate as dynamic water surface boundary conditions take effect. If the user’s intent is to match a discharge, then us a BQ card.

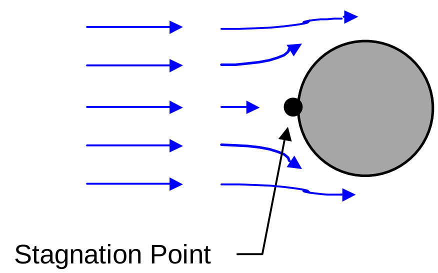
Water-Level Boundary Condition (Head)

A water surface elevation boundary condition, or *head*, is normally provided at the tailwater location. The most popular technique for assigning the head boundary condition is to specify the head across a *continuity check line* with a BH card using the *by line* option, L. An alternative method is to supply the head at each *boundary node* with the BH card using the *by node* option, N, or the BCN card. If the head is specified in one of these ways, the model will solve for the *x* and *y* components of velocity, and the flow angle at the associated node(s).

At an exit boundary, RMA2 adjusts the water depth as needed to obtain the optimum solution to the finite element equations. The minimization principles in the solution technique reach optimums when all heads, even those along the exit boundary, can be adjusted individually to preserve overall network coherence. The result is water surfaces at the exit boundary nodes that may differ slightly from the coded boundary conditions.

Stagnation Point Boundary Condition

Occasionally, it may be appropriate to use the stagnation boundary condition, which means that both the *x* and *y* velocity components are set to zero at the specified node. One situation where this may be useful is when there is a perpendicular flow at a boundary, such as at a bridge pier. The perpendicular flow at such a point is zero. The zero velocity components are explicitly assigned for that node on the BCN card.



Reflection/Absorption Boundary Condition

There are some circumstances when a boundary must be located within the tidal portion of a system and there is no known water level data to support that boundary location. Under these circumstances a permeable or reflection/absorption boundary condition (BRA card) is warrented.

The reflection and/or absorption boundary condition are considered advanced features, and will be discussed in the section entitled “Boundary Permeability (Reflection/Absorption)” on page 93.

Wind Field Boundary Condition

It is possible to assign a wind stress boundary condition by assigning the wind speed and direction to the model domain. This is accomplished with the BW Card. The storm event is a more elaborate form of applying a wind field. Storms are accomplished with the ST Card.

Wind stress and storm events are both considered advanced features, and will be discussed in the section entitled "Applying Wind Friction" on page 113.

Wave Field Boundary Condition

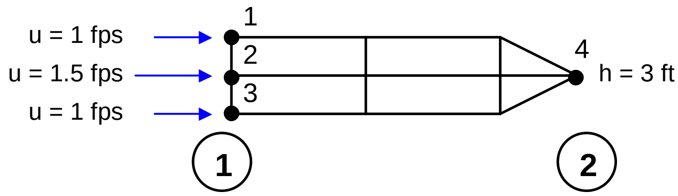
It is possible to request an input file to be read by RMA2 that is representative of the wave heights for the model domain ($L card, INWAVE). Typically a wave model, such as STWAVE, has built this file apriori to running RMA2. The file is structured such that there is a set of wave heights for each RMA2 simulation time step. The wave/current interaction switch (HS card, IWCUR) actually invokes the feature. The modeler also has the discretion to apply scales and constraints to this input file to eliminate extremes in the wave data (HS card, IWAV).

Using radiation wave stress in the RMA2 model is a relatively new feature and is considered to be an advanced feature, and will be discussed in the section entitled "Applying Wave Radiation Stress" on page 111.

Typical Boundary Condition Examples

Nodal Assignments

1. Specifying *x* and *y* velocity at a node (BCN card)
2. Specifying water surface elevation (head) at a node (BCN card)

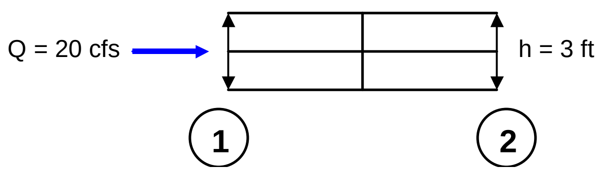


Example: **BC Assigned by node**

```
 .
 .
BCN  1  11000  1.0  0.0  0.0    at Node 1
BCN  2  11000  1.5  0.0  0.0    at Node 2
BCN  3  11000  1.0  0.0  0.0    at Node 3
BCN  4    200  0.0  0.0  3.0    at Node 4
END of time step
 .
 .
```

Assignments Along Boundary Lines

1. Specifying discharge along a boundary line (BQ card with the *by line* option, L)
2. Specifying water surface elevation (head) along a boundary line (BH card with the *by line* option, L)



Example: BC Assigned by line

```
 .
 .
BQL  line1  20.0  0.0
BHL  line2  3.0
END of time step
 .
 .
```

Note: Alternatives to the BCN card are sets of cards (BA – BS or BQ – BH) on which the same three boundary parameters are coded as on the BCN cards, but they are coded in alternate formats which may be more convenient. The first set allows an azimuth to be used along with a current or discharge to establish the inflowing velocity components: BA card = azimuth of the boundary velocity vector, BS card = speed of the boundary velocity, and BH card = the water-surface elevations (The BQ card can be used in place of the BS card). The value of NFIX is determined by the program based on card types present. The BA card should precede the others and azimuths on it will be used to calculate either velocity components or unit discharge components until another BA card is read. Only those values which differ from previous values must be changed.

Boundary Condition Gotcha's

Over Specification

An error will occur if you attempt to assign more than one type of boundary condition to the same location.

It is possible to *overspecify* the boundaries of the problem. For example, do not code water elevation and flow velocity at the same node because that attempts to force both a slope and the water discharge. Theory does not allow such in subcritical flow, nor can it be accomplished in a flume unless the roughness is a dependent variable. Also, overspecification of the problem is more likely when including internal nodes in the boundary conditions data set. Therefore, establish model performance criteria very carefully to make certain the results are not contaminated when boundary specifications are being assigned to internal nodes.

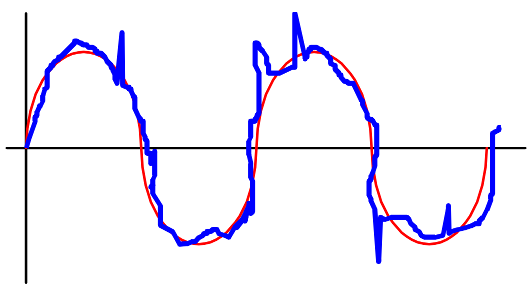
Too Close For Comfort

It is advisable to place all boundary conditions as far away from the area of interest of your study as possible and reasonable. If boundary conditions are close to your study area, the results which interest you can be unduly influenced by the values assigned for those boundary conditions.

Tip: One-Dimensional Elements are useful when you want to remotely place boundary conditions without adding a lot of resolution.

Noisy Boundary Condition Signal

A noisy/sporadic boundary condition signal with spikes is common with collected prototype data. Unless the signal is filtered or hand smoothed, RMA2 may diverge.



Dry Nodes On A Boundary Condition

If any node associated with a boundary specification becomes dry, the simulation will very likely die. Unfortunately, there are few if any diagnostics reported for this error.

Too Quick, Too Fast

If the change in elevation or discharge at the boundary varies too quickly between successive time steps, RMA2 may diverge. Generally, the change in water surface elevation should not exceed 1 foot per 1 hour time step, and the change in discharge should not exceed approximately 5,000 cfs per 1 hour time step. If you are modeling extreme tidal ranges, this suggestion may be too stringent of a restriction. If so, beware of situations where tidal bores may occur; RMA2 will not handle large Froude numbers.

See "Time Step Control" on page 36 for more information on time steps.

Bed Friction And Resistance To Flow

The bed friction energy transfer computation, or *bottom roughness*, is one of the primary verification tools for RMA2. Changing the bed friction provides some control over the fluid velocity magnitude and direction.

The bottom shear stress, τ , is defined as

Equation 5

$$\tau = \rho g R S$$

where

ρ is the density of water, g is the acceleration of gravity, R is the hydraulic mean radius, and S is bed slope.

Bed friction is calculated with Manning's equation if the input roughness value is < 3.0, otherwise a Chezy equation is used. By far, the popular choice is Manning's n-value, and these roughness values may be assigned globally throughout the mesh by material type, or on the elemental level.

The Manning's equation for uniform flow is:

Equation 6

$$V = 1.49 * \frac{R^{2/3} * S^{1/2}}{n}$$

where V is the velocity, and n is the n-value

By solving the Manning's equation for S and substituting the results into the equation for bottom shear stress, we have

Equation 7

$$\tau = \rho g \left(\frac{n}{1.49} \right)^2 \frac{V^2}{R^{1/3}}$$

Since the hydraulic mean radius, R, is the cross section area divided by the wetted perimeter of the cross section is approximately equal to the depth for wide channels, the final form of the components for the bottom shear stress in terms of Manning's equation is

Equation 8 (a and b)

$$\tau_x = \rho g \left(\frac{n}{1.49} \right)^2 \frac{u\sqrt{u^2 + v^2}}{h^{1/3}}$$

$$\tau_y = \rho g \left(\frac{n}{1.49} \right)^2 \frac{v\sqrt{u^2+v^2}}{h^{1/3}}$$

where h is the channel depth.

Note: RMA2 provides the means to input only the bottom roughness, not the side wall roughness.

Tip: Because there is no wall roughness in RMA2, you may need to exaggerate the bed roughness on the elements forming the edge of the waterway in order to approximate the wall roughness.

Defining Roughness

There are two ways to assign roughness:

- Manual Roughness Assignment
- Automatic Roughness Assignment (for Manning's n only)

Manual Roughness Assignment

You may assign particular n-values using the HN card or EV card. Assignments can be made by individual element or by material type. Global assignment is also possible.

Automatic Roughness Assignment

You may choose to have RMA2 automatically assign the *Manning's n* roughness values for each iteration according to the calculated depth at each Gauss point with the RD card (see "Automatic Friction Assignment" on page 99).

Modeling Turbulence

Turbulence may be defined generally as the effect of temporal variations in velocity, and the momentum exchange associated with their spatial gradients. In particular, turbulence is viewed as the temporal effects occuring at time scales smaller than the model time step.

Some numerical model formulations require the addition of a minimal level of artifical diffusion in order to obtain a "stable" solution that converges in the Newton-Raphson iterative scheme. The Galerkin method is one of these, because the basis numerical procedure includes no inherent artifical diffusion. Other numerical methods, e.g. finite difference methods, include artificial numerical diffusion within the formulation.

The Galerkin method of weighted residuals employed by RMA2 does not include any inherent form of stabilization other than the eddy viscosity terms. The Galerkin formulation does require a certain amount of added turbulence to achieve stability. However, one may get useful solutions by specifying a higher turbulence value than is physically justified. This technique will produce a stable model without the need to add too much resolution. However, if taken to excess, the velocity distributions will be smeared in space and time.

What Is Turbulence?

The eddy viscosity terms in the governing equations actually represent the molecular viscosity and the effects of turbulence from the Reynold's stress terms.

Equation 9

$$E_{xx}\frac{\partial^2 y}{\partial x^2} = \mu\frac{\partial^2 y}{\partial x^2} + \frac{\partial}{\partial x}\frac{\overline{\partial u' v'}}{\partial x}$$

where

μ	=	Molecular viscosity
u', v'	=	Turbulent deviations of the instantaneous velocity from the temporally averaged velocity *u*
$\overline{u' v'}$	=	Time averaged of the velocity products over a time step

Thus the eddy viscosity, E_{xx}, includes both effects; but under most flows of interest the Reynold's Stress terms are much larger than the effects of the molecular viscosity.

These fluid momentum transfers due to exchanges of fluid masses moving at different speeds is called turbulence exchange. The English units for turbulence are lb-sec/ft^2. The program allows for the exchange coefficient to be specified in a local coordinate system for each element

Although it is difficult to establish the value for E, analogy with physical conditions suggests that turbulence exchanges depend on the momentum of the fluid and the distance over which that momentum is applied (i.e. length of the element). Therefore, as the element size increases, E should increase, or as the velocity increases, E should increase.

Specifying Turbulence

Turbulent exchanges are sensitive to changes in the direction of the velocity vector. Conversely, small values of the turbulent exchange coefficients allow the velocity vectors too much freedom to change magnitude and direction in the iterative solution. The result is a numerically unstable problem for which the program will diverge rather than converge to a solution. One recourse is to continue increasing the, E, until a stable solution is achieved.

There are three basic ways to control the turbulent exchange coefficient, E.

- Direct Assignment
- Automatic Assignment By Peclet Number
- Automatic Assignment By Smagorinski Coefficient

Direct Assignment

The first and direct way to assign the turbulent exchange coefficient, E, is to assign a particular value for each individual material type with the EV card. All material types < 900 that exist in the mesh must be specified with an EV card.

As a guideline for selecting reasonable values for the turbulent exchange coefficients for a given material type you should:

- determine a representative length of the elements within the material type

- estimate a maximum streamwise velocity for the given material type
- solve the Peclet equation for E (Equation 10), given a Peclet number of 20

Several modelers choose to expedite this process. A utility program called MAKE_EV_DF will calculate the average elemental size of each material type and create a table of recommended eddy viscosity values for each material type in the mesh. The need for this utility program has been replaced by automatic methodologies for assigning the turbulence parameters.

The table below provides some representative ranges of turbulent exchange coefficients.

Type of Problem	E, lb-sec/ft^2
Homogenous horizontal flow around an island	10-100
Homogenous horizontal flow at a confluence	25-100
Steady-State flow for thermal discharge to a slow moving river	20-1000
Tidal flow in a marshy estuary	50-200
Slow flow through a shallow pond	0.2-1.0

Automatic Assignment By Peclet Number

The second way to assign the turbulent exchange coefficient, E, is to allow the model to automatically adjust E after each iteration, based upon a provided Peclet number, which is based upon the unique size and calculated velocity within each element. The Peclet number defines the relationship between the average elemental velocity magnitude, elemental length, fluid density, and E.

Recall the formula for Peclet number (P), where P is recommended to be between 15 and 40.

Equation 10

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

where:

Coefficient			English Units	Metric Units
ρ	=	fluid density	1.94 slugs/ft^3	1 grams/cm^3
u	=	average elemental velocity	Fps	mps
dx	=	length of element in streamwise direction	Ft	m
E	=	eddy viscosity	lb-sec/ft^2	Pascal-sec

Automatic Assignment By Smagorinski Coefficient

The third method of assigning the turbulence coefficients (eddy viscosity) is by using the automatic Smagorinski methodology. By using the SM card, you can provide *real time* adjustment of eddy viscosity based upon the computed velocity. The strength of this automatic feature over the Peclet technique is that it takes into consideration the gradients of velocity to determine the appropriate turbulence coefficient to meet the conditions in the hydrodynamic simulation.

Generally, larger elements, and elements with higher velocities will have larger eddy viscosity values. The Smagorinski coefficient, TBFACT specified on the SM card, has a recommended value range between 0.094 and 0.2. Larger values will result in larger values of eddy viscosity.

Equation 11

$$E = TBFACT * A\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + 2\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)\right]$$

where:

A=the area of the element

$\partial u/\partial x$ and $\partial v/\partial x$ =average elemental change in the velocity component

E=eddy viscosity

For details on how to manual and automatic features interrelate with one another, see "Automatic Turbulence Closure Assignment" on page 100.

Results Listing Control

In addition to the output displayed to the computer screenduring execution, there are two primary types of results listings:

1. Screen Listing
2. Full Results Listing
3. Summary Results Listing

Screen Listing

The screen listing is a temporary file, that provides the convergence status and statistics of the simulation during the course of the program execution. Once the information scrolls off the screen monitor it is lost, unless the contents are redirected (>) to become a permanent file.

Since this file is compactly designed for quick examination, there is very little that can be done to reduce the contents presented. The IECHO and the debug ITRACE variables on the TR card are the only significant controls.

Full Results Listing

To create a full results listing file, set the variable IOUT on the $L card to any positive number. Results are written to the full listing depending upon settings on the TR card (see "Customizing The Full Results Listing File" on page 86 for more information).

Coldstart

For a Coldstart, it is recommended that new users request a complete full listing at the end of each time step. This is helpful when diagnosing problems with a simulation

Example: To obtain a full results listing when Coldstarting, use the TR card set as follows:

```
  .
CO   Coldstart, write results after every time step
TR   1  -1   1   1   0
  .
  .
```

Hotstart

For a Hotstart, it is recommended that new users request a full listing at the end of each time step. A complete listing of Hotstart initial conditions is virtually the only way to verify that the model restarted correctly.

Example: To obtain a full results listing of initial conditions when Hotstarting, use the TR card set as follows:

```
  .
CO   Hotstart listing, write results every 12th step
TR   2  -12   1   0
  .
  .
```

Conduct a short hotstart run with the full listing option to verify that the run is started correctly. Then restart the run with the full listing minimized to avoid creating an extremely large listing file.

See Also: "Results Listing File" on page 27, "Understanding The Full Listing File" on page 59, and "Customizing The Full Results Listing File" on page 86.

Summary Results Listing

To create a summary results listing file, set the variable ISPRT on the $L card to any positive number, and supply a list of nodes with TRN cards. Summary results are written at the completion of each time step.

Tip: The summary listing for specified nodes provides a means to use RMA2 results in spreadsheet software for graphing and further analysis.

See Also: "Summary Results Listing" on page 27, "Understanding The Summary Listing File" on page 68.

Stopping The Simulation

Aside from turning off your computer, there are several ways to stop a simulation run:

1. Normal Run Completion

2. Internal Logic Control:STOP
3. Math Library Error: STOP
4. Mannually Stopping A Simulation

Normal Run Completion

The intended way for a simulation to end is for it to run to completion. If the simulation stops normally, any one of the following must have been satisfi

- Total Simulation Time
- Total Number of Time Steps

Upon normal completion, RMA2 will close all input and output files, and erase any temporary files it may have written during the run.

Total Simulation Time

RMA2 will stop after completing TMAX hours of simulation. TMAX is specified on the TZ card.

Total Number of Time Steps

RMA2 will stop after completing NCYC time steps. NCYC is provided on the TZ card.

Internal Logic Control:STOP

The program has several internal check points that will cause the model to stop.

- A STOP Card
- Error Detected
- A Diverged Simulation
- Expired Executable

A STOP Card

A STO card (Stop card) should always be included in the run control file. The STO*P* card tells RMA2 that there are no more data to read and the simulation will terminate. However, the non-traditional placement of a STO*P* card can cause premature termination.

Note: The STO*P* card should always be preceded by an END card.

Tip: Normally, a STO*P* card is placed at the end of the run control file. However, when debugging, it may be useful to place a STO*P* card at an earlier point in the run control sequence, say, after the first time step for example to compute steady state results only.

Error Detected

If RMA2 detects an error in the input for any reason, the model may hit a logical stop. If this occurs RMA2 will echo out the reasons to either the screen and/or full results listing output.

See Also: "The Simulation Stops Prematurely" on page 203

A Diverged Simulation

RMA2 will stop whenever the convergence parameters indicate a maximum change in depth in excess of 25 feet from one iteration to the next.

Expired Executable

RMA2 will stop whenever the alloted time for the program executable has exceeded the expiration date. Typically the expiration date is 30 October, which is 30 days past the end of the fiscal year.

Math Library Error: STOP

On occasion, RMA2 will be stopped at the operating system level due to an error in a math library routine. These are normally either a (1) divide by zero, (2) a floating-point exception, (3) divide by zero, or (4) an exponential underflow or overflow.

Such errors are usally the result of a diverging solution that creates extreme solution values. As RMA2 evolves, more of these types of errors are being logically trapped and diagnosed for the user prior to causing these obscure messages.

Mannually Stopping A Simulation

If you need to stop the simulation before it has completed, you can interrupt RMA2 using the means available on your system to kill a process, typically by pressing Control-C, or Control-Break, at the keyboard.

Note: If the simulation runs to normal completion, all associated scratch files will be eliminated and all input/output files will be closed normally and saved if applicable. However, If the simulation is abruptly terminated (Control-C from the keyboard, or the CPU limit is exceeded, etc.), large scratch files may be left resident on your disk. The scratch files are typically assigned to unit 9 in batch mode, or, in interactive mode, the filenames may start with a capital 'H'. If these file are left resident on the disk, you should delete them.

Any Comments?

RMA2 allows the use of comments in the run control file. Comments may be placed anywhere in the file, except on the first line, where a Title card is required (see "Using Titles" on page 28).

To include comments, use the CO card.

Verifying The Model

Verification: A Process

Some numerical modelers refer to calibration and verification as a two-step process. Using this terminology, adjustments are made to model coefficients and inputs so as to optimize agreement between model and observed prototype data during the calibration step. Then the model is run to attempt reproduction of a different set of prototype data without further model adjustment. If the second run is satisfactory then the model is considered verified.

This procedure sounds imminently reasonable, but experience suggests that it is a naive approach. The reasons are these:

1. All field data include errors, and sometimes dramatic errors. Thus, they are not an absolute standard.
2. Field measurements include a variety of effects that may not be reproduced in the model (for example, groundwater flow into the model)
3. Conditions often change between field surveys, implying that coefficients should also change (for example, differences in bed forms at different flows may dictate a change in bed roughness coefficients)
4. Most natural waterways cannot be adequately characterized by two field data sets. Five or ten may be needed, but available resources usually limit the field data.
5. If the model reproduces one field data set adequately, but not the second, you must decide whether to:

   a) Proceed with modeling, conceding an incomplete verification.

   b) Continue adjusting/revising to obtain a balanced quality of reproduction.

   c) Conduct a re-analysis/re-collection of field data.

The philosophy exposed here is that verification is a multi-step process of model adjustments and comparisons, leavened with careful consideration of both the model and the data. It is not a simple two-step procedure, and the term calibration should not be used. The purpose of numerical modeling, as stated by W. A. Thomas, a retired research hydraulic engineer from WES, is to “gain insight, not answers”.

Checking For Continuity

Continuity refers to checking the water mass flux. The objective when simulating is to retain the correct amount of fluid flow from one point to the next, within a tolerance of about plus or minus 3%. *Continuity check lines* provide a means to determine if your steady state simulation is locally maintaining mass conservation at a given location.

Continuity check lines are typically used to estimate the flow rates at cross-sections perpendicular to the flow path and serve as an error indicator. The RMA2 model globally maintains mass conservation in a weighted residual manner. Locally, continuity check lines can be used to check for apparent mass changes in a different way, by direct integration. Large discrepancies between the results of these 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 $\pm$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.

Of course, all this assumes a converged solution.

NOTE: It is best to use the continuity check lines to assess the accuracy of the model for steady state simulations. For a steady state simulation, all inflows must balance all outflows and intermediate flux checks within the model directly reflect accuracy. However, once the model is run with transient boundary conditions, the instantaneous fluxes IN will balance the fluxes OUT *plus* changes to STORAGE within the domain of the model. There is no direct diagnostic printout to obtain a time-varying water balance accuracy check.

Continuity Check Lines

Continuity check lines provide two services:

1. Provides a convenient means to assign a boundary condition.
2. Allows for reporting the total flow passing through the line.

Continuity check lines are typically assigned using a GC card, by selecting corner nodes from the right bankline to left bankline, facing downstream. They should be oriented perpendicular to the direction of flow. Avoid placing check lines in an area where an eddy may form, unless your intent is to study the eddy.

After The Simulation

Critical Check Points

When you run a simulation, you should first determine if the solution is numerically stable and realistic. Critical check points include the convergence parameters, and total flow across continuity check lines found in the full results listing, and examination of the solution by means of a velocity vector plot and water surface contour plot.

Continuity Checks And Conservation Of Mass

Poor continuity , or “leaking”, may be the result of poor mesh resolution, sharp boundary breaks, or severe curved edge related problems. The difference in convergence parameters should decrease with each iteration within a given time step (acceptable exceptions would involve wetting and drying or revisions of the network within a time step).

Continuity checks can be used in steady state to determine if boundary conditions are holding properly and whether mass is conserved. Graphical analysis of a stable model will not have abnormal vector behavior or spurious contours.

Tip: In general, curved short element boundary edges improve continuity. Conversely, long boundary elements with sharp breaks in direction make for poor mass balance, .

Effects Of The Initial Conditions

For dynamic runs, the initial conditions can adversely affect the results during the first part of a simulation. This is because the model is “shocked” by the initial conditions if they are not very close to what is expected.

For example, to model and stabilize a large ponded area may require the initial water surface elevation and boundary conditions be artifically elevated. If you then proceed directly to a dynamic simulation, it may take a long simulation to “draw down” the water levels to a reasonable soluton. A better approach would be to “revise” (RE card(s)) the initial solution sequentially down to a reasonable level to provide a more realistic starting point for the dynamic simulation.

Check to insure that the time interval you are concerned with is not affected by the initial conditions. A time series plot of both water surface and velocity at a valid control point will illustrate when the model begins to repeat the solution. This technique is valid only if you are running a repetitive boundary condition.

Tip: For a tidal simulation, use a smooth sinusoidal repetitive water surface elevation boundary condition signal as input to estimate spin-up time.

Is The Model Realistically Simulating Wetting And Drying?

A network designed such that the edges of elements fall on contours which are critical to wetting and drying will usually perform satisfactory. However, a poorly designed network constructed without attention to wet/dry boundaries will oscillate and sporadically pull elements in and out of the solution.

A quick check point is to scan the total number of active nodes for each iteration. A more specific indicator is to look in the full results listing to see exactly which element numbers have been added or eliminated per time step. In addition, graphical analysis will display the network as it exists for that time step. Instability may result when the dried edge results in a jagged boundary.

For wetland applications it is advisable to run the marsh porosity technique in conjunction with traditional wetting and drying by element. The marsh porosity technique allows elements to transition gradually between wet and dry states.

If Your Verification Is Unsatisfactory

If you are experiencing difficulties verifying the model, you should check the items listed below.

Wrong Choice Of Model

If you attempt to model three-dimensional current phenomenon with a two-dimensional depth averaged model, the situation is basically hopeless. You should use a three-dimensional model such as the TABS-MDS, formally known as RMA10.

If you have attempted to model a very complex cross-section with a RMA2 1D trapezoidal section, you may be disappointed with the results. You may have better success using RMA2 fully 2D or use a more sophisticated 1D model with a more accurate cross-section.

Geometry Problems

Geometry is more than just the assemblage of nodes and elements, it incorporates the idea of study design and model domain. There are several geometry issues which may cause verification problems.

Element Problems

Check for *ill-formed elements* as described in the GFGEN Geometry manual.

Boundary Location Problems

Boundary conditions should be located away from the study area where data is reliable. Look for locations such as a dam, lock, head of tide, or null point in the flow. Avoid assign them at locations with high velocities.

Wetting And Drying Problems

Incorrectly simulating wet/dry conditions can lead to numerical shocks, large boundary break angles, and/or water storage problems. See "Wetting and Drying" on page 102 for details on wetting and drying.

Water Storage Problems

Too little, or sometimes too much, tidal storage, or backwater storage, may be present in the geometry. Storage areas will affect velocities. Add, or remove, storage areas as necessary to match the real world system.

Bathymetry Problems

Be sure the mesh is in agreement with the prototype conditions which are being simulated. Also, be sure the bathymetry is fairly smooth (maybe you can swap an element edge between corner nodes). RMA2 is based upon the Mild Slope Assumption (see "Limitations Of RMA2" on page 3).

Resolution Problems

Resolution must be appropriate to the phenomena modeled. To model an eddy, for example, you must have enough resolution to capture the eddy.

Boundary Condition Problems

Use good, reliable boundary data at all boundaries. For example, ship traffic effects should be removed from the boundary condition used to drive the RMA2 model. None of the boundary conditions should include noise from turbulence at a time scale shorter than the model time step. It may be necessary to apply a low-pass filter, to the boundary condition prior to running the model.

The boundary conditions must match the conditions during which the field data was collected.

Choose an appropriate type of boundary condition for the boundary location (elevation, discharge, rating curve, wind speed and direction, fluid speed and direction, tidal reflection/absorption)

Check for erroneous boundary conditions, such as a sporadic value in the input signal or an improper flow angle.

Roughness Problems

Roughness affects the water surface profile (steady state river simulation). It can affect the speed of the flow in the river passage. Wetting and drying may be affected.

Because there is no wall roughness in RMA2, you may need to exaggerate the bed roughness on the elements forming the edge of the waterway in order to simulate the wall roughness.

Eddy Viscosity Problems

Turbulence issues can create problems during a simulation. Since viscosity is involved in the equations of motion, it effects the velocity distribution, which will effect the depth, which will determine the effects of roughness, which may in turn effect the phase and amplitude of the tidal signal, etc.

If the eddy viscosity is too high, velocity magnitudes will appear uniform across the channel. Eddies will not form in the waterway if the eddy viscosities are too high. Picture a high viscosity value having the effect of turning the water to “syrup”.

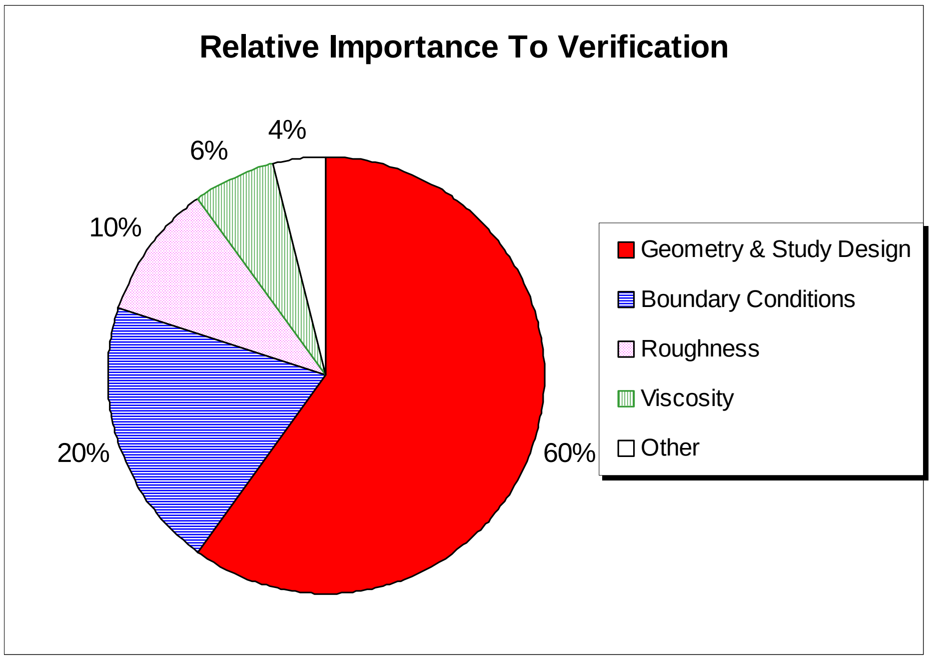
Excessively high eddy viscosity may cause instability. This is particularly true when flow is being directed into a side channel. If the flow is too “viscous” the flow does not have the lateral flexibility to change direction. Lowering the eddy viscosity may yield a stable solution.

If the eddy viscosity is too low, the velocity contour patterns will appear erratic. The textbook case of a two delta-X oscillation, defined as an unrealistic neighboring high/low/high/low contour pattern, may arise.

Note: Eddy viscosity and Peclet number are inversely related. For more details see "Modeling Turbulence" on page 46, and "Automatic Turbulence Closure Assignment" on page 100.

Influence On Verification

This pie chart illustrates the approximate relative importance to the simulation of the different aspects of an RMA2 simulation study, as described above. As you can see, the structure of the geometry and overall study design are the most significant, followed by the boundary condition assignments. The “other” category include field data issues, amount of time devoted to the effort, approach chosen to analyze data. Study design includes model choice and boundary placement. The quantitative percentages are intended only to give the user a feel for what is important in achieving a successful study.



Interpretation Of Results

Overview

RMA2 is capable of generating more data in one dynamic 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, convergence parameters, steady state continuity checks, and mini-statistics. Most of these key segments are located in both the full results listing file and the screen output. Additional global and detailed information can be obtained from post-processing utilities, and/or a graphical user interface such as SMS.

To become an good RMA2 user, you should be aware of what is included within the three types of RMA2 output.

- Understanding The Full Listing File
- Understanding The Summary Listing File
- Interpreting The Solution

Understanding The Full Listing File

The Full Results Listing file is the most comprehensive analysis tool available in RMA2. It may seem overwhelming with all its lists of numbers and computer lingo, but it's really not.

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

- Model Information
- Input Interpretation
- Marsh Porosity Parameters
- Element Connection Table
- Echo Of Boundary Conditions
- Nodal Specifications Or Initial Conditions (Hotstart)
- Steady State And Dynamic Simulation Progress And Statistics
- Steady State And Dynamic Simulation Nodal Results

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 keywords which are unique to each section. These keywords are presented below within the description of each section.

TIP: When first initiating a model run, the user should turn on as much diagnostic printing as possible and run the model for a short run. The resulting full listing will be a manageable size for editing. If the model is run for a long simulation with full listing options activated, the file may be too voluminous to examine. Therefore, longer simulations should set input controls to minimize the length of this file.

Model Information

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

File Names

The file names supplied to an interactive RMA2 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, look at the beginning of the full results listing file.

Version Number And Modification Date

The version number and last modification date of RMA2 are provided here. This information can be very important when obtaining technical support.

To find this section in the full results listing file, search for the text "**RMA2 VERSION**".

Here is a typical representation:

```
RMA2 VERSION 4.52   1D & 2D CAPABILITY.
LAST MODIFICATION DATE:  09-27-2000
```

Program Dimensions

The program array dimensions used for the simulation are listed here. If RMA2 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**. If reordering was performed in SMS using the *frontal method*, the maximum front width is reported within SMS.

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

Here is a typical representation:

```
THIS PROGRAM IS DIMENSIONED AS FOLLOWS

      MAX NO. OF NODES                                 20000
      MAX NO. OF ELEMENTS                               6500
      MAX NO. OF EQUATIONS                             40000
```

```
      MAX FRONT WIDTH                                     600
      MAX NO. OF CONTINUITY CHECK LINES                   100
      MAX BUFFER SIZE                                  100000
      MAX PRINT-SUMMARY BUFFER                          50000
```

If your problem size exceeds any one of these reported dimensions, the user will be alerted. At this point the only recourse is to either decrease your problem size or redimension RMA2 by modifying the include file and recompile the program. Of course you must have access to both the source code and a FORTRAN compiler to accomplish a redimension and recompile.

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 RMA2 by increasing this dimension in the RMA2 source code. For details, see "Reducing The Number Of Temporary Files" on page 200.

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

- Run Control Parameters
- Geometry Input Summary

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:

- Selected options
- Logical unit number assignments for files
- The machine identifier
- Various geometry parameters
- Timing controls
- Peclet number, Smagorinski coefficient, and eddy viscosity information
- A list of assigned continuity check lines.

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

Note: This section may appear in multiple locations if revisions (REV card) have been specified.

Geometry Input Summary

This small section of the full results listing file is to inform you as to what RMA2 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 RMA2 has properly read your geometry data.

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

Here is a typical representation:

```
     .....NETWORK INPUT COMPLETE.....

          MAX ELEMENT NUM =  200
          MIN ELEMENT NUM =    1
          MAX MATERIAL NUM =   3
          MIN MATERIAL NUM =   1
          MAX NODE NUM =     640
          MIN NODE NUM =       1
```

Marsh Porosity Parameters

An echo of the nodal marsh porosity parameters: AC1, AC2, AC3, and AC4. This information will always be included in the full results listing file if the marsh porosity option is activated with the DM card. If RMA2 was run without a DM card, this section will contain only the text **MARSH ELEMENTS INOPERATIVE**.

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

Element Connection Table

The element connection table contains detailed information on all the elements in the mesh. This information is omitted from the full results listing file if IPRT on the TR card is set to zero.

The element connection table lists all of the element numbers along with their associated node numbers, as well as the Coriolis (Latitude) values for each element. Additionally, this table will allow you to determine the nodes which make up an element, the material type, the elemental area, and any modifications which may have been made to the angle specified for the eddy viscosity tensor.

To find this section in the full results listing file, search for the text "**`NODAL 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. An element with *zero* area may indicate that no eddy viscosity has been specified for the material type associated with the element. Alternatively, elements with zero area may be a control structure, junction, or an element whose material type is zero.

SMS Note: Safeguards within the FastTABS and SMS mesh generator help prevent you from making common network errors, such as, an element with a negative area (referenced in the "Full Results Listing" on page 27 and other element connection errors. However, these are indeed fatal errors, which must be corrected.

Echo Of Boundary Conditions

This section of the full results listing file lists the boundary conditions that were specified for the simulation run. Boundary conditions defined as element flow, flow across a check line, and elevation along a check line can be listed here. The type of boundary condition is stated, then the associated parameters defining the boundary condition.

Note: If you are simulating using rainfall or evaporation, the flow rate value listed for boundary conditions defined as element flow for the affected elements will be different from what you specified. The values have been converted to be in the units used by RMA2 during calculations.

This section may also contain information on any flow control structures that are defined.

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

Note: This section may appear in multiple locations if revisions (REV card) have been specified.

Nodal Specifications Or Initial Conditions

At this point in the full results listing file you will find either the Nodal Specifications or the Initial Conditions. What is located here depends upon the value of IPRT on the TR card.

Nodal Specifications

In the full results listing file, the nodal specifications include the *x* and *y* location of each node, its bottom elevation, and its network slope. Generally, a network slope value of zero indicates an interior node not located on a boundary.

Additionally, the nodal specifications are useful for determining whether a particular node has a boundary condition assigned, and if so the type of specification. The type is listed as a number in the column marked **B-C FIX** (see the BCN card for a description of these numbers). If the BCFIX type is zero, no boundary condition is assigned at that node. If a node has a boundary condition specified as flow, the flow rate in the *x* and *y* directions will be listed for the node. If the boundary condition is specified as an elevation, the elevation will be listed.

The nodal specifications also include one-dimensional and two-dimensional nodal cross-sectional information. The node surface width at zero water depth, left and right side slopes, and storage width are included. For the nodes without cross-sectional storage, these data fields should be zero unless 2D off-channel storage is being used.

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

Note: The Nodal Specifications only appear when IPRT=1 on the TR card.

Initial Conditions (Hotstart)

To assure that RMA2 Hotstarted at the correct location from the previous run, look in the full results listing file and verify that the information within the **ACTIVE NODAL STATISTICS** for the first iteration of the Hotstarted run match the **ACTIVE NODAL STATISTICS** from the last iteration of the previous run.

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

Note: The Hotstart Initial Conditions only appear when IPRT on the TR card is set to 2. This section is only useful if Hotstarting the model.

Steady State And Dynamic Simulation Progress And Statistics

If a steady state simulation was performed, it is used as the initial condition for the subsequent dynamic simulation.

This section contains information which shows how the model solution is being solved during the simulation. The information will appear both in the steady state and the dynamic sections of the full results listing file. This section has the same format for both the steady state and dynamic portions of the simulation, so this text is applicable for describing it for both portions.

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 stability problems.

For each iteration, various information is written to the full results listing file. Each iteration will include the simulated hour and the iteration cycle. For steady state, the simulated hour is always zero.

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 60.

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

Buffer Blocks

Before RMA2 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 60, is not large enough to allow RMA2 to store the matrix in memory, RMA2 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 text "**`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 RMA2 by increasing this dimension in the RMA2 source code. For details, see "Reducing The Number Of Temporary Files" on page 200.

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 front width and number of equtions may change during the course of a wetting and drying simulation.

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

Here is a typical representation for a small application:

```
TOTAL NUMBER OF ACTIVE SYSTEM EQUATIONS =   1296
BUFFER BLOCKS WRITTEN=   18       FINAL LQ SIZE=   434
MAXIMUM FRONT WIDTH=     82
```

Convergence Parameters And Nodal Statistics

The convergence parameters and nodal statistics are written to the full results listing file, and to the terminal display, after every iteration. They inform you as to how well the model is performing at any given iteration. The convergence parameters and nodal statistics are normally examined together. In situations where the model is diverging, these sections can provide you with the location of the problem.

Here is a typical representation. This solution is diverging and the problem seems to be at node number 1. The section marked in navy blue will appear only if an automatic parameter calculation was requested during the simulation (PE or RD cards).

```
RESULTS AT THE END OF    0 TIME STEPS...
       TOTAL TIME =     .000000 HOURS... ITERATION CYCLE =   14

          CONVERGENCE PARAMETERS

        DF        AVG  CHG        MAX CHG     LOCATION
         1          .0316       -22.9952         1    X-VEL
         2          .0191         1.9483        32    Y-VEL
         3          .0196         -.2465      7259    DEPTH

          ACTIVE NODAL STATISTICS FOR THIS ITERATION

    NODE  XVEL-MAX    NODE  XVEL-MIN    NODE  YVEL-MAX   NODE  YVEL-MIN
     119     1.126       1   -21.617       1    12.016    119      -.930

    NODE  ELEV-MAX    NODE  ELEV-MIN   AVE-ELEV  NODES ACTIVE/TOTAL
      31   835.875    6992   835.460    835.566       8751 / 8760

          AUTOMATIC PARAMETER STATISTICS FOR THIS ITERATION
    ELEM    EV-MAX    ELEM    EV-MIN
     #     value       #      value
    ELEM  RUFF-MAX    ELEM   RUFF-MIN   NODE  DEPTH-MAX   NODE DEPTH-MIN
     #     value       #      value      #      value      #     value
```

The convergence parameters and nodal statistics can help you find problem areas in your mesh. If the solution is diverging, you can look at the convergence parameters at previous iterations to find the time when, and the nodal location where, the divergence first began to occur.

Convergence Parameters

The column marked **AVG CHG** is the average of all active nodes for each of the three degrees of freedom: *x* component of velocity, *y* component of velocity, and water depth. The column marked **MAX CHG** is the value of the maximum change between this and the previous iteration for each of the three degrees of freedom. The column marked **LOCATION** is the node at which the maximum change occurs for the three degrees of freedom.

For information on setting the convergence criterion, see "Providing A Convergence Criterion" on page 34.

Nodal Statistics

The nodal statistics tell you the nodes where, during a given iteration, the minimum and maximum velocities and elevations occurred, and the associated values. The number of **NODES ACTIVE** indicates the number of nodes flagged "wet" for this iteration. The **NODES TOTAL** indicates the number of nodes present on the input mesh.

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

Negative Depth Warning

A new diagnostic printout was added for version 4.51 or higher of RMA2. The warning follows the nodal statistics section. This message will occur if negative depth(s) have been computed for this computational time step/iteration. This type of warning is indicative of wetting and drying problems.

```
 NEGATIVE DEPTH WARNING:  For entire mesh=   1    For active mesh=   1
```

Although it is possible that RMA2 will continue the simulation with the negative depth(s) warning, the modeler should take this warning seriously. Negative depths can contaminate the hydrodynamic solution file with nonsense values. If this solution is passed along to the transport models (RMA4, SED2D,etc.), they will produce even more nonsensical results. In fact the transport model may not run at all under these conditions.

The quickest way to further diagnose and identify the areas of the mesh that are experiencing negative depths is to activate the debug trace (TR card, set ITRACE=1). This will cause a file named "r2negdepth.dat" to be generated. All occurances of negative depths are written to this file in a tabular SCAT2D file format. The contents of this file has all of the vital nodal information for the problem node(s).

To find this section in the full listing file, search for the text "**NEGATIVE DEPTH**".

Continuity Checks

These statistics are the last information written to the full results listing file for an iteration, unless wetting and drying is requested. In this section you will find the amount of flow across each check line you have defined. The flows are listed as **TOTAL**, **X FLOW**, **Y FLOW**, and **PERCENT**. The percent field is always with respect to the *first* continuity check line and is important for analyzing any loss or gain of mass at the check line. The value here should ideally be 100% for a converged steady state simulation, if the full inflow is passing the first continuity line.

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

Wetting And Drying Information

If wetting and drying calculations are requested with the DE card, the changes made to the mesh, if any, are listed prior to the start of each time step. This information is listed whenever wet/dry checking is done (after every set of LI iterations, as specified on the DE card).

The information lists any element or node numbers which have been removed from or added to the mesh geometry since the last wet/dry check. Here is a typical example with three quadrilateral elements removed:

```
THE FOLLOWING ELEMENTS HAVE BEEN ELIMINATED
  23   24  145
THE FOLLOWING ELEMENTS HAVE BEEN ADDED
THE FOLLOWING NODES HAVE BEEN ELIMINATED
 118  119  124  125  561  562  563  566  567  568
 572
THE FOLLOWING NODES HAVE BEEN ADDED
```

Note: When an element or node is removed from or added to the mesh, the total number of **ACTIVE NODES** and **ACTIVE SYSTEM EQUATIONS** for this iteration will have changed from that of the previous iteration.

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

Time Step Revisions

When revisions are made during a time step, the input run control parameters in the full results listing file are rewritten. After this listing, the run continues as usual.

Note: No parameters are explicitly flagged as being revised. You must manually compare with the previous run control parameters to find the differences.

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

Steady State And Dynamic Simulation Nodal Results

This section of the full results listing file contains the latest nodal results of the simulation run. RMA2 will write the nodal results after the final time step has converged, and also at the iteration or time step interval specified on the TR card.

A "Column of Data" in this context consists of the headings ***NODE****,* ***X-VEL****,* ***Y-VEL****,* ***DEPTH****, and* ***ELEV****.*

The nodal results are listed as three columns of data, each containing node numbers, along with each node's *x* and *y* velocity components, depth, and water surface elevation. The three columns of data are demarcated by the *single* (sub)column with the heading **NODE**. 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, then moving to the top of the third column of data.

This section also contains **FLOWS AT NODES FOR 1-D ELEMENTS** and the **TOTAL VOLUME IN STORAGE BY ELEMENT TYPE**, which lists the fluid volume in storage for each material type you have defined.

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

Understanding The Summary Listing File

The summary listing is simply a brief synopsis of information for any nodes which you specify on TRN cards (see "Summary Results Listing" on page 27 for details on how to obtain a summary listing). For each node, the summary listing file will provide a table for each time step hour, which contains the

- *x* component of velocity
- *y* component of velocity
- total velocity
- flow
- depth
- elevation

The nodes for which data is requested are provided on TRN cards. A typical TRN card, requesting four nodes, will appear similar to the following example.

Example: Request Nodal Summary Tabular Listing

```
 .
$L      0   62   60   64   0   3   1   0   0   0
TRN   94   96   2472
 .
```

An illustrative example of a summary listing file (one for a steady state simulation, and one for a dynamic simulation) is shown below.

All requested nodes (specified on TRN cards) are summarized in one table for the steady state results (Hr=0).

For dynamic simulations, there is also one table per node that reports results for each time step (ie., hydrograph).

The Summary Tables can be fed into a Spreadsheet program to make plots.

```
 RMA2 TITLE:  Example Steady State Problem.

 RESULTS AT THE END OF     0 TIME STEPS...
 TOTAL TIME =     0.000000 HOURS... ITERATION CYCLE =    5

 NODE  X-VEL   Y-VEL TOT-VEL  FLOW  DEPTH  ELEV   CORD(X-Y)
       (FPS)   (FPS)   (FPS)  (CFS)   (FT)  (FT)     (FT)
   94   1.967  -0.106   1.970   0.0  3.881 3.881  472.9  329.5
   96   0.558  -0.030   0.559   0.0 13.928 3.928  468.4  286.7
2472  -0.134   0.524   ...
  . . .
```

```
 GFGEN TITLE=Houston-Galveston 1D/2D Ship Nav Study
 RMA2 TITLE= Dynamic, Base 1-D/2-D RMA2 Run. Hotstart hr=48

               HYDROGRAPHIC FOR NODE  2472
 TIME   X-VEL   Y-VEL    DEPTH   WS-ELEV    VEL-MAG   NDRY
 48.0   -.266   .610    39.987    99.987       .665      1
 49.0   -.601  1.310    40.148   100.148     1.441      1
 50.0   -.807  1.732    40.301   100.301     1.911      1
 . . .
103.0   -.751  1.572    40.339   100.339     1.742      1
--> MAX VEL FOR NODE= 2472  Vel= 2.34 Hr= 101.0 Depth= 40.14
```

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 RMA2 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.

If RMA2 has been run on one computer (a 32-bit unix workstation for example) and the SMS graphic user interface is on a sperate machine (a windows PC), then it is possible to move the binary solution file directly to the SMS environment to view. This is possible if the binary formats are compatable and the file was tranferred using a standard file transfer protocal (ftp) in *binary mode.* If SMS recognizes the file, proceed, otherwise there is an incompatabilty and utility programs (BIN2ASC and ASC2BIN) will be required to move from one computer to the other.

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

- *x* component of velocity
- *y* component of velocity
- Depth
- Water surface elevation
- A flag to mark as wet or dry (NDRY)

The solution file also contains the material type number for each element, which is written as negative if the entire element is dry.

It is advisable to examine contours of velocity magnitude, water surface elevations, and velocity vectors at every time step. The following checklist of items should be addressed.

- Are there signs of irregular or oscillating velocity magnitudes?
- Are there any two delta-X instabilities present?
- Are the directions of the velocity vectors reasonable?
- What is the maximum Froude number?
- Have irregular boundaries occurred (such as when wetting and drying)?
- Are some elements abruptly changing between active and inactive states (wetting and drying)?
- Are the boundary conditions specified correctly?

Advanced Techniques

Using Special Elements

One-dimensional element characteristics are defined using GN and GW cards with the N option; normally done in the geometry file. Refer to the GFGEN geometry manual for more details on creating one-dimensional elements in the geometry file.

SMS Note: As of this writing, in order to allow FastTABS and/or SMS to recognize one-dimensional element widths and side slopes, you must use a GN card with the N option to specify the *x* and *y* coordinates and the bottom elevation, and a GW card with the N option to specify the surface width, side storage width, and left and right side slopes.

There are 5 types of special elements:

- The Basic One-dimensional Element
- Transition Elements
- 1D Junction Elements
- 1D Control Structures
- 2D Control Structures

Each of these types has a specific purpose.

Using Basic One-dimensional Elements

The process of incorporating and using one-dimensional elements is not a difficult concept, but you may want to consider the pros and cons.

Pros

- One-dimensional elements provide a very computationally efficient means to extend the mesh domain in order to move the boundary conditions away from the study area.
- It may be useful and informative to develop a preliminary mesh composed predominantly of 1D elements prior to going to developing a detailed mesh.

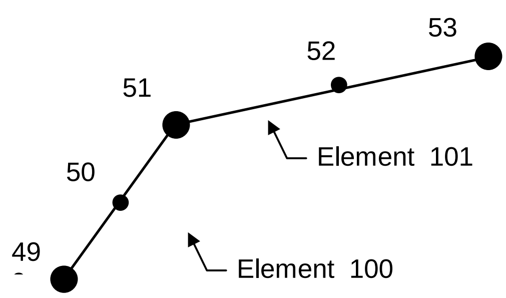
Cons

- Older versions of the TABS sediment model could not handle one-dimensional elements. (Problem rectified in 1996 with SED2D version 2.0 and higher)
- At the time of this writing, the graphical user interface, SMS version 6.08 or higher, permits limited editing, reordering, and parameter modification for one-dimensional elements. However, there have been instances of "lost 1D information" after manipulating information in SMS. This adds a degree of "manual" editing to the modeling process and opens a Pandora's box of potential error. Users beware.

1D Connection

A 1D element is defined as a 3-noded element that may be either straight or curvilinear (isoparametric). The figure below shows the elemental connections for two elements, number 100 and 101. The 1D element has only 3 nodes with the remaining 5 positions on the GE card set to zero. Any material type between 0-900 is valid for a basic one-dimensional element, this example shows the material type to equal 3.

Example

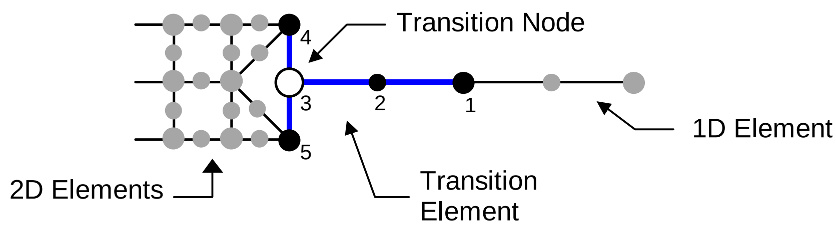


GE	**100**	**49**	**50**	**51**	**0**	**0**	**0**	**0**	**3**
GE	**101**	**51**	**52**	**53**	**0**	**0**	**0**	**0**	**3**

Using One-dimensional Transition Elements

Transition elements are required to connect one-dimensional elements with two-dimensional elements.

The T-shaped 5-node element connection table is shown in the figure below. The figure illustrates a suggested technique for transitioning the two-dimensional resolution down to one-dimensional. Element material type (IMAT) numbers for transition elements may be between 0 and 900. It is important to remember to maintain an angle that is very close to 90 degrees at nodes 4 and 5 in order to prevent "leakage". A GN card with the N option must be provided for nodes 1, 4, 5, and also for node 3, the Transition node. The bottom elevations of nodes 3, 4, and 5 must be identical in order to conform to the trapezoidal channel assumption. The coordinates of the transition node must be exactly half way between nodes 4 and 5.



GE elem# 1 2 3 4 5 0 0 0 imat#

Philosophical View

A clear way of developing a valid transition element is to take a philosophical view. This view may be later extended to proper transitions in the 3D model TABS-MDS (formerly known as RMA10).

The transition element must take on all of the assumptions and restrictions of both the 1D formulation and the 2D formulation. These assumptions and restrictions are summarized below:

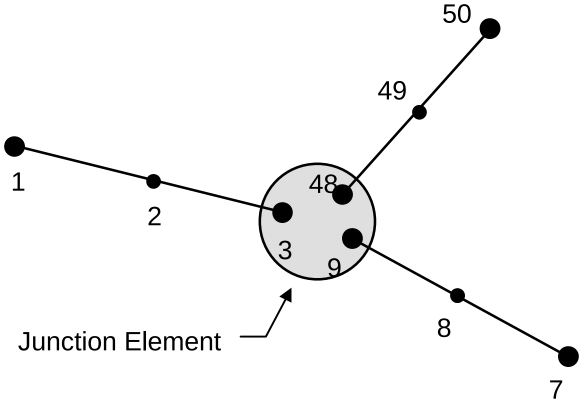
Formulation	Assumption	Restriction for Transition
1D	Lateral uniform velocity with some magnitude and direction	The direction of the velocities at nodes 3,4, and 5 must be the same. Therefore, the shorelines on either side at nodes 4 and 5 must be nearly parallel.
1D	Trapezoidal shape with uniform depth	The 2D cross section as defined by nodes 4 and 5 must be of uniform depth.
2D	Vertical side boundary walls	The 1D parameters for node 3 must have no side slopes.
2D	Width is defined by distance between nodes 4 and 5	An initial nodal width must be defined for node 3. *RMA2 makes an internal correction to insure this matches the 2D width.*
2D	Earlier version of RMA2 did not allow for 2D off-channel storage.	IF no 2D off-channel storage is used, then the off-channel storage for node 3 must be zero.

Using One-dimensional Junction Elements

Junction elements provide a means to connect two or more one-dimensional elements together at a central location.

Assume there are *n* one-dimensional elements connected at a junction. At the junction, each element has 2 degrees of freedom; water surface elevation and discharge, for a total of 2*n* degrees of freedom. A well posed mathematical problem will be established if one degree of freedom is “constrained” for each node at the junction. Therefore, we need *n* equations for the junction to provide these constraints.

In the illustration below, three branching channels are represented by one-dimensional elements forming a junction element composed of three nodes: 3, 9 and 48. Each of these nodes may have unique width, side slopes, elevations, and storage widths, but they must all have identical *x* and *y* coordinates.



GE elem# 3 9 48 0 0 0 0 0 901 ≥ imat# ≥903

Any time three or more one-dimensional elements intersect (8 is the maximum), RMA2 needs to be told how to interpret this junction. The material type controls the type of junction. There are three types of junctions:

- Water Surface Junction (IMAT = 901)
- Total Head Junction (IMAT= 902)
- Momentum Junction (IMAT= 903)

You may control the type of junction by specifying a material type (IMAT) of 901, 902, or 903 for the junction element. The material type is specified on the element's GE card.

Note: The material type (IMAT) for a junction element must be either 901, 902, or 903.

Water Surface Junction

The first type of junction is a "water surface junction" (IMAT = 901). This mandates that mass is conserved, and that all water surface levels will match at the junction.

Total Head Junction

The second type of junction is a "total head junction" (IMAT = 902). This mandates that mass is conserved, and that all total energy heads match at the junction.

Momentum Junction

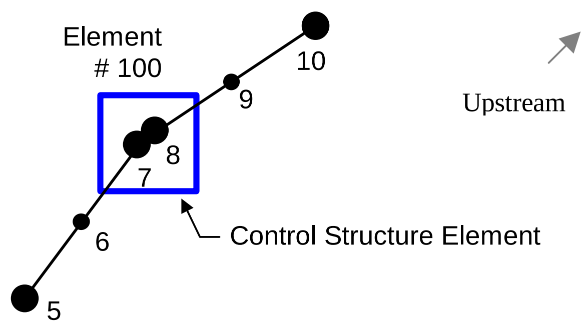
The third type of junction is a "momentum junction" (IMAT = 903). This mandates that mass is conserved. However, the first two nodes in the element connection table of the junction element define the primary channel within which momentum will be conserved. The bottom elevation, width and side slopes should be the <u>same</u> for these two nodes.

Using Control Structures

Flow control structures are a means by which you can simulate conditions (weirs, gates, dams, etc.) which were previously impossible to accomplish with RMA2. For this version of RMA2, flow control structures are permitted both for one- and two-dimensional sections of the mesh. There are graphical visual benefits to the 2D variety.

1D Control Structure Example

A 1D control structure element consists of two corner nodes, normally collocated with the same (x,y) coordinates using GN cards with the N option. These two nodes define the structure at two adjoining one-dimensional elements. The type of flow to be associated with the control structure element is defined using the FC card. To create a control structure element, the element material type (IMAT) must be greater than or equal to 904. Below is a typical example of a 1D control structure element and the GE card defining it.



GE 100 8 7 0 0 0 0 0 imat ≥ 904

Note that the GE card connection order for a control structue element defines the nominal upstream node (first) and the downstream node (second) on the RMA2 flow control card (FC card). Nodes 8 (upstream) and 7 (downstream) in the figure are shown at slightly different locations only for the purpose of illustration.

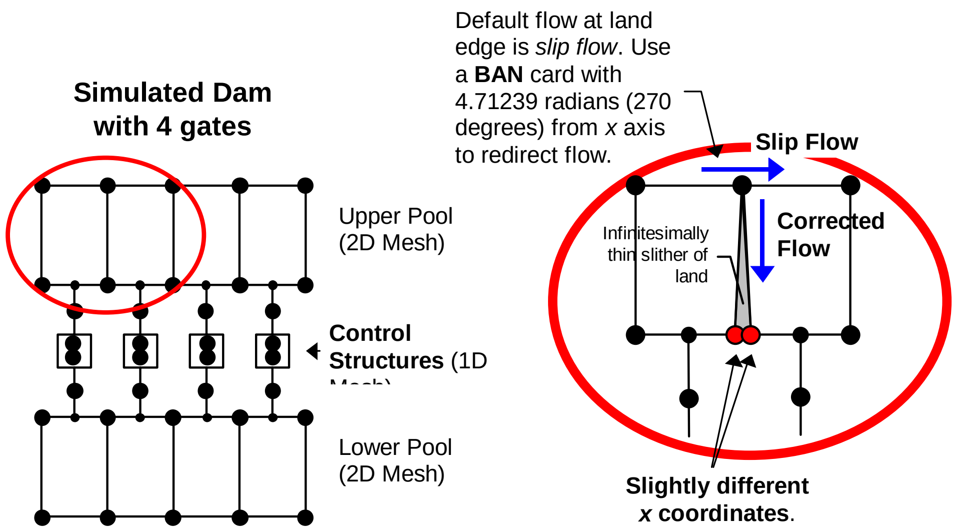
When constructing a 1D control structure element in SMS, both nodes will have the exact same location (the GNN card for each of these two nodes will have the same (x,y) coordinates). It is advised to manually inspect the saved files from SMS to insure the upstream/downstream convention was properly defined.

However the user can functionally relate any two 1D corner nodes together as a control structure. The nodes need not be at the same (x,y) location, they could be at opposite sides of the mesh. Therefore, the user must use care when manually creating or altering the connection of a control structure, since the RMA2 model will accept a typographical error as a “worm-hole” for water. This feature may be useful when constant pump diversions are move over some distance.

Note: The element connection order for a control structure element defines the nominal upstream node (first) and downstream node (second) on the FC card.

Simulating Spillway Gates with 1D Control Structures

There have been some lessons learned regarding 1D control structures. For instance, if you wish to model several gates composing a dam, as shown in the left figure below, several side-by-side control structures might be used to separate the upper and lower pools. A few advanced modeling techniques must be applied, as indicated by the zoomed figure on the right. To "trick" the model into handling this situation, the *x* coordinates of the flagged corner nodes are separated by approximately 0.20 ft to create a slight "land" boundary between the two elements. The BA card with the N option is required to set the angle to 4.712 radians for each interior two-dimensional corner node approaching and exiting the structure.



Large dot are corner nodes,
Small dots are Transition nodes.

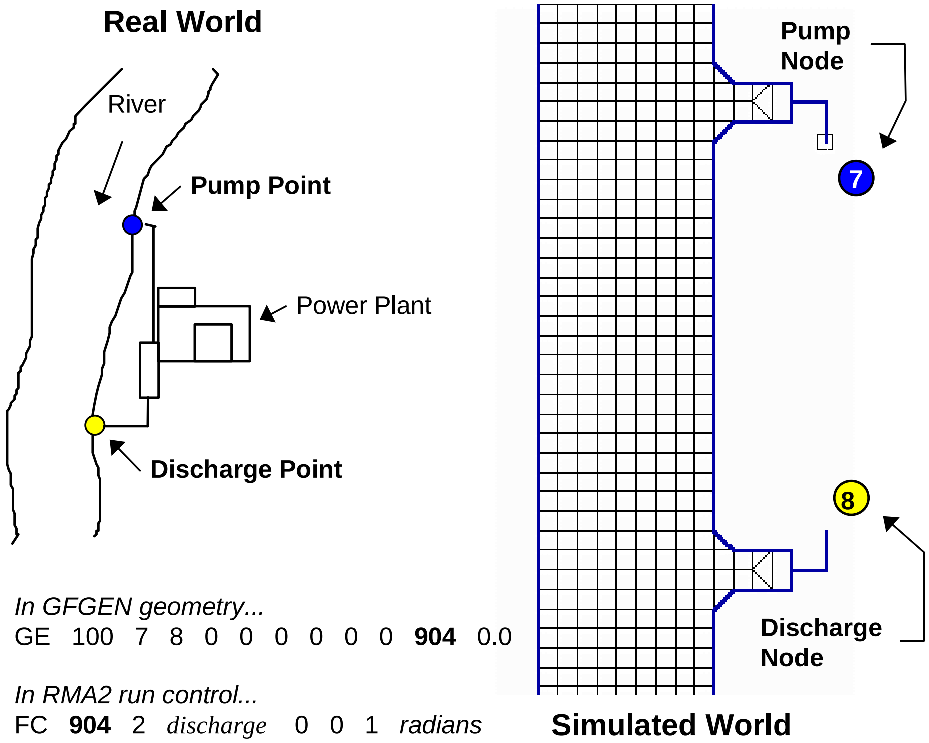
An alternative to this approach is to use a series of two-dimensional control structures.

Pumping And Discharging Using 1D Control Structures

To form a normal 1D control structure element, both nodes usually are located close to one another. However, it is theoretically possible to create a "pump" and "discharge" scenario if the nodes defining the control structure are placed at different locations. This technique has been applied to a simulation involving a power plant, where water is pumped over a condenser for cooling, then discharged back to the estuary.

To accomplish this scenario and specify the discharge, add a Type=2 control structure element to your mesh, separating the two nodes which constitute the control structure element as needed. Place the first at the *pumping* location, and the second at the *discharge* location. Also, when constructing the mesh, use smooth boundary break angles, and orient both the pump and discharge nodes in the same direction. The transition from the two-dimensional section to the one-dimensional section containing the control structure must be made following the rules for combining two-dimensional and one-dimensional elements, as described in "Using One-dimensional Transition Elements" on page 72.

Such a schematization will work well for a constant pumping rate. However, if the pumped discharge has a transient rate and there is some lag between the pumping point and the discharge point, then the system between the 2 points must be schematized as well. This can be accomplished by initially defining the structure type and modifying the rate on the FC card during the transient simulation.



2D Control Structure Example

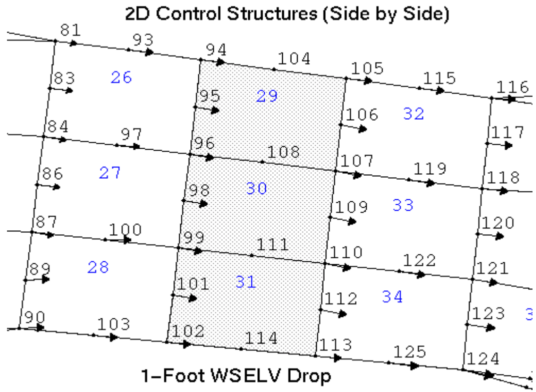
A two-dimensional Control Structure element has a unique connection table and an IMAT value ≥ 904. A two-dimensional control structure has some unique construction rules, but they are easier to use than their 1D counterpart, particularly for simulations which require side-by-side structures. In addition, there is a definite visual (graphical) advantage in using 2D control structures.

At the time of this writing, SMS versions prior to 6.08, cannot create the 2D control structure. The alternative is to build the 2D quadratic element mesh with SMS, taking care to build the element orientation such that the upstream side of the element lies corresponds to the first 3 nodal positions of the element connection table, assign these elements with a unique material type, renumber the mesh, and save the geometry file.

What makes the connection table for the 2D Control Structure element unique is that the element has to be numbered such that the upstream edge lies in nodal positions 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 midside nodes connecting the “upstream” side of the structure with the “downstream” side (4^{th} and 8^{th} nodal position in the connection table) of the structure must be set to zero.

In the example figure given below, there are three shaded elements (29-31) that represent three 2D control structures, located side-by-side, that span from bank line to bank line. To convert these elements to control structure elements, the placeholders for midside 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.

Example: 2D Control Structure



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**

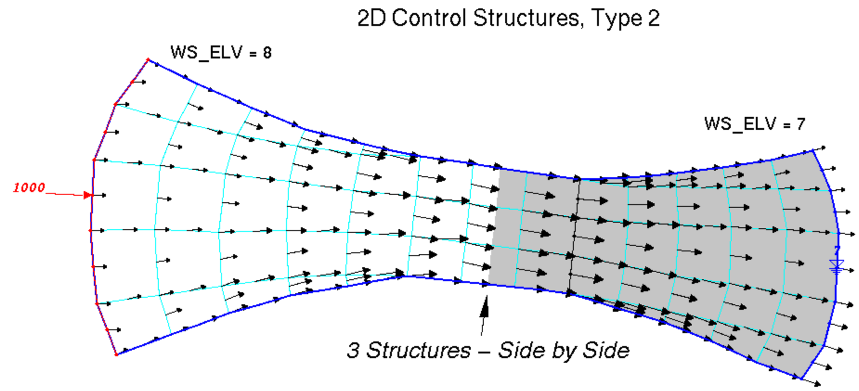
The table containing the RMA2 run control file, illustrates how the structure based on head loss, Type 2, was applied. The entire mesh was assigned as material type 1 (except for the structures) and a flat bottom of -5.0 ft. The initial water surface elevation was set to 7.0 ft, with an inflow of 1000 cfs, and tail water of 7.0 ft elevation. The FC card was designed to provide about a 1-ft drop across each structure for this condition. It is important to note that for a 2D structure, the discharge referenced in the equations is per unit width. The FC card defines the type 2 structure, as flow is a function of head loss across the structure.

per unit width q = FLZ3 + FLZ4 (HN1-HN2-FLZ5)**FLZ6 in direction of FLZ7

Where (HN1-HN2) = 1.0 ft is desired for an inflow rate of 1000 cfs, and FLZ6 = 1.5 was arbitrarily chosen for this problem. The FLZ5 term can be viewed as a critical head difference required to initiate flow. For this example, assume FLZ5=0 because each 2D control structure element is 43-ft wide for this problem, the unit q = 1000/(43*3) = 7.752 cfs/ft. Now determine the remaining coeffiicient (FLZ4) on the FC card,

q = FLZ4(1-0)**1.5, solve for FLZ4= 7.752

The simplied equation becomes: $q = 0 + 7.752\,(H1 - H2)^{1.5}$



Upon completion of the RMA2 simulation, SMS was used to examine the solution, and the summary print file was examined, as shown above and below. The water surface on the upstream side of the structure is indeed the value of 7.9 ft.

Example of RMA2 Run Control (BC) file for SS, 2D Control Structure

```
T3  Type 2, 2D Control Structures, Flat Bottom Test
SI  0
$L  0  0  60  64  0  3  59
$M  4
TR  1  -1  1  0
TRN  94 95 96 98 99 101 102 1 05 106 107 109 110 112 113
GCL  1      1    3    5    7    9    11    -1
GCL  2     209  203  197  191  185  183    -1
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
TZ  0  0  0  0  0
TI  8  0  0.000001  0.00
FT  17
IC  7.0  0  0.25
CO No eddy viscosity or n-value needed for Structures
EV   1    2  2  2  2     0.025
FC  904  2  0  7.752  0  1.5  -.2
PE    1   20   0.25   1 1 1 1
BQL   1 1000   6.2295
BHL   2  7.0
END  Simulation at time =  0.00
STOP
```

Excerpts from the RMA2 Summary Print File

```
 RESULTS AT THE END OF      0 TIME STEPS...
         TOTAL TIME =      0.000000 HOURS... ITERATION CYCLE =    8

 NODE     X-VEL    Y-VEL    TOT-VEL     DEPTH      ELEV
          (FPS)    (FPS)    (FPS)       (FT)       (FT)
    96    0.588   -0.119    0.600      13.000     8.003      Upstream
   107    0.637   -0.129    0.650      11.995     6.995      Downstream
```

Resuming A Stopped Simulation

Resuming a previous simulation is known as Hotstarting. In order to Hotstart, you must have specified the creation of a Hotstart file in the previous run. The starting point, or initial conditions, for the Hotstart will be non-zero velocities and a sloping water surface elevation for all active elements.

If a hotstart file output was requested, the file is rewound and variables and their derivatives are saved at the end of each iteration or time step, as specified by the IBRO variable on the TS card. If IBRO is left to default, then the Hotstart file is rewound and information is saved only at the end of the last iteration or time step. Therefore, for default conditions, the last computed iteration is saved to the Hotstart output file.

To resume a simulation, the modeler must take extreme care to supply RMA2 with enough starting information so that the initial condition in the hotstart record matches the appropriate iteration and time step in the input run control file (a.k.a. boundary condition file). If the intent is to do more iterations within a given transient time step, then the MBAND). parameter on the TZ card must be activated. MBAND is the logic control variable that directs the program to either skip to the next time-step (MBAND=0) or remain at the last time-step of the previous run (MBAND=1) for the first computations of the Hotstart run. Always check and double-check a restarted simulation.

If multiple records of hotstart information were saved to the hotstart output file. The modeler must request which saved record will serve as the initial condition for the re-started run. These restart retrieve parameters are located on the RSC card, page 185, and include time, iteration control for both the u-v-h and vorticity computation.

Why Hotstart?

There are several reasons you may wish to Hotstart RMA2. Here are a few.

- Remove Effects of the Initial Conditions — Eliminate model spin-up, then continue
- Data Security — Provide a safeguard/backup in case of machine failure, etc.
- CPU Availability — You may have a limited amount of CPU time available for each job run
- Data Management — If a single run fills your disk capacity, then breaking the simulation into pieces may be necessary
- Hind site - Fore site — Provide a means to correct a minor oversite

How To Hotstart

Hotstarting consists of two procedures:

- Creating A Hotstart File
- Restarting Using An Existing Hotstart File

You may Hotstart as many times as you like. Remember that in order to Hotstart, you must have saved a Hotstart file from the previous run.

Creating A Hotstart File

Saving a Hotstart file is simple. You need only to specify a positive value for NLL on the $L card. When you run RMA2, you will be prompted to enter a name for your Hotstart output file.

If you wish to create a multi-record hotstart file which contains a hotstart save at designated points along the simulation, then specify the IBRO variable on the TS card. For more details, see the TS card, on page 195.

Restarting Using An Existing Hotstart File

Here is the Restart procedure:

1. Specify that the initial conditions be read from an existing Hotstart file ($L card, IHOTN = any positive integer).
2. Set the initial condition print option, (TR card, IPRT=2)
3. Specify the mid-iteration switch (TZ card, MBAND). MBAND is the logic control variable that directs the program to either skip to the next time step (MBAND=0) or to remain at the last time step of the previous run (MBAND=1) to make additional dynamic iterations at the final time step of the prior run. MBAND=1 activates the time terms in the equations.
4. If refinement of additional iterations of the previous steady state solution is desired, set the steady state iteration counter to a non-zero value (TI card, NITI>0), and the mid-iteratation for dynamic to zero (TZ card, MBAND=0). RMA2 will compute this number of additional steady state iterations toward the previous solution.
5. If refinement of more iterations at the old time level of a previous dynamic solution is desired, set the steady state iteration counter to zero (NITI), and the desired number of additional dyanamic iterations (NITN) (TI card, NITI=0, and NITN>0). Set the mid-iteration switch to activate additional iterations (TZ card, MBAND=1).
6. If Hotstarting a dyanmic simulation, the NITI variable on the TI card should always be zero. If it is positive, then a steady state simulation will be attempted which will negate all dynamic spin-up.
7. Specify the number of boundary condition sets (END cards) to skip in the run control file (TZ card, NSTART). Typically set NSTART equal to the last time step from which you are using as a restart initial condition, **plus one, minus MBAND** (see #3 above).
8. If a multi-record hotstart file is being used, then specifically provide the hotstart retrieve parameters (RSC card, HOT_TET, IT_UVH,IT_VOR).

Restarting Check Points

An example is provided below which illustrates the necessary check points you should examine to assure that the Hotstart initial conditions have been read in correctly, and that the correct boundary condition set is active as the model resumes the simulation. The convergence parameters and active nodal statistics from the last time step (in this example, hour 120, time step 5) of the Coldstart run are circled in the first box below. As highlighted in the second box, you should modify the $L, TI, TR, and TZ cards in the RMA2 run control file to prepare the restart parameters.

As a check point in the restarted simulation, verify that the active nodal statistics are the same as the in the previous run. One last check is to verify that the boundary condition set for the first time step in the Hotstart is valid. An example of the checkpoint procedure for a hotstart (restart) of a dynamic simulation is provided below.

Example: Re-Starting Check Points

Previous Run

```
RESULTS AT THE END OF    5 TIME STEPS...
      TOTAL TIME =   120.000000 HOURS... ITERATION CYCLE =  3

         CONVERGENCE PARAMETERS

        DF         AVG  CHG        MAX CHG     LOCATION
         1          0.0001          0.0004        259    X-VEL
         2          0.0001         -0.0003        283    Y-VEL
         3          0.0000          0.0001        309    DEPTH

          ACTIVE NODAL STATISTICS FOR THIS ITERATION

 NODE  XVEL-MAX    NODE  XVEL-MIN   NODE  YVEL-MAX   NODE   YVEL-MIN
  354   10.952      164  -10.021     158     1.850     52    -12.285

    NODE  ELEV-MAX    NODE  ELEV-MIN   AVE-ELEV  NODES ACTIVE
     326   121.356      18   110.845    116.797         932
```

Modificatio To Run Control for Hotstarting

```
  T3 Vital information for Hotstart run control
  $L 63 62 60 64 0 3 59
  TR 2 -99 1 0
  CO NSTART=6, and MBAND=0 because NITI=0 for this dynamic HOTSTART
  TZ 24 720 999    6    0
  TI 0  5  .0001  .001
  BQL 1 500000 6.17
  BHL 2 111
1 END                        Steady State hr=0
  BQL 1 535000 6.17
2 END                        hr =  24
  BQL 1 575000 6.17
3 END                        hr =  48
  BQL 1 575000 6.17
4 END                        hr =  72
  BQL 1 600000 6.17
5 END                        hr =  96
  BQL 1 650000 6.17
6 END                        hr = 120   ← Previous run stopped here
  BQL 1 700000 6.17
7 END                        hr = 144
  BQL 1 750000 6.17
8 END                        hr = 168
```

After Hotstart Run

```
..... INITIAL CONDITIONS .....

   RESULTS AT THE END OF     0 TIME STEPS...
          TOTAL TIME =   120.000000 HOURS... ITERATION CYCLE =    3

          CONVERGENCE PARAMETERS

        DF         AVG  CHG        MAX CHG     LOCATION
         1          0.0000          0.0000          0    X-VEL
         2          0.0000          0.0000          0    Y-VEL
         3          0.0000          0.0000          0    DEPTH

          ACTIVE NODAL STATISTICS FOR THIS ITERATION

   NODE  XVEL-MAX    NODE  XVEL-MIN   NODE  YVEL-MAX   NODE  YVEL-MIN
    354    10.952     164   -10.021    158     1.850     52   -12.285

   NODE  ELEV-MAX    NODE  ELEV-MIN  AVE-ELEV  NODES ACTIVE
    326   121.356      18   110.845   116.797        932

 *** BOUNDARY CONDITIONS DEFINED FOR TIME = 144.0000 HRS

    BQL 1 700000 6.17
```

Common Causes For Hotstart Failure

Invalid Initial Conditions

The initial condition read does not match the specified starting boundary condition. This results from a miss-match between the value of NSTART on the TZ card and the requested iteration/time step on the hotstart input file that is used as the initial condition.

The variable NSTART on the TZ card is used to "skip" through the RMA2 run control file, counting the number of "END" cards. After the skipping has been performed, then RMA2 reads the next BC set as the one to use for its first computation.

A simple formula, such as the one described, may help determine the appropriate value.

Equation 12

$$NSTART = 1 + NTSP - MBAND$$

where

1	represents the Boundary Conditions for steady state.
NTSP	number of previous dyanmic time steps simulated (ie. If the model previously ran 25 hours at ½ hr time step, then NumTStep = 50
MBAND	either a 0 or 1 as described on the TZ card.

Invalid Restart In Mid-Iteration Of A Dynamic Simulation

Failure to set MBAND correctly on the TZ card could result in a hotstarted simulation with noticeable noise in the water surface and velocity time series plotted data.

Insufficient Knowledge Of The Contents Of The Hotstart File

Occasionally we are confronted with not being familiar enough with a given project to know what times were saved to the hotstart file. The specific details regarding each record written to the hotstart file are recorded on a separate file named "r2hot.dat" and also to the "screen" while the run is in progress. Look for text similar to the example shown:

WRITE HOTSTART RECORD=   1. HR= 0.50 UVH_ITER = 3 VOR_ITER = 0 TS=   0

There are several utility codes which can echo the times and iteration specifics of each record present on the hotstart file. The utility code R2_HOTFIX can be used for this purpose. This information is useful when setting the RSC card. For an example, see "Determining the Records Available on a RMA2 HOTSTART File" on page 233.

Mesh Modifications Made Between Runs

A common mistake is to modify the mesh after a RMA2 run, and attempt to use the old hotstart file as an initial condition for the newly modified mesh. Even a minor depth change is enough to cause divergence. As a rule, beware of modifications that alter the geometry. Altering an input variable on any of these “G” cards should be avoided: GE, GN, GS, GT, GW, or GZ. Some subtle changes are permitted, but as a rule you cannot create or delete nodes or elements, modify the connection table, or change the bottom elevation. Remember by modifying the marsh porosity parameters (DM type cards), you are actually changing the geometry.

What to do if you get caught?

What do you do if you have an *almost* wonderful dynamic simulation that took 3-days to finish and you have discovered that there are bad bathymetry values in a section of the mesh? Before you completely discard the old run, try using R2_HOTFIX, a utility program written by TABS-MD personnel, that repairs the hotstart file. This program permits the modeler to make bathymetric changes to a new mesh, repair the old faulty hotstart file, and create a new “fixed” hotstart file. For more information, see the section, R2_HOTFIX, page 233.

Specifying Units

The default for RMA2 is English units. However, if you wish to operate in metric (SI) units, the following is required:

- Use an SI card to specify metric units and position it immediately after the $L card in the GFGEN geometry file.
- Run GFGEN to create a new binary geometry file containing data in SI units to be used as input for RMA2.
- Use an SI card in the RMA2 run control file (boundary condition file) instructing RMA2 to use SI units.
- Provide all mesh revisions (GN card, GS card, GZ card, and GW card, etc.) in meters.
- Non-default values for wetting and drying on the DE card and DM card must be entered in meters.
- Automatic roughness by depth (RD card) must reference depth in meters.
- Turbulent Exchange coefficients (eddy viscosities) must be entered in SI units (Pascal-sec) if using an EX card, EY card, or EV card.
- Rainfall/evaporation units must be entered in cm/hr.
- Non-default fluid density values must be entered in SI units (kg/m^3) on the FD card.
- Use SI units for Coldstart initial conditions on the IC card.
- Use SI units for *all* boundary condition specifications.

Customizing The Solution File

RMA2 provides ways to customize the solution file. These include the TO card and/or the Hotstart technique.

Eliminating Initial Condition Contamination

There may be times when you do not want to save any part of the solution that may have been contaminated by the initial conditions (spin-up). There are three basic ways to exclude this time period from the solution file:

- Run the simulation through spin-up, then Hotstart (see "Resuming A Stopped Simulation" on page 80.
- Use the TO card to specify the decimal hours you want to save. Only the specified time periods will be written to the solution file. However, if any instabilities cause the run prematurely stop, it is difficult to diagnose the problem without saving the solution file.
- Post process the solution file with the MERGAVG utility program to remove the spin-up section of the solution.

Note: The **MERGAVG** program is a utility available, which can assist with post processing the solution. See "Merging RMA2 Solution Files" on page 232 for details.

Customizing The Full Results Listing File

There are several options available for controlling the information that is written to the full results listing file. Customizing gives you control over the types of data which are written to the listing file, and how often. The TR card is used to specify what will be in the full listing.

When To Write To The Full Listing File

The full listing can become huge if not properly controlled. To reduce the file size you may request that data be written to the full listing either on an iteration basis, or a time step basis. This is done by assigning a value to ITSI on the TR card as follows:

- Positive integer values for writing at *iterations*
- Negative integer values for writing at *time steps*

The value you specify is used as the interval at which the data will be written. For example, if you specify a –3, the results and associated data at every third *time step* will be written to the listing file.

Tip: If disk space becomes limited, change the value of ITSI to –12, or some other larger negative number (such as –99). This will cause RMA2 to only write data for time steps at this specified interval (except for wetting and drying). Alternatively, to prevent writing a full listing file altogether, you might set the value of IOUT on the $L card to zero, and enter "`null`" at the results listing filename prompt.

Including Startup Conditions

There are several types of startup condition data, which can be included in the full results file. These are:

1. Echo of the run control data
2. Geometry information
3. Wetting and Drying/Marsh Porosity information
4. Elemental area information
5. Nodal specifications defined at startup
6. Roughness (Manning's n-value) assignments
7. Front width size and total equations

Suppressing Node And Element Data

If you are confident about your geometry data and you want to reduce the size of the full listing file, you can eliminate node and element data from the listing file. To do so, set IPRT on the TR card to zero. This will eliminate items 2 through 5 in the above list of startup conditions.

Suppressing Coldstart Initial Conditions

The convergence parameters written to the listing file for initial conditions at Coldstart are always zero because RMA2 has not yet obtained any solution. Therefore, it is uninformative to see this information. To eliminate the Coldstart initial conditions, set IPRT on the TR card to 1. This will also eliminate the echo of the convergence parameters at startup.

Requesting A Summary Of Nodal Results

It is sometimes convenient to request a summary results table for a set of nodes. This permits quick comparisons of model results to field data. It is particularly useful if you like to use spreadsheets to create plots.

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

- A table of the steady state solutions of *x* and *y* velocity components, depth, etc., for each node listed on the TRN cards
- A dynamic hydrograph of the above information for each node listed on TRN cards

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

SMS Note: Neither FastTABS nor SMS will create or modify TRN cards.

Note: None of the parameters on the TRN card will affect the summary listing. Data is written to the summary listing file after every time step.

Plugging In Mesh Modifications

Normally, mesh modifications are made prior to the RMA2 run, and no mesh modifications are necessary at the time of the run. Occasionally it is desirable to create or modify an existing mesh from within the RMA2 run control file. This ability allows for quick testing without having to go back through GFGEN.

SMS Note: Neither FastTABS nor SMS will not recognize mesh modifications made from within the run control file. If you change the number or location of nodes or elements in the run control file, the solution file you obtain after the RMA2 simulation run will no longer match the original geometry file.

Modifying Nodes

Nodes can be added to the mesh, or moved, by using a GN card in the run control file.

Adding Nodes

Any node which is added to the geometry must be connected to the mesh via an element. Nodes are typically added only when adding a new element. Each node must have a unique number. Unused numbers or voids in the numbering sequence is not desirable.

Corner Nodes

To add a corner node, modify the run control file and use a GN card with the N option. Assign a node number which is larger than the largest node number currently existing in the mesh, accompanied by the *x*, *y*, and *z* coordinates for the new node. Modify or create the element connection table using a GE card to include the new node in the mesh.

Midside Node

The RMA2 model requires quadratic basis elements (see "Element Types Supported" on page 5). To add a midside node, modify the run control file and use a GE card to describe the element connection table for the element to which the node belongs. Assign a node number which is larger than the largest node number currently existing in the mesh. If the midside node lies on an element edge that requires curving, then a GN card is also required.

Moving Nodes

To move an existing node, modify the run control file and use a GN card. Specify the node number of the node which you want to move, along with the new *x* and *y* location for the node.

Modifying Elements

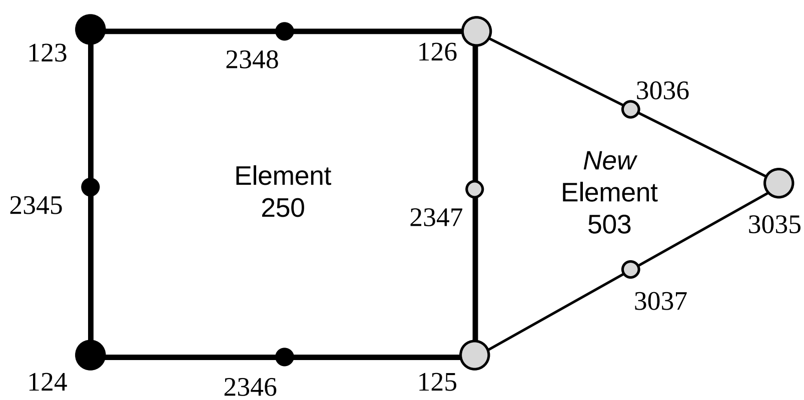
Elements can be added to or removed from the mesh, or the material type changed by modifying the run control file.

Adding Elements

Before adding an element, *all* corner nodes necessary to build the element must have been previously added to the mesh (see “Adding Nodes” above). It is imperative that you know all the corner and midside node numbers of the face of the element to which you are attaching the new element. You will need to include these numbers in the new element’s *connection table* (GE card). Start the connection table with a corner node and proceed around the element in a counterclockwise fashion.

The node numbers for the uncoupled midside nodes created for the new element must be unique. You will need to know any node numbers which are unused in the mesh. If there are none, the maximum node number, midside nodes included, of all the nodes in the mesh will be your starting point for numbering the new element.

Prior to modification,
The largest *element* number is 502.
The largest *node* number is 3034.



Example:

```
CO  Original element (Quadratic element)
GE  250  123 2345  124 2346  125 2347 126 2348    1   0.0
 .
 .
CO  New element (Triangular element)
GE  503  125 3037 3035 3036 126 2347    0    0    1   0.0
```

To add an element, insert a GE card in the run control file. Specify an element number not previously defined in the mesh, along with the list of nodes in a counterclockwise direction, and a material type.

Removing Elements

To remove an element, simply assign a material type of zero for the element. This can be accomplished by using the GT card, or alternatively with the GE card, to set the material type.

Note: Removing elements by setting the material type to zero does not *physically* change the geometry, but effectively removes the element by converting it to land. The element can be re-activated at any time by changing the material type to a value other than zero.

Changing An Element's Material Type

Normally, an element's material type is set in the GFGEN input geometry file. You can change the material type for the RMA2 run for an element or elements by using a GT card. Specify the element number along with the material type number you wish to assign.

Note: Any new material type numbers must have roughness and turbulent exchange coefficients defined, typically accomplished using an EV card.

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).

Curving Element Edges

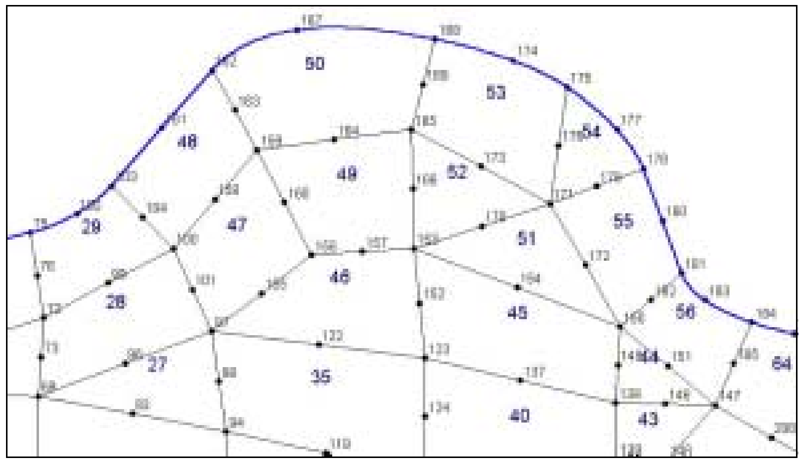
There are several reason why you may want to curve elements.

- Add aesthetics
- Add length to a section without adding resolution
- Represent meandering channels
- Reduce boundary break angles without adding additional resolution

Curving can be useful to help improve mass conservation and to achieve a more natural appearance along the mesh boundary. You may curve both one-dimensional and two-dimensional elements.

By default, a midside node lies exactly between the corner nodes comprising an elements side. To *curve* an element side, move the (*x*, *y*) location of the midside node from the default location. Curving is accomplished by adding a GN card to the geometry or to the RMA2 run control file, specifying the new *x* and *y* location for the midside node that lies on the element edge which is to be curved. The bottom elevation for a midside node is ignored because it is calculated from the elevation values of the two adjacent corners.

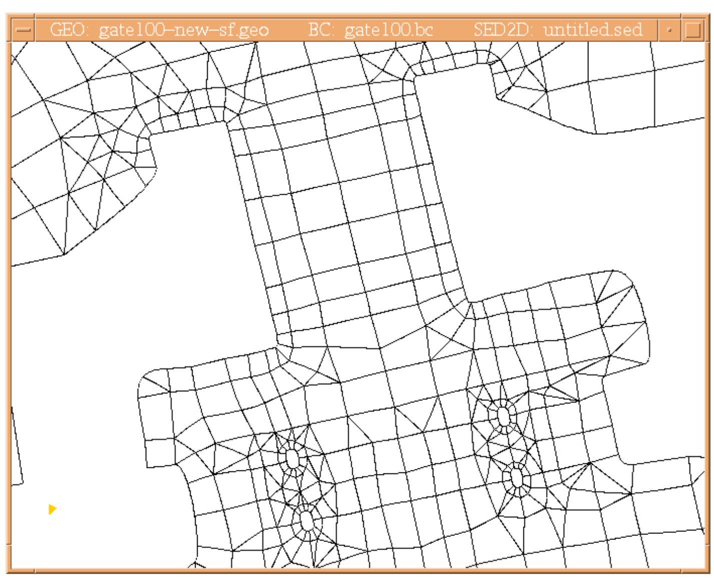
The example below shows a portion of a curved upper exterior of a mesh. Elements 29, 48, 50, 53, 54, 55, 56, and 64 within view are curved.



A semi-automatic means of curving all viable exterior edges of the mesh was developed in an utility code, “SLOPEFIX”. See the utilities section entitled,“Exterior Curving and Eliminating Bad Boundary Break Angles” on page 234.

New York South River Project Example:

The SLOPEFIX utility was initially used to curve the exterior boundary of this RMA2 flood gate study. Of note are the smooth corners and rounded bridge piers. After running RMA2, the “AMW warning” was flagged for a few locations. These areas required manual curving.



Note: Caution is warranted when curving interior element edges.

Warning: Be aware that curving element edges may result in an ill-formed element when the new location chosen for the midside node is too far away from where it would be normally.

Changing Bottom Elevations

Within the RMA2 run control file, the bottom elevation may be changed on the GZ card, or the GN card with the N option.

Warning: The GN card with the N option also specifies the *x* and *y* coordinates, as well as the bottom elevation, *z*.

Resizing The Mesh

RMA2 provides the capability to resize your mesh by applying scale factors. All *x*, *y*, and *z* coordinates of the mesh are multiplied by the XSCALE, YSCALE, and ZSCALE factors located on the GS card. To resize the mesh, specify values other than 1 for the *x*, *y*, and/or *z* scale factors. This technique has been used to convert from English to metric, or to double the size of a study area for the sake of sensitivity.

Changing An Element's Eddy Viscosity Tensor

The direction of the eddy viscosity tensor for a given element (radians, counterclockwise from the +*x* axis) may be modified by using the GV card, or alternatively, the GE card.

Note: Changing the element's EV tensor is usually not necessary.

Advanced Boundary Condition Techniques

Many modeling applications are unique in their requirements. The modeler is responsible for making appropriate engineering decisions regarding the applicability of these techniques.

Changing The Direction Of Flow

All exterior boundary nodes that touch a land edge are automatically assigned an appropriate flow angle. Nodes at flow boundary condition locations, however, should be manually assigned flow angles which indicate the true direction of flow. The angle may be specified on the BA card, BQ card, BRA card, or BRC card.

Tip: Occasionally, you may need to force a flow angle for a particular node to be a given direction, such as when using side by side 1D control structure elements (i.e., multiple flood gates, etc.). A BA card may be used to directly specify the flow angle.

Changing The Speed Of The Current

An alternative to the conventional means of specifying the flow conditions at a boundary is the current speed/direction option. The boundary current speed, or velocity magnitude, is defined on the BS card. If a BS card is used, a BA card must also be specified to define the flow angle.

Note: The current speed/direction boundary condition is independent of the water surface, therefore the total flow across the nodes using a BS and BA card will fluctuate with the tidal signal.

Varying Discharge During A Simulation (Rating Curve)

Most rivers have a boundary rating curve defined from historical data. The BRC card allows you to supply the coefficients of an equation to provide a fit to that historical data. Once the equation is defined, RMA2 will compute the boundary discharge based upon the calculated water level for that time step

Revising Boundary Conditions During A Simulation

Revising boundary conditions can help start the run without divergence by wetting the entire grid and gradually obtaining the actual water surface through revisions.

The boundary conditions which were defined during the initial time step of the simulation will hold until you specifically reference them to change. A new boundary condition may be assigned in the middle of a time step after a REV card, or at the end of a time step after an END card.

The REV card allows you to revise the run control in the middle of a time step. The revision may reference the boundary conditions, time or print controls, roughness, eddy viscosities, wet/dry parameters, etc. However an error will result if you attempt to revise any boundary condition that has not been initially defined.

Note: Geometry modifications other than modifying the material type will likely result in a diverged solution.

Example: Revising during a steady state simulation

```
T3
$M
$L
 .
 .
 .
EV     1    500   500   500   500    0.03
TI     4      0   .01
 .
```

.
```
BQL    6  50000   -1.2
BHL    7      100
REV      Now lower coefficients and increase Q for 2 iterations
TI       2         0    .01
EV       1      300   300   300   300  0.027
BQL    6  75000   -1.2
REV      Now lower coefficients more and increase Q
EV       1      150   150   150   150  0.025
BQL    6 100000  -1.2
END        Marks the end of the time step
```

Solve for Hour 0.0

Solve for Hour 0.0 given new data

Solves for Hour 0.0 and writes the solution using EV = 150 and Q = 100000.

Boundary Permeability (Reflection/Absorption)

When a boundary is located within the tidal portion of a system and there is no tidal forcing from that boundary (i.e., there is no additional connection to the ocean via that boundary), then there will be a dynamic tidal response of the velocities and water surface at that boundary. The magnitude and timing of that response will be dependent on the tidal energy that has moved through the model form the other boundaries with tidal forcing and the geometry of the system outside the model domain beyond that boundary.

Using the method of characteristics from linear wave theory, the harmonic solutions over the model will cross that boundary as either a progressive or a reflected wave.

The reflection/absorption boundary condition is useful whenever a boundary condition cannot be extended to the head of tide or null point. This boundary condition has been found to work far better when applied to a one-dimensional element rather than to a two-dimensional element.

The reflection/absorption boundary condition (BRA card) is a special form of the rating curve (BRC card). It calculates the discharge based upon a calculated water surface elevation and the surface area of tidal storage beyond the boundary. The variable BETA on the BRA card is a decimal number that may vary between 0.0 for total absorption to 1.0 for total refection of the tidal wave.

Hotstarting using this boundary condition is a little unusual, in that the steady state water surface value (SSWSE) for the first node on the continuity check line referenced by the BRA card must be supplied to at least 4 places past the decimal. The variable SSWSE is used as the mean value about which the wave oscillates.

Bendway Correction (Vorticity)

Traditional RMA2 depth averaged calculations of flow around a bend frequently over predicted streamwise velocities on the inside bank of a river bend. When water flows around a corner, a radial acceleration is developed that forces the surface water to the outside of the curve and the water near the bed to the inside of the curve. This is the familiar “secondary” or “helical” flow pattern which causes sediment aggradation on the inside of curves and soil erosion on the outside of curves. RMA2 cannot normally adequately predict the effect of this behavior on the depth-averaged velocities. In order to improve predictions of depth-averaged velocities around curves, a secondary flow corrector, coined as bendway correction, option was added to RMA2.

The Principle Of Bendway Correction

The theoretical basis of the bendway correction was developed by Dr. Bernard of WES-HL for his depth averaged finite difference numerical model, STREMR. For additional information, reference US Army Engineers technical report HL-92-9, Bernard, R. S., and Schneider, M. L. (1992), "Depth-Averaged Numerical Modeling for Curved Channels".

The bendway correction is accomplished by first solving an additional equation for the transport of streamwise vorticity. Vorticity is a measure of rotation of flow. Streamwise vorticity at a point is the measure of rotation of the fluid about the axis in the streamwise direction of flow. Streamwise vorticity is in the vertical plane perpendicular to the direction of flow and is related to the radial accelerations that cause the helical flow pattern.

The transport equation for streamwise vorticity is:

Equation 13

$$\frac{\partial\Omega}{\partial t}+\frac{\partial\Omega}{\partial x}+\frac{\partial\Omega}{\partial y}=\frac{A_S\sqrt{C_f}\,|u|^2}{Rh\left(1+9h^2/R^2\right)}-D_S\sqrt{C_f}\,\Omega\frac{|u|}{h}+\frac{1}{h}\nabla\left(\nu h\nabla\Omega\right)$$

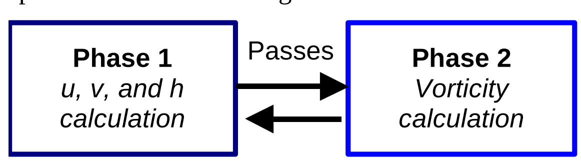
where Ω is the streamwise vorticity, $|\mathbf{u}|$ is the magnitude of the velocity vector, R is the local radius of curvature, C is the friction coefficient, h is the depth, $A_S = 5.0$, and $D_S = 0.5$. The units of vorticity are sec^{-1}.

The additional shear stress caused by the secondary, helical flow is then calculated from streamwise vorticity at each node. The components of this shear stress are added to the other terms (friction, slope, Coriolis) which occur in the governing equations.

The Bendway Correction Solution Scheme In RMA2

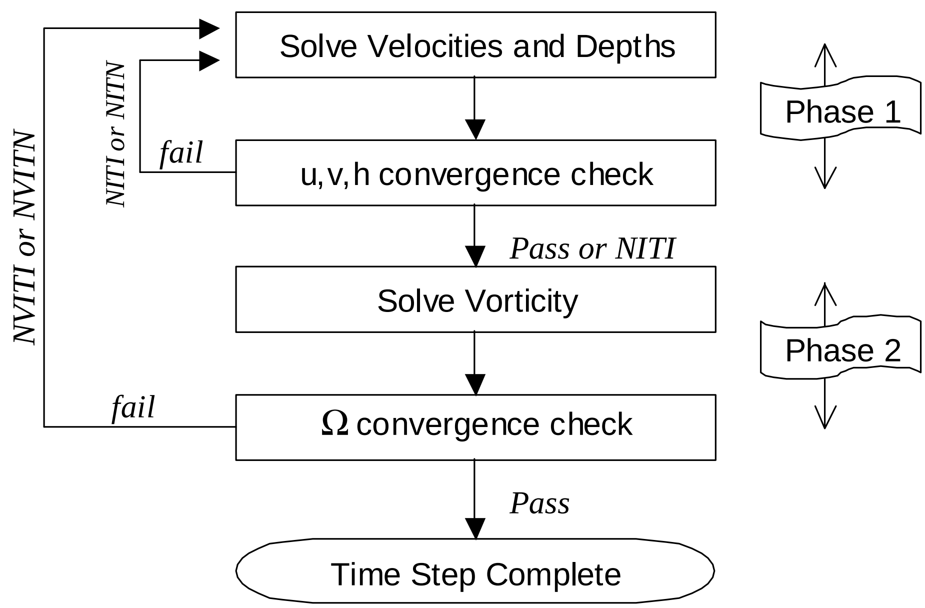
*The vorticity calculation in RMA2 is a two phase process: the traditional **u, v, h** phase, and the **vorticity** phase.*

The addition of the bendway correction required a number of changes to RMA2. In order to minimize computer storage requirements, it was decided to calculate streamwise vorticity separately from velocities and depth. The calculation of vorticity (phase 2) begins after the calculation of velocities and depth (phase 1) has converged within the non-linear Newton Raphson iterative scheme. The relationship between the two phases is shown in the figure below.



Velocity and depth are held constant during the vorticity calculations, and vorticity is held constant during velocity/depth calculations. By holding the velocity and depth constant for the vorticity equation, the equation is linear and a single iteration calculation is required. (NOTE: Although only one vorticity iteration is required, the iterative structure was set up to allow for an iterative solutioin of the vorticity equation if non-linear terms are added at a later date.) Once the vorticity calculation has been made, the program returns to the velocity/depth calculations. As is true for the velocity/depth phase, the computation of vorticity is done iteratively. In iterative calculations, the solution is approached in a series of “correcting” steps. If either phase converges to a solution before the maximum allowable number of “correcting” iterations, then this is considered to be a “good pass”. A vorticity calculation is considered to have converged if the corrections being made are less than the value of the variables SSVCRT or USDVCC on the VO card. The maximum number of iterations allowed for the vorticity calculation is set by the variables NVITI or NVITN on the TV card. If the number of passes between the phases reaches NPASS1 or NPASS2, then the calculation has failed to converge. If the number of “good passes” between the two phases exceeds NGOODMAX, the calculation is finished and the code can proceed to the next time step.

The solution scheme when solving the bendway correction is a nested iterative scheme. The flowchart sumarizes the scheme.



How To Apply Bendway Correction

The bendway correction option is deactivated by default. Under normal conditions, the bendway correction will require the $L card, BV card, TS card, TV card, and VO card. The VO card *enables* the bendway correction.

Vorticity Boundary Condition

The vorticity calculations require that a vorticity boundary condition be specified. The BV card with the L option is used to apply the boundary condition for the vorticity calculation. Boundary conditions for vorticity will normally be at the inlets of the flow problem. Provide variable J1 with the continuity line number at the desired boundary, and the variable VOR with the value of vorticity along this line. VOR will usually be set to zero.

It is best to locate the inflow boundaries at the ends of relatively long reaches of straight channel if possible. When the vorticity equation is being solved, this will allow any secondary currents to dissipate such that a zero vorticity inflow boundary condition is accurate. If the study area has no straight reaches, then the upstream boundary should be located several river bends upstream so that any errors in the vorticity boundary condition will be minimized for its effects on the study area.

Iteration Control For Vorticity Calculations

Variables on the TV card control the maximum number of iterations of the vorticity phase, and the maximum number of passes between phases (see "The Bendway Correction Solution Scheme In RMA2" on page 94 for a discussion of the vorticity phases). For steady flow problems, NPASS1 sets the maximum number of passes, and NVITI controls the maximum number of vorticity iterations. For unsteady flow problems, these variables are NPASS2 and NVITN, respectively. For either steady or unsteady flow problems, NGOODMAX sets the minimum number of “good passes” required before the calculation is stopped; i.e., when both phases are converging well for NGOODMAX number of times.

Setting Up The Vorticity Calculation

Parameters on the VO card *enable* the bendway correction, set the convergence criteria for the vorticity calculation, and provide the constants for the vorticity transport equation. If the bendway correction is desired, enter the number 1 for the variable IVOR. A small positive number (.00001 is the default) should be entered for SSVCRT if it is a steady flow problem, or for USDVCC for unsteady flow problems. As the vorticity calculation proceeds, changes are being made to the current estimate of vorticity at each node. The magnitude of these changes is referred to as the “norm”. After every iteration, the norm is compared to SSVCRT or USDVCC. If the norm is less than SSVCRT or USDVCC, then the vorticity calculation is considered to have converged, and the vorticity calculation finishes before the number of iterations has reached NVITI or NVITN. The last three variables on the VO card should be entered as follows: ASEC = 5.0, DSEC = 0.5, and RCMIN = 6 feet (or 2 meters).

Even if the number of good passes exceeds NGOODMAX, and the calculation stops, you are cautioned to examine the convergence of the individual phases before concluding that the whole calculation has converged. In general, each subsequent pass to the vorticity phase should require fewer iterations before the norm reaches SSVCRT or USDVCC. Similar behavior should be seen during the (**u**, **v**, *h*) phase. Also, the norm for the first iteration of a phase should decrease as the calculation proceeds. Indeed, in a satisfactory calculation, the norm of the *first* iteration of each phase should approach the value of SSVCRT or USDVCC before the whole calculation is considered to have converged.

Tip: Although experience dealing with vorticity in RMA2 is limited, the technique of converging the traditional (**u, v,** *h*) phase of the RMA2 simulation first, then applying a REV card to activate the vorticity calculation has proved successful. This brings the solution to the correct boundary conditions, eddy viscosities, etc. before applying vorticity calculations to the model.

Obtaining Vorticity Results

RMA2 permits the results of the vorticity calculations to be saved to a unique file with an RMA4 binary solution format. If an output file of vorticity results is desired, a non-zero value must be supplied for NOPTV on the $L card and for IBVO on the TS card. Enter a positive integer for NOPTV and a positive or negative integer for

IBVO, depending on the desired frequency of results. These results are useful only from a diagnostic viewpoint.

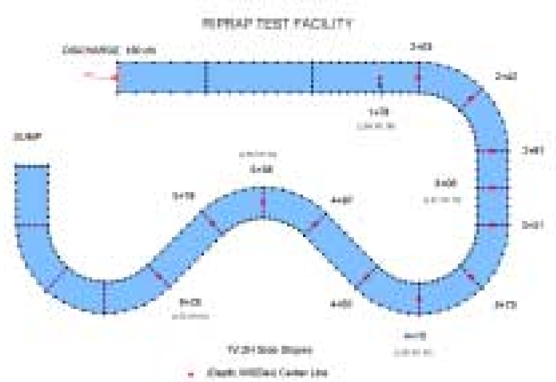
The effects of vorticity on the velocity are incorporated within the traditional (**u, v,** *h*) RMA2 solution file.

When To Apply Bendway Correction

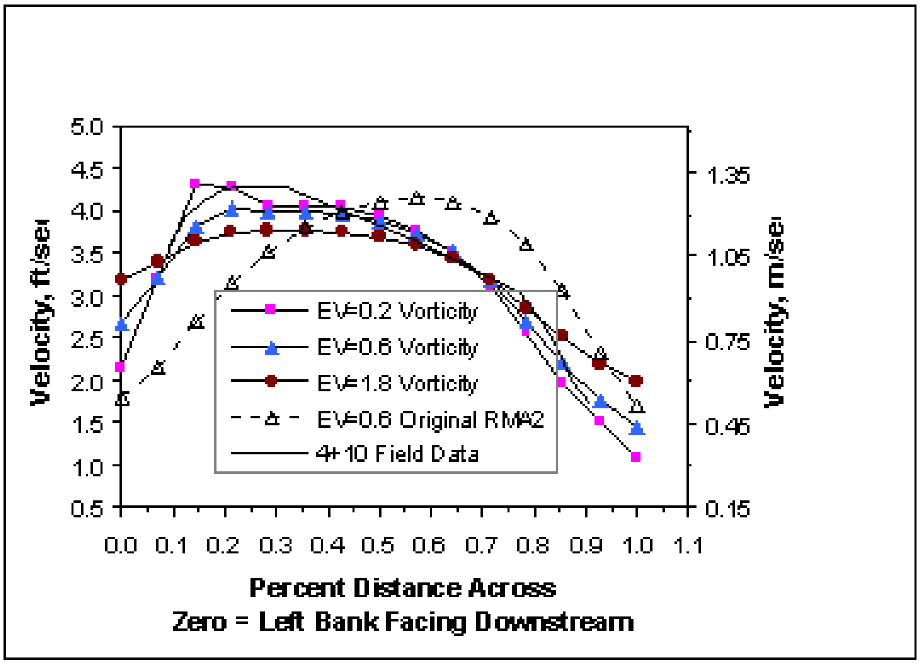
The limited experience using the vorticity correction has shown that the corrections are most significant when a rigid uniform cross section exists through a river bend. However, in a fully "evolved" river cross-section, where the helical flows have created a shallow point bar on the inside bank and a deep channel on the outer bank, the combination of geometry and friction result in a velocity distribution that is in "equilibrium" and the bendway correction make little difference.

Example Application of the Bendway Correction

The ideal field location to test the bendway correction was the WES Riprap research physical model test facility. This facility has 4 bends and 2 reversals in curvature.



The 150 cfs/sec data set from the Riprap research facility was used to compare the RMA2 model performance both with and without the bendway correction. The example graphical comparison is representative of the benefits of the bendway correction. The full derivation, theory and Riprap verification is presented in the research paper cited below.



Example: RMA2 Verification, with and Without Vorticity

Research Paper:

Finnie, John I., Donnell, Barbara P. , Letter, Joseph V., and Bernard, Robert S., "Secondary Flow Correction for Depth-Averaged Flow Calculations", Journal of Engineering Mechanics, Vol 125, No. 7, page 848-863, July 1999.

Fluid Properties

Fluid Density

RMA2 provides the capability to alter the fluid density to meet a certain physical condition. The fluid density may be modified on the FD card. The default density value is that for fresh water: 1.935 slugs/ft^3.

In systems with a dramatic baroclinic (density) effect on mean water surface, it may be prudent to consider both spatial and temporal variation in fluid density. The sacremento San Jaquine River delta is an example where the mean water level in the delta is elevated about 0.3-0.4 ft due to the baroclinic effects of salinity on density. For definition of the tidal prism in a large delta where several tenths of a foot in elevation can result in large area and volume changes, the effects can be significant.

If the salinity field can be defined externall to RMA2 (e.g. RMA4), then the fluid density can be computed and FDN cards defined over the entire mesh. It is possible to include these cards for each time step, but the primary effect can be obtained with a static density field.

Note: RMA2 has been found to be indifferent as to whether you specify the density of fresh water or salt water.

Fluid Temperature

RMA2 provides the capability to alter the fluid temperature to meet a certain physical condition, or perhaps a season of the year. The temperature may be modified on the FT card. The default temperature value is 15 degrees Celsius.

Note: Differences in results between model runs when making minor changes to the temperature value have been found to be insignificant.

Automatic Friction Assignment

For most geometry settings, the automatic friction assignment is a very powerful tool, and permits an realistic means by which to model bottom roughness. By using the RD card, you can specify coefficients to define a functional curve that will be used after each iteration to re-calculate the Manning's roughness coefficient for every element. The depth at each Gauss point of the element is used as the depth for the friction equation.

Tip: Remember that RMA2 does not account for wall friction. You may want to compensate by assigning the elements along the "wall" a unique material type. The RD card with the T option will permit you to exaggerate the roughness in that area.

Automatic Turbulence Closure Assignment

All material types (except junctions and structures) must have a turbulence exchange coefficient assignment. As you recall, this was traditionally accomplished by assigning a value for eddy viscosity with an EV card for each material type. These values were held constant for the entire simulation unless specifically revised by the modeler. The inappropriatly selected eddy viscosity parameters were a potential source of numerical instability problems. This traditional approach is often inadequate, particularly for transient tidal simulations.

Automatic turbulence exchange coefficient assignment is a very powerful tool. This feature permits a real time adjustment to the turbulence parameters (a.k.a. eddy viscosity) based upon the computed velocity field. There are two automatic methodologies: Peclet and Smagorinsky. They may be used separately or in combination.

How Methods of Assigning Turbulence Interact with One Another

It is becoming commonplace for an estuarine application with complex geometry to utilize multiple methodologies for assigning turbulence. Any one method may be used alone or in any combination with one another. It is imperative to understand how each of these methods interrelates with one another.

The rules of turbulence method precedence are as follows:

1. Turbulent Exchange Coefficients, as provided on the EV Card(s)
2. Peclet method for automatic assignments of turbulence, PE Card.
    - All EV Card(s) turbulence parameters are superseded
    - The entire mesh is controlled by the Peclet Method
3. Smagorinsky method for automatic assignments of the turbulence, SM Card.
    - EV Cards for the specified material type are superseded
    - PE Cards for the specified material type are superseded
    - Only the section of the mesh specified by the user is controlled by the Smagorinski Method

The table below illustrates the various combinations of assigning turbulence.

EV	PE	SM	Permissible. Follow Up Comment
X			Yes. All material types <900 must be specified.
	X		Yes. Peclet must be at least global; (PE *blank*)
		X	Yes. Smagorinski must be at least global; (SM *blank*)

X	X		Yes. Peclet rules universally throughout the mesh
X		X	Yes. Smagorinski rules wherever there's an overlap
	X	X	Yes. Smagorinski rules wherever there's an overlap
X	X	X	Yes. Peclet overrules EV. Smagorinski rules over Peclet wherever there's an overlap (ie. a duplicate specification).

NOTE: Overlap means a duplicate specification

Peclet Method

For most geometry settings, the automatic assignment of elemental turbulence coefficients (eddy viscosity) by Peclet number is a very powerful tool. By using the PE card, you can provide *real time* adjustment of eddy viscosity based upon the computed average elemental <u>velocity magnitude</u> and individual size of each element.

The Peclet Method is based upon a Peclet numbers (P), where P is recommended to be between 15 and 40. Generally, larger elements, and elements with higher velocities will have larger eddy viscosity values. Smaller Peclet numbers will result in larger values of eddy viscosity.

Equation 14

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

where:

ρ =fluid density

u=average elemental velocity

dx=length of element in streamwise direction

E=eddy viscosity

Embedded within the Peclet methodology is the variable, VPEC, which specifies the minimum velocity magnitude used for computation of the Peclet number control. The control logic will substitute the user specified value of VPEC whenever the calculated average elemental velocity is smaller than VPEC. With the assigned Peclet numbers, the equation is solved for the eddy viscosity (E).

Note: The velocity used for the Coldstart variable VPEC must be sufficiently large as to permit a successful first guess with the automatic feature.

Tip: Experience has shown that if instabilities arise immediately during initial iterations using the Peclet method, the model will be more stable if the coldstart condition begins using the traditional eddy viscosity assignment made with the EV card, then revising (REV card) the run control file to allow for automatic assignment of elemental turbulence (PE card).

Example: Revise EV-Card with the Peclet Method

….

```
EV     1    500   500   500   500    0.03
TI     8       0    .01
 ….
BQL    6  50000   -1.2
BHL    7      100
REV    Over-rule the explicit eddy viscosity with automatic Peclet=20
PE     1    20   1.0   1 1 1 1
REV    Lower coefficients more by specifying the automatic Peclet=40
PE     1    40   1.0   1 1 1 1
END    Marks the end of the time step
```

Solve for Hour 0.0

Solve for Hour 0.0 given new data

Solves for Hour 0.0 and writes the solution using Automatic Peclet = 40.

Smagorinski Method

For complex geometry settings such as around groins or structures, the automatic assignment of elemental turbulence coefficients (eddy viscosity) by the Smagorinski methodology is a very powerful tool. By using the SM card, you can provide *real time* adjustment of eddy viscosity based upon the computed velocity field. The Smagorinski method is assignment is based on material type. The strength of this automatic feature is that it takes into consideration the gradients of velocity to determine the appropriate turbulence coefficient to meet the conditions in the hydrodynamic simulation. Outstanding results have been achieved with the Smagorinski method.

Generally, larger elements, and elements with higher velocities will have larger eddy viscosity values. The Smagorinski coefficient, TBFACT specified on the SM card, has a recommended value range between 0.094 and 0.2. A larger Smagorinski coefficient will result in larger values of eddy viscosity.

Equation 15

$$E = TBFACT * A\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + 2\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)\right]$$

where:

A=the area of the element

$\partial u / \partial x$ and $\partial v / \partial y$ =velocity component gradients (evaluated over the element)

E=eddy viscosity

Paper: The pioneering paper by J. Smagorinski in 1963 provided the foundation for this method. There are other detailed papers on this methodology: (1) C. Speziale's article in the AIAA Journal (Vol 36, No. 2, Feb 1998); and (2) T.G. Thomas and J.J.R. Williams' article in the Journal of Hydraulic Research (Vol 33, No. 6, 1995)

Wetting and Drying

In the real world, there may be times when portions of an area may become devoid of water, and later become wet again. This process of flooding and draining is referred to as “wetting and drying”. RMA2 provides for elements along the boundary to wet and dry by means of the off-channel storage concept. RMA2 provides two global means by which to simulate these events; elemental elimination (DE card) and marsh porosity (DM card). Either of these methods may be used alone or in combination.

Paper: For details on the wetting and drying methods in RMA2 version 4.35, there is a reference by one of the authors. Roig, Lisa C., (1995) “Mathematical Theory and Numerical Methods for the Modeling of Wetland Hydraulics,” in Water Resources Engineering, Proceedings of the 1995 First International Conference, San Antonio, Texas, August 14 – 18, 1995, American Society of Civil Engineers, New York, pp. 249 – 253.

Off-Channel Storage

RMA2 permits an off-channel storage assignment to any element on a boundary. The off-channel assignment should be thought of as the average combined left and right over bank volumetric contributions. The volume of the off-channel storage interacts with the continuity equation, but makes no contribution to the momentum equation.

The input parameters for off-channel storage are assigned with the GWN cards. These parameters include: (1) WIDS, the off-channel storage width, (2) WSCRIT, the critical water surface elevation to activate off-channel storage and (3) SSS, an off-channel storage slope.

Off-channel storage is activated when WIDS is specified as non-zero.

Prior Strategy for Off-Channel Storage

Previous versions of RMA2 (prior to version 4.52) had an over simplified off-channel storage strategy. Furthermore, these versions permitted only one-dimensional elements to have an off-channel storage. In prior versions of RMA2, the off-channel storage width is always active regardless of the water surface elevation (WIDS), the incremental volume exchange with the off-channel area “per unit length” of the element is WS dz/dt. Also note that the effect that the off-channel storage has on the RMA4 calculation is dependent on the local ambient channel concentration. In other words there is no “memory” of the concentration of the water when it moved into the off-channel storage zone. These transport issues instigated the off-channel storage enhanced features of a critical water surface elevation and storage slope (RMA2 version 4.52 or higher).

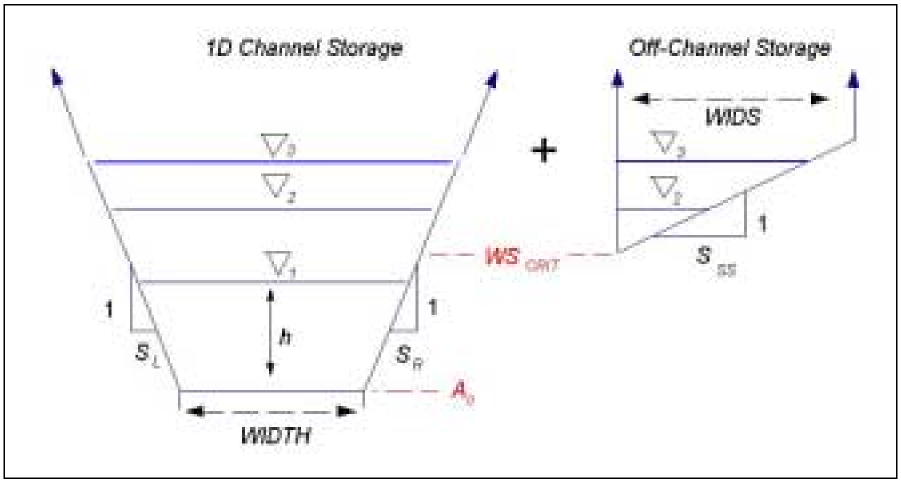
1D Off –Channel Storage

The basic one-dimensional element is composed of two corner nodes and one midside node, and may be either straight or curved. By definition, all basic one-dimensional elements are on the boundary and may be assigned the off-channel storage feature. For simplification of certain numerical issues, it is recommended that the 1D nodes at junctions, transitions and control structure have no an off-channel storage. A representative cross section of 1D channel storage and off-channel storage is shown below.

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 elevation, a bottom width (when the depth=0), a left and right bank slope (S_L and S_R). The designation of left and right bank slope is arbitrary. All slopes reference a given distance for one unit of rise. If the values of S_L, S_R, and the off-channel storage width are zero, the total trapezoidal shape reduces to a rectangle.

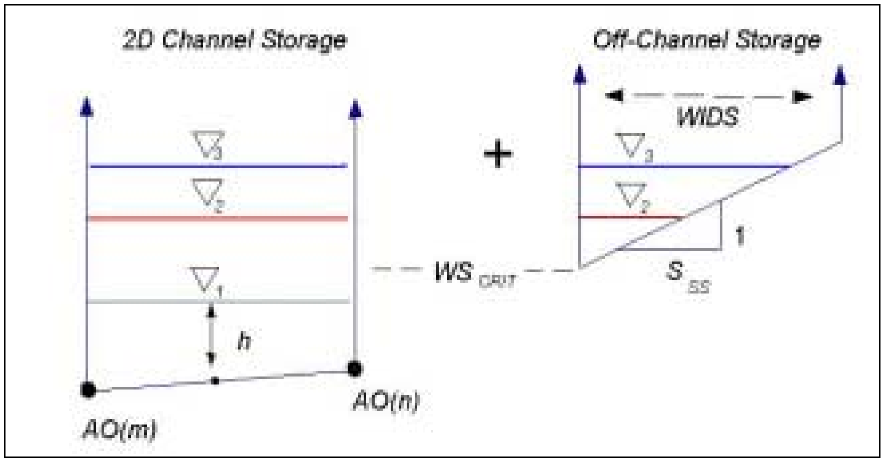
If needed, additional off-channel storage contributions are added by assigning an off-channel storage width (WIDS), critical water surface elevation to activate off-channel storage (WS_{CRIT}), and an off-channel storage slope (S_{SS}). If the values of S_{SS} and WS_{CRIT} are not specified, the enhanced off-channel storage features degrade to that of the previous versions of RMA2 (prior to version 4.52). The earlier versions of off-channel storage functioned as if S_{SS}>>WIDS and WS_{CRIT}<A_o , where A_o is the bottom elevation specified on the GNN card.

.



2D Off –Channel Storage

Any two-dimensional element on the boundary is a viable candidate for an off-channel storage assignment. An element is defined as a boundary element if it contains at least one midside node that is not shared by any other element in the mesh. The traditional 2D channel storage is formed by assigning bottom elevations to the corner nodes (GNN card, AO). The additional storage is assigned by specifying the corner node for off-channel storage (GWN card, WIDS, WS_{CRIT}, S_{SS}).



Elemental Elimination

When used alone, the wetting and drying by element technique eliminates or dries the entire element when any *one* node on the element has a depth less than or equal to the value of DSET on the DE card. Once an element "dries", flow must go around the newly formed land boundary until the projected water depth calculated by RMA2 exceeds the value of DSETD (DE card) for *all* nodes of the element. To prevent an element from oscillating between wet and dry states, it is recommended to make DSETD > DSET.

The mesh should be constructed such that the edges of the elements fall on smooth contour lines such that the storage area wetted or dried at any given moment is minimized. Mesh detailing should be concentrated in the transition regions between wet and dry states. Numerical instabilities can be introduced if an irregular "saw tooth" shoreline results from the newly formed dry land boundary, or if a large storage area is abruptly gained or loss.

Ponds

If the prototype bathymetry of the system dictates the appearance of pond areas, you may have to alter the bathymetry to achieve a monotonically sloping bed to effectively remove the ponds from the simulation. If an isolated pond forms, RMA2 is likely to diverge when the pond is reattached to the wetted network.

Frequency Of Wet/Dry Checking For Elemental Elimination

If a DE card is present, a wet/dry check is always made at the end of a time step. The value of LI on the DE card controls the frequency of checking for wetting and drying by element within a time step. If LI is set too small, the model will not be allowed to converge to a solution based upon the newly formed land boundary before it attempts to proceed to the next wet/dry check. It is safer to set LI to a multiple of the iteration counter (TI card) or greater and let the criterion for satisfactory depth convergence determine when the solution is converged. Hence the values of LI on the DE card must work in harmony with the iteration counter NITI, NITN, SSDCRT and USDCRIT on the TI card.

Advantages Of The Elemental Elimination Method

When the mesh is properly constructed for this wet/dry technique, the RMA2 model will run faster because elemental elimination reduces the number of equations to solve, and the simulation will tend to have a visually realistic wetting and drying appearance.

Disadvantages Of The Elemental Elimination Method

The values of LI, DSET, and DSETD on the DE card are constant for the entire mesh. There is a greater potential of numerical divergence because of the following conditions:

- Irregular boundary shorelines with sharp corners are created if element edges do not align with smooth bathymetric contours.
- Large areas becoming wet or dry during a time step can cause abrupt changes which induce numerical shocks in the simulation.
- Fragmented mesh domains can be created (i.e., potential isolation of mesh regions also known as "puddles" or "ponds").

Typical Problems Encountered While Using Elemental Elimination

When using the DE card, non converging water depth or velocity parameters usually indicate that the simulation is unstable. When the maximum change convergence parameter is greater than 10.0 at the end of any time step, the mesh is probably diverging. Typical causes of this non-convergence include the following:

Problem 1: Oscillations in water surface elevation or velocity

As the water surface drops and an element or series of elements is marked dry, the mass contained in these elements is shifted to the adjacent elements. If the volume of water in these drying elements is relatively large, it can cause the water surface elevation of the adjacent elements to increase enough to re-wet the just dried elements causing oscillations in both water surface elevation and velocity.

Solution

Make elements smaller in areas that are wetting and drying. Also, make sure that the elevation difference on any element that is wetting and drying is not too great. This will minimize the volume of water that is being transferred to a wet element. Another solution is to use the marsh porosity technique (DM card).

Problem 2: Non-smooth interior boundaries

Most users spend a great deal of time making sure the exterior boundary is smooth. However, when a row of elements on the exterior of the mesh becomes dry, the mesh boundary becomes the edge interface between the outside row of elements and the next row toward the interior. If this new boundary is not smooth then all of the problems inherent to a rough boundary will be experienced.

Solution

Move or create new nodes so that the interior element edges are parallel to the contours.

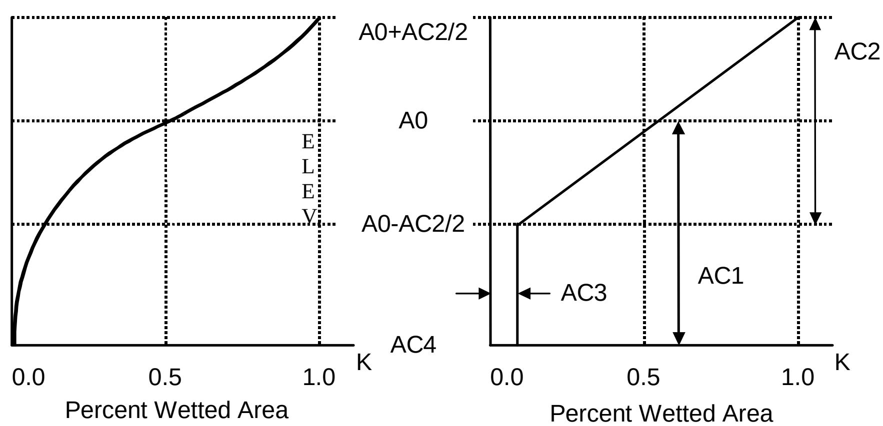
Marsh Porosity

The "marsh porosity" tool in RMA2 allows elements to transition gradually between wet and dry states. This technique allows RMA2 to lower the ability of the element to hold water; like squeezing a sponge. It improves calculation of the shoreline boundary position for intermittently flooded wetlands and tidal flats.

The residual water volume existing on a partially wet element is calculated by vertically integrating a wetted area curve associated with each node of the element. The wetted area curve represents sub-grid scale bathymetry variations that occur in the area surrounding the node. Partially wet elements are retained until all associated nodes become dry. Dry elements re-enter the computations as soon as one associated node is re-wet.

Marsh porosity is either *on* or *off* for all elements in the mesh during the simulation. However, if marsh porosity is on, and all nodal depths for a particular element exceed the transition range of the distribution (i.e., completely wet element), then the element will carry 100% of its capacity, and marsh porosity has no effect within that element.

The wetted area curve defines the surface area of standing water as a function of water surface elevation. This curve can be determined empirically if detailed contour maps or aerial photos are available. An example of an empirical wetted area curve is shown on the left in the figure below. Note that the wetted surface area, K, has been normalized to the total nodal area such that $0 \le K \le 1$. The data required to construct a wetted area curve is frequently unavailable, so that the user must supply an approximate function to describe the wetted area curve for each node. One such approximate function is illustrated on the right. This schematized wetted area curve has the benefit of being easy to integrate, and can be completely described with four parameters: AC1, AC2, AC3, and AC4.



Nodal Wetted Area Curve Obtained From Field Data

Schematized Nodal Wetted Area Curve Obtained By Approximation

The effective wetted surface area will continue to decrease as the depth of additional nodes on the drying element fall below the transition range. If you suspect that all nodes on an element will have the potential to be flagged dry, then the DE card is required to accompany the DM card to allow for proper removal, and later reinsertion, of the element. If *all* nodes on the element have a calculated depth less than DSET on the DE card, then the element is considered to be completely dry and is eliminated from the simulation. Although the element may be eliminated and a newly formed land edge is created, the Marsh Porosity option has specified that the storage of the element has been gradually reduced, and the likelihood of numerical shocks is drastically reduced. Dry elements re-enter the computations as soon as *one* of the nodes on that element has a depth greater than DSETD on the DE card.

Note: The Marsh Porosity rules for completely eliminating dry elements from the network and reinserting rewet elements are different than with the Elemental Elimination method.

The data required to determine the parameters for marsh porosity are typically unavailable, so you must approximate. Care must be taken to have similar marsh porosity parameters between adjacent nodes.

Here are some additional guidelines for selecting the values of AC1, AC2, and AC3 on the DM card.

- A value of AC1 too small will cause the node to dry prematurely and flow volume will be underestimated.
- A small AC2 makes the distribution behave like a step function which will result in abrupt wet/dry changes similar to a narrow feeder channel opening abruptly onto a wide flat marsh plain.
- AC3 must be small (on the order of 0.01) but always greater than zero.

Many of the same mesh construction guidelines for elemental wetting and drying should be followed when using marsh porosity.

Frequency Of Wet/Dry Checking For Marsh Porosity

When the Marsh Porosity is on, RMA2 automatically calculates the effective nodal depth at every iteration, but if a DE card is present at the same time, the LI parameter controls the frequency of checking for when elements need to be removed from or reinserted into the mesh.

Advantages Of The Marsh Porosity Method

- The mesh design can be more lenient.
- You have total control of parameter assignments.
- The solution will be more stable, and tidal storage is appropriately handled.
- The TABS sediment model, SED2D version 2.0 or higher, has marsh porosity compatibility.

Disadvantages Of The Marsh Porosity Method

At the time of this writing, SMS will display partially dry marsh elements as all wet even though the water surface elevation may be below the bed elevation. Therefore, the FastTABS or SMS user is encouraged to examine *depth contours* rather than *water surface elevation* values.

Typical Problems Encountered While Using Marsh Porosity

Problem 1:

Instability due to different values for marsh porosity parameters being located side by side in the mesh.

Solution

Choose the marsh porosity parameters so that there is not a sudden change between adjacent areas. Select values so that there is a somewhat gradual change between areas.

Problem 2:

Instability due to the selection of AC1 on the DM card.

Solution

The first guess for the appropriate value of AC1 should be the difference in cross sectional elevation along the edge of the element. If instability persists, then try gradually increasing the value of AC1.

Element Inflow And Outflow

RMA2 provides a means to add fluid to or remove fluid from an element or multiple elements. This ability can be utilized to simulate events such as pumping and discharging, or rainfall and evaporation. The way of thinking about these two situations may differ slightly, but the process is the same.

Pumping And Discharging via Element Flow

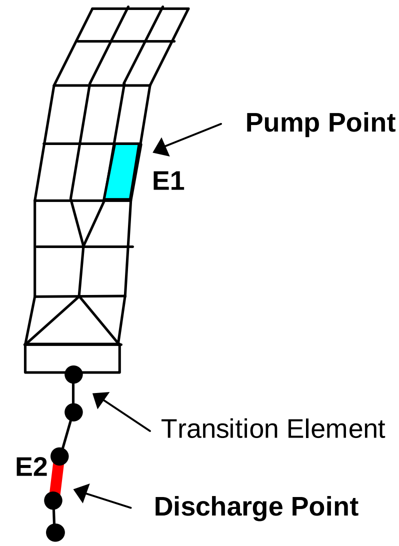
Simulating a pumping and/or discharge event can be accomplished by specifying an outflow (pump) and/or inflow (discharge) boundary condition at an element. This technique is used by adding fluid to or subtracting fluid from the specified element. For a two-dimensional element, the outflow or inflow is measured per unit area (ft^2 or m^2) of the element, whereas for a one-dimensional element, the outflow or inflow is measured per unit length (ft or m) of the element.

RMA2 expects a value for *inflow* only. To provide an outflow, you must use a negative value for inflow.

Note: A negative inflow is an outflow.

Example: Pump and Discharge using Element Side Inflow

The figure below illustrates pumping for a two-dimensional network and discharging for a one-dimensional network . For example, to pump 5000 cubic feet per second (cfs) from a single 2D element, **E1**, with an area of 1000 ft^2, the value for the elemental inflow per unit area will be *negative* 5.0 ft/sec. To discharge 5000 cubic feet per second (cfs) from a single 1D element, **E2**, with a length of 50 ft, the value for the elemental inflow per unit length will be 100 ft^2/sec.



CO   Pumping (outflow)
BQE   *E1*   *(-Discharge/Area$_{E1}$)*
CO   Discharge (inflow)
BQE   *E2*   *(+Discharge/Area$_{E2}$)*

For pumping and discharge applications using control structures, see "Pumping And Discharging Using 1D Control Structures" on page 76.

Adding Rainfall And Evaporation

"Rainfall and evaporation" is a special type of boundary condition. Rainfall and evaporation can be applied to a single element, or to many elements in the network. Rainfall or evaporation is applied in *inches/hour* if using English units, or *cm/hour* if using Metric units. These values are internally converted to feet/second or meters/second depending on the designation on the SI card.

The elements you want to influence with rainfall or evaporation can be specified with the RA card using the *by element* option, E, or the *by material type* option, T. All elements in the mesh can be included by using the RA card with no option.

Note: You cannot apply rainfall or evaporation to junction elements or control structure elements, or elements which have a material type (IMAT) greater than 900. When globally assigning rainfall and evaporation, these elements are automatically skipped by the program.

Rainfall and evaporation can occur at any time during the simulation. For example, if you want rainfall to start at hour 30 and stop at hour 33, and evaporation to begin at hour 50 and stop at hour 60, start the simulation with SIDF on the RA card set to zero. At hour 30, update the parameter on the RA card, making it a positive value. At hour 33, update again returning the rainfall value to zero. At hour 50, update the RA card parameter yet again making it a negative value, denoting evaporation, and at hour 60 update the RA card again to set the value back to zero.

Compensating For The Rotation Of The Earth

The Coriolis force is the inertial force caused by the Earth's rotation. RMA2 provides two ways to control Coriolis calculations:

- Average Latitude Of The Mesh
- Specifying Latitude By Material Type

Warning: Instabilities on the southwest quadrant of the solution have been observed when using Coriolis Force correction in the Northern hemisphere. If this is a concern, a workaround is to apply graduated values of latitude (Coriolis). This is done by setting the latitude value for elements near the area of instability to zero, and increment the value of latitude in rows or layers of elements toward the point in the mesh where you want the value of latitude to be the true latitude. An increment of about 25% of the true latitude should be sufficient.

Average Latitude Of The Mesh

You can provide the *average* local latitude of the entire mesh in one step on the LA card. This is useful when you want to include one Coriolis force to be applied throughout the entire mesh domain, as opposed to specifying the latitude by material type.

Specifying Latitude By Material Type

If the study domain is very large, you may want to provide several Latitude assignments which will encompass all of the modeled area. The LA card with the T option permits you to specify the latitude according to material type. This method will provide more customized Coriolis force calculations than when specifying globally the average latitude of the domain. Alternatively, this material type option permits you to gradually decrease the Coriolis forces if instabilities on the southwest quadrant are observed.

Applying Wave Radiation Stress

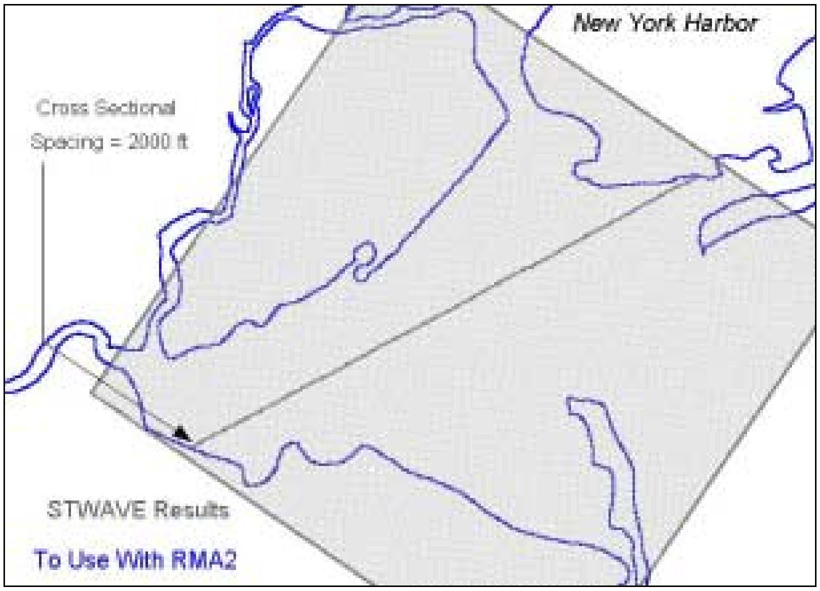
Applying wave radiation stress to the RMA2 simulation is optional. This boundary condition is useful for specialized cases. The first numerical model application that incorporated the radiation wave stress capability was New York Harbor. Wave induced currents were required in the hydrodynamic simulation to reproduce literal sediment movement in the sediment transport model, SED2D. Because of the lack of experience using wave input in RMA2, this feature remains experimental and subject to format modification.

The radiation stresses from waves may be applied to an RMA2 simulation by reading in a wave file containing wave height and period results ($L card, variable INWAVE) typically generated from an STWAVE model of the same geographical region. STWAVE is a numerical model developed for the specific purpose of generating the wave direction, height, and period. If wave file is available ($L card, INWAVE); the stress is incorporated into the RMA2 calculations whenever the wave/current interaction switch is non-zero (HW card, variable IWCUR) and the wave file is available.

By activating the option (IWAV variable on the HW card), the RMA2 modeler may apply scales and other control parameters to adjust input wave values.

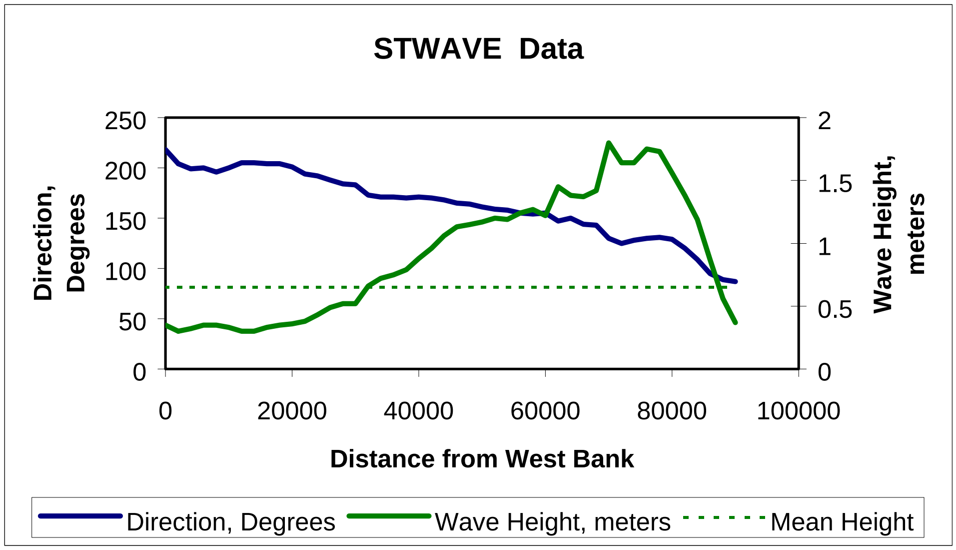
Wave Application Example:

For example, the figure of Raritan Bay and Lower Bay of New York Harbor shows the RMA2 model domain in blue, and the STWAVE wave model boundary in gray. The STWAVE grid and results may be read into SMS, converted to scatter points, and interpolated to the RMA2 geometry. The discrepancy between the model domains required some additional effort and engineering judgment in the areas lacking information.



Wave height and direction were plotted for the 92,400 foot cross section shown, spanning from the western bank line to the eastern bank line, at 2000 ft increments. The plotted data provides an example of STWAVE output for the cross section. For the entire data set, the direction had an extreme of 231 degrees, mean of 152 degrees, and standard deviation of 24.47. The wave height had an extreme of 2.47 meters, mean of 0.65, and standard deviation of 0.848. For this application, the RMA2 modeler chose to scale the New York wave height data set by a factor of 0.20 and set a maximum wave height limit of the mean plus 2 standard deviations. These parameters were set on the HW card of the RMA2 run control input file, as shown.

Hydrodyanamic Wave Card Example: HW 1 1 0 0.20 2 1 1 2 1



Applying Wind Friction

RMA2 has several optional wind shear stress formulations. This gives you considerable flexibility in handling wind effects which are extremely difficult to implement in a two-dimensional model. Wind is de-emphasized and generally not used in this program because wind-driven currents are three-dimensional in nature. However, in systems that are generally shallow with strong wind influences, wind stress may be required to obtain proper setup and system circulation.

Note: Modeling experiences with RMA2 running wind suggest that the wind speed and direction be gradually "ramped" over time between extreme values.

Applying wind friction is a two step process.

- Specify The Controlling Wind Formulation
- Specify Wind Speed And Direction

Once wind friction has been established, it remains in effect until a change is specified. RMA2 version 4.52 and higher have a logic switch to eliminate wind friction on marsh porosity elements when the depth of water is in the "nearly dry" range of the wetted area curve.

Tip: Remember that wind friction can be included or removed at any timestep. To remove wind friction effects after they have been activated, set IWIND=0 on the BWC card.

Specify The Controlling Wind Formulation

RMA2 currently provides eight formulations from which to choose when applying wind friction.

1. RMA2 Original Formula
2. Van Dorn Formula

3. Wu Formula
4. Safaie Formula
5. Ekman Formula
6. Generic Formula
7. Van Dorn Formula with defaults
8. Wu Formula with defaults

The wind stress formula is selected by setting the IWIND parameter on the BWC card. The majority of numerical modelers at WES have opted for the Wu formulation. See the BWC card for further information.

Specify Wind Speed And Direction

The wind speed and direction may be applied globally throughout the mesh domain, or by material type, element, or individual node on the BW card with the appropriate option.

Simulating With Storms

RMA2 provides the capability to include storm fronts in your simulation. This is accomplished by generating spatial variations in wind speed and direction for each time step based upon a user specified set of criterion for the storm(s). These criteria include factors such as:

- storm path
- orientation of the storm along the path
- storm speed
- shape
- temporal growth and decay constant for wind

Note: Storm rotation is based on the Coriolis forces for the northern hemisphere.

To date, the two-dimensional Mississippi River Gulf Outlet study (1991) is the only application at WES in which storms were used to provide the wind stress. Because of lack of experience in this area, this feature remains experimental.

Storms In A Simulation

Many open water systems are strongly influenced by meteorological forcings which can apply pressures at the water surface. These wind stresses can be from a variety of storm types. The wind effects can generally be classified either as steady in nature or associated with a dynamic event. The ability of RMA2 to adequately simulate a real-world event must be carefully evaluated for each of these classes.

The response of a water body to wind forcing will generally be three-dimensional. The surface wind stress will drive a surface current in the direction of the wind. The influence of that surface wind drift will then be dependent on the water depth and the local shoreline geometry. If the currents are directed to a shoreline, the movement of surface water will result in a pilling up of water on that shoreline. As the water level is raised on the shore, a hydrostatic pressure gradient is developed perpendicular to the shore, which will in turn drive a return flow away from the shore in the bottom portion of the water column. This vertical circulation pattern will moderate the increase in the water level at the shoreline. The strength of that return flow will be dependent on the water depth; generally stronger in shallower water.

The process is however a dynamic one. During the initial movement of water toward the shoreline, there is insignificant return flow at the bottom. Only after the water level is dramatically affected does the three-dimensional circulation develop. Wind events associated with stationary meteorological conditions may generate persistent unidirectional winds for prolonged periods of time, which may fully develop vertical circulation cells in the water column. However, there are a large number of wind events that are of a shorter duration and do not involve a fully developed vertical circulation response.

RMA2 Considerations

The three-dimensional nature of the circulation affects how appropriate the application of a depth-averaged model will be to the problem. It has been shown that for a steady-state type of wind forcing, RMA2 will overestimate the setup of the water levels at the shoreline. However, for dynamic events, the model will generally perform well for the initial movement of water. If the dynamics of the storm lead to a change in the wind stress such that a steady-state condition is never approached, then RMA2 will yield reasonable results.

If the shape of the shoreline is more complex, horizontal variations of the water surface may result in localized return flows that do not involve significant vertical circulation. Such circulation patterns are directly analogous to rip tides along a beach in response to the radiation stress of the waves.

The ability of RMA2 to properly simulate wind forcing hydrodynamic response is limited to cases where steady-state conditions are *not* approached. RMA2 will result in a raised water surface, or “setup”, at the shoreline, but because of the depth averaging, will have no return flow. Technically, RMA2 gives the appropriate “average velocity” for such conditions. However, if water quality or any dispersive processes are of interest, the results must be interpreted with care. In addition, the water surface setup will normally be overestimated.

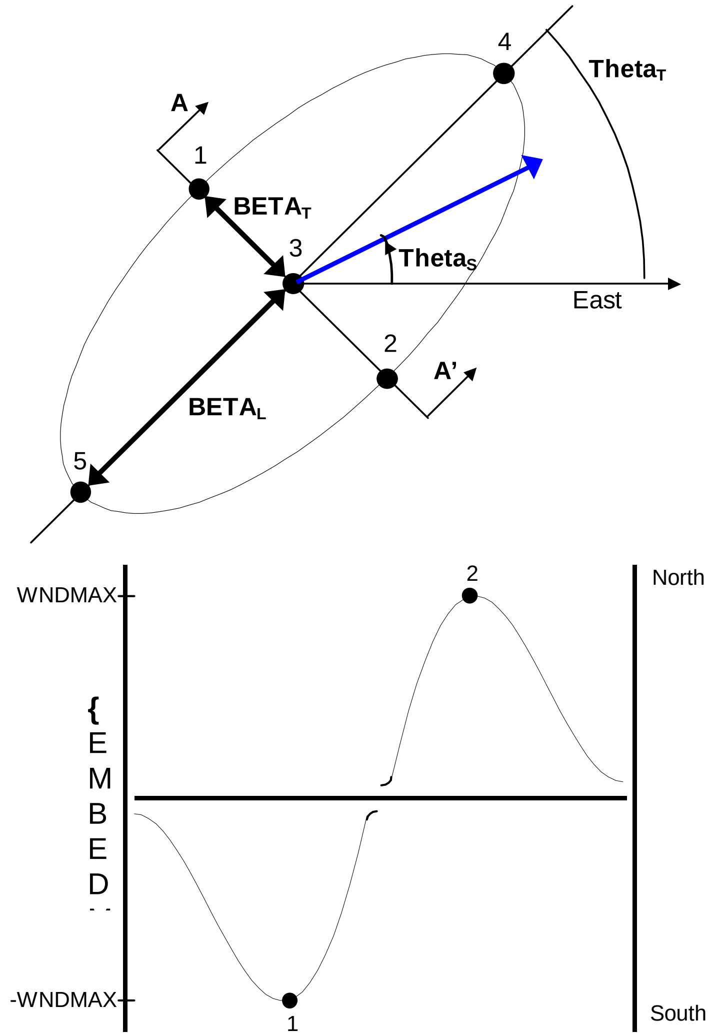
RMA2 can adequately handle certain types of dynamic events. The response of a system to a frontal passage or tropical storm can be evaluated with some confidence level by RMA2, provided the storm does not stall, which will lead to some steady-state issues.

Defining Storm Events

The representation of storms within RMA2 is provided as a means of easily developing relatively complex wind fields. A *“storm”* in RMA2 is defined as a specific tracking of the location of the storm center as a function of time, and the characteristics of that storm.

A storm is defined using the BWS card. Each BWS card creates a single storm. You may simulate multiple storms (presently dimensioned to two) to describe complex wind patterns by including additional BWS cards.

The storm characteristics are as described below. Refer to this illustration.



Storm Cross Section A to A'

The Storm Reference Point

There are several options regarding the placement of the storm reference point relative to the numerical mesh. Referring to the figure, the reference point, ISTYPE, can be defined as either:

1. The location of maximum winds along the positive minor axis
2. The location of maximum winds along the negative minor axis
3. The center of the storm
4. The location of maximum winds along the positive major axis
5. The location of maximum winds along the negative major axis

These options are designed for convenience in obtaining the desired wind direction at a particular place and time when used in conjunction with other variables.

Time And Place Of The Occurrence Of Maximum Winds

The initial position of the storm is back tracked based on the storm tracking speed and direction of storm movement. The RMA2 simulation time, TREF, in decimal hours, is when the storm will arrive at the designated reference node, NREF, where NREF defines the location of the storm reference point. This places the peak winds relative to a specified time of the model simulation.

The pairing of location and time, combined with the storm path direction are used to develop the full storm track during the simulation. Based on the reference storm location and time specified, RMA2 will perform all calculations to determine past and future storm movements.

Storm Tracking Speed

The speed, SSPD, in miles per hour (mph) of the storm across the study domain.

Storm Major Axis Dimension

The longitudinal distance, $BETA_L$, from the storm center to the point of maximum wind speed, in feet or meters depending upon the setting of the SI card. The major axis is generally defined as the long axis.

By defining the major spatial axis as *large* relative to the minor axis dimension, a cold front storm can be simulated. This will normally require the major axis to be *very* large relative to even the model mesh dimensions. A Hurricane type storm can be simulated by choosing both the major and minor axis as having the same value, creating a circular storm pattern.

Storm Minor Axis Dimension

The transverse distance, $BETA_T$, perpendicular to the major axis from the storm center to the point of maximum wind speed, in feet or meters depending upon the setting of the SI card.

Maximum Wind Speed

The maximum wind speed, WNDMAX, in miles per hour (mph) will define the maximum winds along the ellipse defined by the major and minor axes.

Minimum Wind Speed

The minimum wind speed, WNDMIN, in miles per hour (mph) will define the minimum winds within the storm.

Growth And Decay Constant Relative To Maximum Winds

The exponential temporal decay coefficient, DECAY (hr), is a Gaussian variance of wind speed relative to the time of maximum winds. This parameter facilitates the specification of daily wind patterns and tropical storm growth. This constant is normally derived from field data. A typical value for daily winds would be 6 hours.

Orientation Of The Major Axis Of The Storm

The direction $THETA_T$ of storm orientation in degrees counter-clockwise relative to the positive *x* axis. This is significant for frontal passages to get the spatial variation of winds properly prescribed over the model mesh.

Direction Of Storm Movement

The direction toward which the storm is moving, $THETA_S$, is the storm track in degrees counter-clockwise from the positive *x* axis.

Note: This direction does not have to match the orientation of major axis.

Adjusting RMA2 Calculations

RMA2 allows you to modify the temporal derivative calculation. Typically, you will never change this. However, if you feel you must, the CA card is what you use to change the parameter.

The variable USERCA1 on the CA card is used to calculate the temporal derivative. Generally, values of USERCA1 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. A value of 1.5 yields a second order Taylor series expansion. With small time steps, the value of USERCA1 has little or no effect. With larger time steps, try values closer to 1.0 if model stability is a problem.

It has been mathematically derived that the default value of 1.6 will tend toward an unstable solution. However, based upon discussions with the primary author (Dr. Ian King), a large collection of experience has shown that for a Crank Nickolson parameter between 0.62 and 0.69, the value of 1.6 performs well and should remain as the default.

Run Control

Format for Run Control Data

RMA2 obtains its run control data from a set of input data cards provided in run control files. These 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 functional data.

A data card occupies a single line in the run control 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. RMA2 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 RMA2, 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 RMA2 run control file.

Card	Content	Revisable?	Required?
T1-T2	Title cards	No	No
T3	Title card	No	**Yes**
CO	Comments (anywhere except first line)	---	No
$M	Machine type identifier	No	**Yes**
$L	Input/output file numbers	No	No
SI	System international units	No	No
G1	Geometry, general geometry parameters	No	Replaced
GC	Geometry, continuity check lines	No	No
GCL	Geometry, continuity check lines	No	No
GE	Geometry, element connection table	No	No

GN	Geometry, nodal coordinates, bottom elevations	No	No
GS	Geometry, scale factors	No	No
GT	Geometry, element type (IMAT)	Yes	No
GV	Geometry, eddy viscosity tensor	Yes	No
GW	Geometry, one-dimensional cross sectional properties	No	No
GZ	Geometry, bottom elevations	No	No
CA	Special calculation variables	Yes	No
DE	Wet/dry by element	Yes	No
DM	Wet/dry by marsh porosity	No	No
EV	Turbulent exchange and Manning's n coefficients	Yes	**Yes**, If EX and EY and HN not present
EX	Turbulent exchange coefficients, e, x-plane	Yes	No
EY	Turbulent exchange coefficients, e, y-plane	Yes	No
FC	Flow control structures	Yes	No
FD	Fluid density	Yes	No
FT	Fluid temperature	Yes	No
HN	Hydraulics, Manning n-values	Yes	No
HW	Hydraulics, Radiation Wave Stress	Yes	No
IC	Initial conditions, water-surface elevation	No	**Yes**
LA	Latitude	Yes	No
PE	Peclet Method for Automatic Turbulence	Yes	No
RA	Rainfall and evaporation	Yes	No
RD	Roughness assignment by depth	Yes	No
SM	Smagorinski Method for Automatic Turbulence	Yes	No
TI	Timing, number of iterations counter	Yes	**Yes**
TO	Timing for binary output write	No	No
TR	Trace printout controls	Yes	**Yes**
TRN	Trace printout node list	No	No
TS	Timing for writing to (u,v,h) solution	Yes	No
TV	Timing of vorticity iteration	Yes	No
TZ	Timing, of simulation	Yes	**Yes**
BCC	Boundary condition control parameters	Yes	No
BA	Boundary, azimuth of flow	Yes	*****
BS	Boundary, current speed	Yes	*****
BH	Boundary, water-surface elevation	Yes	*****
BQ	Boundary, unit discharge	Yes	*****
BRA	Boundary, reflection/absorption	Yes	*****
BRC	Boundary, rating curve	Yes	*****
BWC	Boundary, wind formulation control	Yes	No
BW	Boundary, surface IWND data field	Yes	No
BWS	Boundary, wind storm	Yes	No
BCN	Boundary condition specified by node	Yes	*
REV	Revise the coefficients or boundary conditions in	---	No

	mid-time step		
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.*

Input Variables

The following is a table of input variables which are the parameters found on the RMA2 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
AC1	Distance from the average regional bed elevation to the minimum regional bed elevation	DM
AC2	Transition range of the distribution	DM
AC3	Minimum wetted surface area factor	DM
AC4	Minimum regional bed elevation	DM
AC1X	Constant for rating curve	BRA, BRC
AC2X	Multiplier for rating curve	BRC
AC3X	Base elevation for rating curve	BRC
AC4X	Exponent for rating curve	BRC
ALFAK	Angle at a node, in radians	BA
AO	The bottom elevation of each node	GN, GZ
ASEC	Coefficient for the vorticity equation	VO
BETA	Reflection/absorption boundary coefficient	BRA
BETAL	Standard deviation of spatial distribution function in direction of storm's movement	BWS
BETAT	Standard deviation of spatial distribution function in direction of transverse to storm path	BWS
CORD	The (x,y) coordinates of the node	GN
DECAY	Exponential temporal decay for wind speed (hr^{-1})	BWS
DELT	Length of computation time step	TZ
DSEC	Coefficient for the vorticity equation	VO
DSET	The water depth at which a wet node is considered to become dry	DE
DSETD	The water depth at which a dry node becomes reweted	DE
ELEV	The average initial water-surface elevation over the mesh	IC
EPSXX	Scaling factor for computed eddy viscosity in xx direction	PE
EPSXY	Scaling factor for computed eddy viscosity in xy direction	PE
EPSYX	Scaling factor for computed eddy viscosity in yx direction	PE
EPSYY	Scaling factor for computed eddy viscosity in yy direction	PE
FLD	Any alphanumeric user comment	CO, END, REV, STO
FLZ3	Flow control equation, base flow	FC
FLZ4	Flow control equation, relational coefficient	FC
FLZ5	Flow control equation, reference elevation or head difference	FC
FLZ6	Flow control equation, exponent	FC
FLZ7	Flow control equation, direction of flow (radians counterclockwise from + x-axis)	FC

GPEC	Peclet number	PE
HFX	Head specification along a continuity line	BH
HMIN	Initial depth for one-dimensional elements	IC
HSFACT	Factor applied to wave height	HW
HSLIM	Maximum allowable wave height	HW
HSVAR	Scaling factor for the variance about the mean wave height.	HW
IBHO	Scheme for saving RMA2 binary (u, v, h) solution	TS
IBUP	Logical unit number for dynamic boundary conditions input to RMA2	$L
IBVO	Scheme for saving RMA2 binary vorticity solution	TS
IC1	Card group identifier, all cards	ALL
IC3	Data type identifier, some cards	SOME
ICON	Continuity line number	BRA
IVRSID	Computer identifier	$M
IDEN	Counter for the number of node that have a non-default fluid density assignment (self-count)	FD
IDNOPT	Marsh porosity option on/off switch	DM
IECHO	Switch to control echo printing of coded input data records	TR
IFILE	Logical unit number for reading GFGEN binary geometry	$L
IFINO	Logical unit number for RMA2 to write results for transfer to RMA4 or SED2D	$L
IGEON	Logical unit number for GFGEN geometric data file	$L
IHGEN	Dynamic counter for input of number of lines across which elevation will be specified (self-count)	BH
IHOTN	Logical unit number for file containing initial conditions	$L
IHOTO	Logical unit number for RMA2 to write-restart file	$L
IMAT	The element type (n-value and eddy coefficients)	GE, GT, LA, PE
INWAVE	Logical unit number for reading wave stress input file	$L
IOUT	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	TR
IQGEN	Dynamic counter for input of number of lines across which total flow will be specified (self-count)	BQ
IRUFF	On/off switch for auto Manning's n-value	RD
ISPRT	Logical unit number for writing summary results listing	$L, TR
ISTART	Starting number for global assignments (local variable)	LA, RD
ISTGEN	Dynamic counter for input of number of lines across which an elevation-flow relationship will be specified	BR
ISTYPE	Storm reference point	BWS
ITRACE	Trace subroutine calls and controllers (debug)	TR
ITSI	Number of time steps between successive full results listings	TR
IVOR	On/off switch for vorticity calculations	VO
IWAV	On/off switch for performing adjustment to the input	HW

	waves file	
IWCUR	On/off switch for wave/current interaction	HW
IWIND	Control for wind field input	BCC, BWC
IWMX	Wind speed unit flag	BWC
IWR33	Logical unit number for writing detailed steady state parameters when using automatic features (viscosity and roughness)	$L
JCH	Continuity line number for head boundary (J = JCH)	BH
JCQ	Continuity line number for total flow (J = JCQ)	BQ
JCR	Continuity line number for rating curve	BRC
LI	The number of iterations between checks for dry nodes	DE
LINE	Corner node numbers for continuity check	~~GC~~, GCL
LMT	Total number of corner nodes on a given continuity line	~~GC~~, GCL
MBAND	Restart in mid-iteration (flag)	TZ
METRIC	System International (flag)	SI
NBX	Number of nodes with boundary conditions specified	BCC, ~~G1~~
NCFLW	Number of flow control structures (self-count)	FC
NCYC	Number of time steps simulated	TZ
NFIX	Array containing logic flags for boundary condition	BCN
NFIXH	Element Reordering list read from GFGEN	GFGEN GO
NGOODMAX	Number of good passes through a series of (u, v, h) and vorticity calculations before the solution has converged	TV
NITI	Number of iterations for initial solution (or steady state computation)	TI
NITN	Number of iterations for each dynamic computation	TI
NJN	Flow control structure identifier (IMAT ≥ 904)	FC
NJT	Flow controller type	FC
NMAT	The number of different sets of turbulent exchange coefficients and Chezy and/or Manning coefficients	EV, EX, EY
NOP	Nodal point-element connection table for RMA2	GE
NOPTV	Logical unit number for writing vorticity solutions in RMA4 format	$L
NPASS1	Maximum number of passes between the steady state (u, v, h) and vorticity iterations	TV
NPASS2	Maximum number of passes between the dynamic (u, v, h) and vorticity iterations	TV
NREF	Nodal point reference for storms	BWS
NSID	Dynamic counter for number of elements for which element inflow is desired	BQ
NSPLPT	Array containing nodes for summary results listing	TRN
NSTART	Starting time-step number used to skip through boundary condition data for restart	TZ
NVITI	Maximum number of steady state vorticity iterations	TV
NVITN	Maximum number of dynamic vorticity iterations	TV
OMEGA	Latitude of mesh (approximate average)	~~G1~~, LAT
ORT	Eddy diffusion and n-value array by element type	EX, EY, HN, EV

PWCORR	Time shift for starting time in wave model input file	HW
QDIR	Flow direction along continuity line	BQ
QF	Total flow along a continuity line	BQ
QXP	Factor for distribution of flow by depth	BQ
QVEC	The unit discharge (cfs/ft)	BQ
RCMIN	Minimum radius of curvature with vorticity	VO
RDR0	Maximum Manning's n for non-vegetated water	RD
RDD0	Depth at which vegetation effects roughness	RD
RDCOEF	Roughness by depth coefficient	RD
RON	The array of nodal fluid density	FD
SIDF	Element inflow (per unit area)	BQ
SPEC	Array containing boundary condition specifications	BCN, BH
SS1	Left side slope for one-dimensional nodes	GN, GW
SS2	Right side slope for one-dimensional nodes	GN, GW
SSDCRT	Steady state satisfactory depth (convergence) criterion	TI
SSPD	Storm speed, mph	BWS
SSVCRT	Steady state vorticity convergence criterion	VO
SSWSE	Steady state water surface elevation	BRA
TAREA	Surface area of tidal storage beyond the boundary	BRA
TAX	Wind direction (degrees counterclockwise from + x-axis)	BW
TBFACT	Array controlling whether a given material type will have Smagorisky coefficient for turbulent exchange.	SM
TBFACTS	Array controlling whether a given material type will have Smagorisky coefficient for diffusion (RMA2 -vorticity).	SM
TBMINF	Array controlling the minimum Smagorisky turbulence exchange factor for a given material type.	SM
TBMINFS	Array controlling the minimum Smagorisky diffusion factor for a given material type. (RMA2 -vorticity).	SM
TBINRY	Array containing user selected hours to save to binary final results file	TO
TEMP	Average initial water temperature	FT
TH	Azimuth of x-direction of an element for specifying eddy diffusion coefficients	GE, GV
THETA	Direction of flow (radians measured counterclockwise)	BRA, BRC
THETAS	Direction of storm track	BWS
THETAT	Orientation of storm	BWS
TITLE	Character identifier for the run and all output files	T1-T3
TMAX	Total time in decimal hours	TZ
TREF	RMA2 simulation time of storm arrival at 'NREF'	BWS
TWX	Wind speed	BW
UNOM	Initial velocity for one-dimensional elements	IC
USDCRT	Dynamic satisfactory depth criterion	TI
USDVCC	Dynamic vorticity convergence criterion	VO
USERCA1	User selected variable for temporal derivative calculation	CA
VOR	Vorticity value	BV

VPEC	Initial guess when using automatic Peclet number	PE
VVEC#	Current speed (fps) at a node	BS
WC1	Wind control coefficient	BWC
WC2	Wind control coefficient	BWC
WC3	Wind control coefficient	BWC
WC4	Wind control coefficient	BWC
WDMC1	See AC1	DM
WDMC2	See AC2	DM
WDMC3	See AC3	DM
WDMC4	See AC4	DM
WIDS	Storage with associated one-dimensional node	GN, GW
WIDTH	Channel width for nodes	GN, GW
WNDMAX	Maximum wind speed (mph)	BWS
WNDMIN	Minimum wind speed (mph)	BWS
WPFACT	Factor applied to the wave period	HW
WPLIM	Maximum allowable wave period (sec)	HW
WPVAR	Scaling factor for the variance about the mean wave period.	HW
XSCALE	Scale factor for x-coordinates	~~G1~~, GS
YSCALE	Scale factor for y-coordinates	~~G1~~, GS
ZSCALE	Scale factor for z-coordinates	GS

RMA2 Execution Job Sheet

JOB EXECUTED __________________ DATE OF RUN ____/____/____ TIME OF RUN ________________

JOB PRINTED _________________ SUBMITTED BY ____________________ CPUs ____ PRIORITY_____

* *

PURPOSE:

____

____

____

____

____

SIMULATION TIME: Start ____________ Finish ____________

FILES:

Primary Run Control File (.rc2)	____________________
Geometry file from GFGEN	____________________
RMA2 Hotstart input	____________________
RMA2 alternate BC file	____________________
RMA2 Full results listing	____________________
RMA2 summary results listing	____________________
RMA2 Hotstart output	____________________
RMA2 final results file	____________________
RMA2 vorticity results file	____________________
RMA2 info on EV/n-value	____________________

GRAPHICAL ANALYSIS:

VELOCITY vector plots	____________________
Time Series plots	____________________

COMMENT:

RMA2 Data Cards

This section describes all of the data cards used by RMA2. 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.

$L Card: Input/Output File Control

Optional

Card Description: Used to specify what types of files RMA2 will read and write.

Field	Variable	Value	Description
0, C 1-2	IC1	$L	Card group identifier.
1	IHOTN		**Hotstart** input.
		0	Initial conditions for RMA2 will be coded in the run control file containing this $L card.
		+	Initial conditions will be read from logical unit #63 (The Hotstart results from the previous run).
2	IHOTO		**Hotstart** output.
		0	No Hotstart file will be written.
		+	RMA2 will write a Hotstart file on unit #62.
3	IGEON		GFGEN geometric data, input for RMA2.
		0	All geometry will be coded in the RMA2 data set.
		+	GFGEN's geometric data opened on unit #60.
4	IFINO		RMA2 solution output (binary final results of hydraulic calculations) will be written to unit #64..
		0	No RMA2 solution output is saved.
		+	The logical unit number for RMA2 solution output.
5	IBUP		Alternate dynamic boundary conditions (input).
		0	No alternate boundary conditions file.
		+	Alternate boundary file read from unit #61.
6	IOUT		Full (standard) results listing file.
		0	No full listing will be created.
		+	Full results listing will be created on unit #3.
7	ISPRT		Summary results listing by node option.
		0	No special list of nodes written.
		+	Summary results listing created on logical unit #59 (TRN card(s) required).
8	IWR33		Detailed steady state parameters when using automatic features (RD card and/or PE card). *Version 4.28 or higher.*
		0	Off
		+	Listing created on logical unit #33.
9	NOPTV		Save vorticity calculation to a file having an RMA4 binary solution format. *Version 4.30 or higher.*
		0	Off
		+	Vorticity calculations saved on logical unit #98.
10	INWAVE		Radiation wave stress input option. *Version 4.42 or higher*
		0	No wave stress input.

+	Read wave stress input file from logical unit #47

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.

Note: If no $L card is present, all file options are off except the full results listing file.

$M Card: Machine Identifier

Required

Card Description: Used to specify the type of computer on which RMA2 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 (i.e. 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.

Example:

$M   1   *Specifies DOS/Windows95-98 PC*

See also: "What Kind Of Computer Do You Have?" on page 29What Kind Of Computer Do You Have?

.

BA Card: Boundary, Azimuth of Flow

Version 4.20 or higher.

Optional

Card Description: Used to specify the azimuth (direction of flow) in radians at the specified nodes. The BA card is typically followed by a BS card describing the same nodes. The BA card specifies the direction and the BS card specifies the speed.

Field	Variable	Value	Description
0, C 1-2	IC1	BA	Card group identifier.
0, C 3	IC3	N	Card type identifier.
1	J1	+	The boundary node number for azimuth ALFAK(J1).
2	ALFAK(J1)	+	The angle at node J1, in radians counterclockwise from the *x*-axis, of the velocity vector (or unit discharge vector).
3	J2	+	Another boundary node number (if desired).
4	ALFAK(J2)	+	The angle at node J2.
5-10	etc.	+	You may continue coding node and angles until the card is full. If necessary continue coding with another BA card.

Note: Either BCC cards or comparable data on BA through BRC cards is required.

FastTABS Note: FastTABS does not support BA cards.

See also: "Changing The Direction Of Flow" on page 91.

BCC Card: Boundary Condition Control Parameters

Version 4.20 or higher.

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	DELT	+	The delta time step length in decimal hours (DELT is revised only if the value is greater than zero).
2	NBX	0, +	The number of BCN cards for which boundary conditions are specified. If zero, the program will *self-count* the number of BCN cards for this time step (if and only if an END card is used to mark the end of the time step).
3	IWIND	-, 0, +	Control for wind field input. **Note:** Reference the BWC card for IWIND values.

Note: If this card is present, any boundary condition parameters specified on previously read cards (TZ card, BW card, or previous BCC cards) will be overruled.

See Also: "Revising Boundary Conditions During A Simulation" on page 92, TZ card, and the REV card

BCN Card: Boundary Conditions by Node

Version 4.20 or higher.

Optional

Card Description: Assigns a boundary condition to the specified node.

The three primary boundary condition parameters can be coded on this card type: *x* velocity component, *y* velocity component, and water surface elevation (head). A five-digit number that tells the program the parameter type is coded, as well as the nodal point number. Code one BCN set for each boundary node.

Note: Both corner and midside boundary nodes require boundary conditions.

Field	Variable	Value	Description
0, C 1-2	IC1	BC	Card group identifier.
0, C 3	IC3	N	Card type identifier.
1	J	+	Code the node number receiving the boundary condition.
2	NFIX(J)		This 5 digit number tells RMA2 what type of boundary condition to use. Coding leading zeros is not required.
		00200	Water surface elevation (head) specified at node J in data field 5.
		01000	Assign a slip flow boundary at node J. **Note:** This is not required in Version 4.2 or higher because slip flow boundaries are automatically generated for edges.
		01200	Combination slip flow with head specified at node J. Supply the water surface elevation (head) value in data field 5.
		11200	Assign a combination of *x*- and *y*-velocity with a water surface elevation (head). Supply values in data fields 3, 4, and 5. **Warning:** Beware of over-specification.
		11000	Both *x*- and *y*-velocity components are specified at node J. Supply these values in data fields 3 and 4.
3	SPEC(J,1)	-, 0, +	The *x*-component of velocity, ft/sec or m/sec.
4	SPEC(J,2)	-, 0, +	The *y*-component of velocity, ft/sec or m/sec.
5	SPEC(J,3)	-, 0, +	The water surface elevation (head), feet or meters.

Note: If BCN cards are not used, comparable data on BA cards through BRC cards is required. Code one node per BCN card.

If NBX on the BCC card is non-zero, and an old format style alternate dynamic boundary file is specified, then BCN cards should be the last set of input cards for a given time step.

See also: "Specifying Boundary Conditions" on page 38.

BH Card: Boundary Head

Version 4.20 or higher.

Optional

Card Description: Used to assign a water surface elevation at the specified node(s) as an alternate to BCC cards.

NFIX (see BCN card) is assigned *xx*200 at each node where boundary head data exist, where the *xx*'s denote optional values which may be assigned by a BS card or BQ card.

Field	Variable	Value	Description
0, C 1-2	IC1	BH	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** The water surface elevation will be used for all boundary nodes equal to or greater than J. **Warning:** Beware of over-specification.
		N	Specifies **Option 2:** The water surface boundary condition is coded at node J.
		L	Specifies **Option 3:** The water surface boundary condition is coded by continuity check line number J.
1	J		The node or continuity 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 *continuity check* line number.
2			Code SPEC(J) or HFX(J) depending on which option was chosen above with IC3 in data field 0, column 3.
	SPEC(J)	-, 0, +	**Option 1:** Water surface elevation (ft or m) for all nodes equal to or greater than J.
	SPEC(J)	-, 0, +	**Option 2:** Water surface elevation (ft or m) for node J.
	HFX(J)	0, +	**Option 3:** Water surface elevation (ft or m) for the continuity check line J.

Note: Only one value per BH card is permitted (comments may follow if you wish).

See also: "Specifying Boundary Conditions" on page 38

BQ Card: Boundary Discharge

Version 4.20 or higher.

Optional

Card Description: Used to assign a discharge at the specified node(s), element(s), or line(s).

This card type can be used instead of the BS cards. The program will assign NFIX on the BCN card as 13*x*00 or 31*x*00 where the *x* denotes the values to be assigned by BH card data.

Field	Variable	Value	Description
0, C 1-2	IC1	BQ	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Use the unit discharge in data fields 2 and 3 for all boundary nodes equal to or greater than J.
		N	Specifies **Option 2:** The *node* number for inflow value.
		E	Specifies **Option 3:** The *element* number for element side inflow.
		L	Specifies **Option 4:** The *continuity check line* number for inflow value.
1	J		The node, element, or continuity 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 *element* number.
			Option 4: The *continuity check line* number.
2	QVEC	-, 0, +	**Option 1:** The unit discharge, (ft^3/s/ft, or m^3/s/m), at all nodes equal to or greater than J.
			Option 2: The unit discharge (ft^3/s/ft, or m^3/s/m) at node J.
			Note: The program will calculate the *x* and *y* components of unit discharge vectors from QVEC by using the azimuth given on the BA card. The signs of the *x* and *y* unit discharge components are calculated from azimuth and grid orientation specified on the BA card.
	SIDF	-, 0, +	**Option 3:** The elemental *inflow* per unit area (for 2D) or unit length (for 1D) as appropriate for the element. Positive values represent inflow, negative values represent outflow.
	QF	0, +	**Option 4:** The total flow (ft^3/s, or m^3/s) crossing the continuity line.

3	QDIR	-, 0, +	**Option 4:** The direction of flow in radians measured counterclockwise from *x*-axis. **Note:** The program adjusts the boundary directions to maintain flow parallel to the edges for nodes at the ends of the continuity line.
4	QXP	0 to 1	**Option 4:** Distribute flow by depth. (default = 0) *Version 4.296 or later.* Decimal values between 0 and 1 are allowed. If QXP = 0, the total flow is equally distributed among all nodes across the continuity check line. If QXP = 1, the total flow is weighted by the depth of each node occurring on the line (shallow nodes get less flow, deeper nodes get more flow).

Note : Only one set of data per BQ card is permitted (comments may follow if you wish). If a BQ card is not used, then comparable data must be provided on either BCC cards or on BA cards through BRC cards.

Tip: If specifying discharge along a continuity line (GC card), the continuity line is recommended to extend from bankline to bankline.

See also: "Specifying Boundary Conditions" on page 38, and "Element Inflow And Outflow" on page 109.

BRA Card: Boundary, Reflection/Absorption

Version 4.26 or higher.

Optional

Card Description: A numerical means of controlling the level of reflected energy at the boundary, from full reflection to full absorption (radiation boundary condition).

Field	Variable	Value	Description
0, C 1-2	IC1	BR	Card group identifier.
0, C 3	IC3	A	Card type identifier.
1	ICON	+	Continuity line number.
2	AC1X	-, 0, +	Average (net) river discharge ft^3/sec or m^3/sec , (Variable Q_o in the equation below).
3	BETA	0 to 1	Reflection/Absorption coefficient used to calculate A_2 in the rating curve equation below. BETA = 1 specifies *total reflection* at the boundary. BETA = 0 specifies *total absorption* at boundary.
4	SSWSE	-, 0, +	The steady state solution for the water surface elevation at the first node on the specified continuity line number. If SSWSE is less than or equal to zero, RMA2 will incorporate the steady state solution (E_o) before advancing to dynamic. **Note:** If Hotstarting a *steady state* run, use a value of -1. If Hotstarting a *non-steady state* (dynamic) run, you must supply SSWSE with at least 4 significant figures past the decimal.
5	THETA	-, 0, +	Direction of flow into the mesh in radians measured counterclockwise from the *x* axis. **Note:** The program adjusts the boundary directions to maintain flow parallel to the edges for nodes at the ends of the continuity line.
6	TAREA	0, +	Surface area of tidal storage beyond the boundary. Used in defining standing wave flows. TAREA is the maximum tidal discharge divided by the maximum fall rate of water surface for a given time step. TAREA = $Velocity_{max}$ * Width * Depth / R_{max} $R_{max} =$ Δ WSELV / (ΔT * 3600)

Note: The special reflecting or non-reflecting (absorption) rating curve described by the following equation supplies a discharge to RMA2.

$$Q = Q_o + A_2 * (ELEV - E_0)^1$$

E_0 in this equation will be supplied after the steady state solution has been computed. Best results are obtained if this feature is applied to a continuity check line of a one-dimensional element.

See Also: "Boundary Permeability (Reflection/Absorption)" on page 93Boundary Permeability (Reflection/Absorption)

BRC Card: Boundary Rating Curve

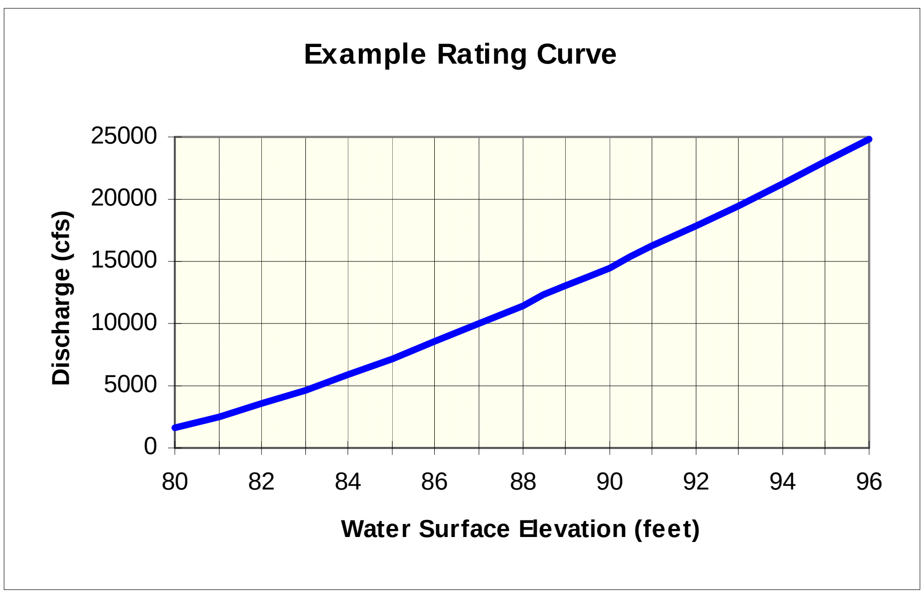
Version 4.20 or higher.

Optional

Card Description: Used to specify a boundary rating curve. The rating curve supplies a discharge to RMA2 using the following equation:

$$Q = A_1 + A_2 * (ELEV - E_0)^C$$

Field	Variable	Value	Description
0, C 1-2	IC1	BR	Card group identifier.
0, C 3	IC3	C	Card type identifier.
1	JCR	+	Continuity line number.
2	AC1X	-, 0, +	A_1 in the above equation.
3	AC2X	-, 0, +	A_2 in the above equation.
4	AC3X	-, 0, +	E_0 in the above equation.
5	AC4X	-, 0, +	C in the above equation.
6	THETA	-, 0, +	Direction of flow (radians measured counterclockwise from + x-axis). **Note:** The boundary directions are adjusted to maintain parallel flow on the exterior edge.



*Discharge = 1000 + 600 * (Elevation - 79)$^{1.3}$*

See also: "Varying Discharge During A Simulation (Rating Curve)" page 92.

BS Card: Boundary Current Speed

Version 4.20 or higher.

Optional

Card Description: Used to specify the magnitude of velocity for the specified nodes. The BS card must be accompanied by a preceding BA card describing the same nodes. The BS card specifies speed and the BA card specifies direction.

The magnitude of the velocity vector is coded on this card type. The input data program will convert BS card data to U and V velocity components using the azimuth on the preceding BA card. The sign of the component is calculated from its azimuth and the specified grid orientation. NFIX on the BCN card is assigned a value of 11*x*00 at each node having a BS value, where the *x* denotes a value to be assigned by the presence of BH card data.

Field	Variable	Value	Description
0, C 1-2	IC1	BS	Card group identifier.
0, C 3	ISI	N	Card type identifier.
1	J1	+	Node number.
2	VVEC1	0, +	Water current speed in ft/sec or m/sec at node J1. The sign will be determined from the azimuth of the vector.
3	J2	+	Next node.
4	VVEC2	0, +	Water current speed at node J2, ft/sec or m/sec.
5-10	*like 3 and 4 above*		Continue coding node number and speed sets until the card is full, then use another BS card.

Note: If BS cards are not used, then comparable data on either BCC cards or on BA cards through BRC cards is required.

FastTABS Note: FastTABS does not support BS cards.

See also: "Changing The Speed Of The Current" on page 92.

BV Card: Boundary Inflow Vorticity

Version 4.30 or higher.

Optional

Card Description:

Field	Variable	Value	Description
0, C 1-2	IC1	BV	Card group identifier.
0, C 3	IC3	L	Card type identifier.
1	J1	+	The continuity line number for inflow value.
2	VOR	0, +	The value of vorticity at the specified continuity line. Usually vorticity is set to zero at the inlet. All other vorticity boundary conditions are zero gradient and require no action to apply.
3	FLD		Comments

Note: The RMA2 vorticity boundary conditions may only be applied to one, and only one, boundary.

See also: “Bendway Correction (Vorticity)” page 93, VO card.

BW Card: Boundary, Wind Speed / Direction

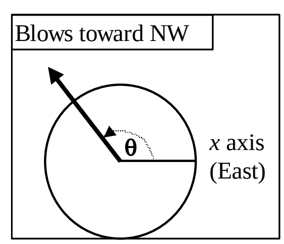
Version 4.20 or higher.

Optional

Card Description: Used to assign wind speed and direction.

Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
0, C 3	IC3		Card type identifier.
		b̸	Specifies **Option 1:** The wind data in data field 2 and 3 of this card will be used for all nodes equal to or greater than J.
		N	Specifies **Option 2:** Wind data are coded by specific *node* number.
		E	Specifies **Option 3:** Wind data are coded by *element* number.
		T	Specifies **Option 4:** Wind data are coded by *material type* number (IMAT).
1	J		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* number.
			Option 2: The *node* number.
			Option 3: The *element* number.
			Option 4: The element *material type* number.
2	TWX(J)	0, +	The wind velocity. **Note:** Units are defined by variable IWMX on BWC card.
3	TAX(J)	-, 0, +	Direction toward which the wind is blowing, measured in degrees counterclockwise from the positive *x*-axis.

For Example, a SE wind, as reported by the conventional meteorological terms, blows toward the NW. This would be an angle of 135 degrees counterclockwise from a positive *x*-axis (with the positive *x*-axis oriented to the east).



Note: Generally, wind forces should not be necessary unless you have a large fetch length (possibly > 100 miles) in the area of interest. The required order for wind assignments should be BWC, BW *blank*, BWT, BWE, and BWN.

See also: "Applying Wind Friction" page 113, and the BWC card.

BWC Card: Boundary Wind Formulation Control

Version 4.20 or higher.

Optional

Card Description: Specifies the numerical formula to be used when calculating wind stress forces. The BWC card must be followed by a BW card type in order to assign wind speed and direction.

Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
1	IWIND		Control for wind field input.
		0	No wind input.
			Code the type of wind stress formula to use:
		1	Specifies **Option 1: Original RMA2** formulation with *default* coefficients.
		2	Specifies **Option 2: Van Dorn** Formula with *user supplied* coefficients.
		3	Specifies **Option 3: Wu** Formula with *user supplied* coefficients.
		4	Specifies **Option 4: Safaie** Formula with *user supplied* coefficients.
		5	Specifies **Option 5: Ekman** Formula with *default* coefficients.
		6	Specifies **Option 6: Generic** Formula with *user supplied* coefficients.
		7	Specifies **Option 7: Van Dorn** Formula with *default* coefficients.
		8	Specifies **Option 8: Wu** Formula with *default* coefficients.
2	IWMX		Flag to identify the units of the wind parameters (recorded at anemometer height = 10 m).
		0	for miles/hour
		1	for meters/second
			Specify data field 3-6 as needed
3	WC1		Data field 3-6 will be the coefficients required for the wind formulation specified by the variable IWIND above. See "Wind Table 1: Wind Special Instructions for descriptions.
4	WC2		
5	WC3		
6	WC4		

Note: Except for option 1, all wind input will be specified in Metric units to conform with the units of the coefficient values used in the wind formulas, regardless of the setting of the SI card. The wind calculation is performed using Metric units. The results of the calculation are converted to match the units as specified by the SI card.

Wind speed and direction are specified on the BW card.

Unless noted otherwise, there are no default values for any parameters for formula options 2, 3, 4, and 6 as listed in Wind Table 1. If there is no value specified for a parameter, zero will be assumed.

If the chosen formula option is 1, 5, 7, or 8 then WC1, WC2, WC3, and WC4 are ignored and the program will assign default values.

Warning: Presently this method is in an evaluation stage and should be used with caution.

See also: "Applying Wind Friction" page 113, and the BW card.

Wind Special Instructions

There are several options for including wind stress in the flow solution. The formulas available in RMA2 are taken from wind stress literature and are listed in Wind Table 2 on the following page. The wind stress formula is selected by setting the IWIND parameter on the BWC card. Wind Table 1 below provides instructions for coding the wind stress parameters.

Wind Table 1. Wind Special Instructions.

Option	Formula	WC1	WC2	WC3	WC4
2	Van Dorn	Currently unused. Code any value to skip this field.	Smooth water wind stress coefficient	Rough water wind stress coefficient.	Critical wind velocity for wave formulation (m/sec).
3	Wu	Currently unused. Code any value to skip this data field.	Wind stress coefficient.	Air Density (g/cm^3). *WC3 > 0, specify the air density in units of g/cm^3. WC3 ≤ 0, default of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).*	Unused.
4	Safaie	Charnock's constant. *WC1 > 0, specify the constant. WC1 ≤ 0, default of .0332 is used.*	Dynamic roughness (cm).	Acceleration due to gravity (cm/sec^2) (affects wind formula only). *WC3 > 0, specify the acceleration in units of cm/sec^2. WC3 ≤ 0, default value of 979.965 cm/sec^2 is used.*	Air density (g/cm^3). *WC4 > 0, specify the air density in units of g/cm^3. WC4 ≤ 0, default value of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).*
6	Generic	Currently unused. Code any value to skip this field.	Wind stress coefficient.	Empirically derived exponent.	Air density (g/cm^3). *WC4 > 0, specify the air density in units of g/cm^3. WC4 ≤ 0, default value of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).*

Wind Table 2. Wind Formulas used in RMA2.

Option	Formula Name	Formula
1	RMA2 Original Formula	$T_S = (3.8E\text{-}6) \cdot W^2$ where T_S = wind stress ($lb/(ft \cdot sec^2)$) W = wind speed (miles/hour) at 10 meters anemometer height *
2	Van Dorn Formula	$T_S = rho_w \cdot (A^2 \cdot W^2 + B^2 \cdot (W - W_{crit})^2)$ where T_S = wind stress ($g/(cm \cdot sec^2)$) W = wind speed (cm/sec^2) at 10 meters anemometer height * rho_w = water density (g/cm^3) A = empirical Alpha coefficient (smooth water) B = empirical Beta coefficient (rough water) W_{crit} = critical wind speed for wave formulation (m/sec)
3	Wu Formula	$T_S = rho_a \cdot C \cdot W^2$ where T_S = wind stress ($g/(cm \cdot sec^2)$) W = wind speed (cm/sec) at 10 meters anemometer height * rho_a = air density (g/cm^3) C = wind stress coefficient
4	Safaie Formula	$T_S = (rho_a \cdot eta \cdot g) / A$ where T_S = wind stress ($g/(cm \cdot sec^2)$) rho_a = air density (g/cm^3) eta = dynamic roughness (cm) A = Charnock's constant (default is 0.0332) g = acceleration due to gravity (cm/sec^2)
5	Ekman Formula	$T_S = (3.2E\text{-}6) \cdot W^2$ where T_S = wind stress ($g/(cm \cdot sec^2)$) W = wind speed (cm/sec) at 10 meters anemometer height *
6	Generic Formula	$T_S = rho_a \cdot C \cdot W^{exp}$ where T_S = wind stress ($g/(cm \cdot sec^2)$) W = wind speed (cm/sec) at 10 meters anemometer height * rho_a = air density (g/cm^3) C = wind stress coefficient exp = empirically derived exponent
7	Van Dorn Formula with defaults	$T_S = rho_w \cdot (A^2 \cdot W^2 + B^2 \cdot (W - W_{crit})^2)$ where default values are W = **10 cm/sec** wind speed (cm/sec^2) at 10 meters anemometer height * rho_w = **1.0 g/cm³** water density (g/cm^3) A = **1.1E-3** empirical Alpha coefficient (smooth water) B = **1.5E-3** empirical Beta coefficient (rough water) W_{crit} = **5.60 m/sec** critical wind speed for wave formulation
8	Wu Formula with defaults	$T_S = rho_a \cdot C \cdot W^2$ where default values are W = **10 cm/sec** wind speed (cm/sec) at 10 meters anemometer height * rho_a = **0.001226 g/cm³** air density (g/cm^3) C = wind stress coefficient ***(see Wind Table 3)***

* *Wind speed and direction are specified on the BW card.*

Wind Table 3. Calculations for Default Wind Stress Coefficient.

Wind speed	**Wind Stress Coefficient, C =**
Wind Speed Less than 100	$1.25 / ((\text{Wind Speed}/100)^{0.2}) \cdot 0.001$
100 < Wind Speed < 1500	$((\text{Wind Speed}/100)^{0.5}) \cdot 0.001 / 2$
Wind Speed Greater than 1500	0.0026

See also: "Applying Wind Friction" page 113, and the BW card.

BWS Card: Boundary, Wind Storm

Version 4.20 or higher.

Optional **EXPERIMENTAL**

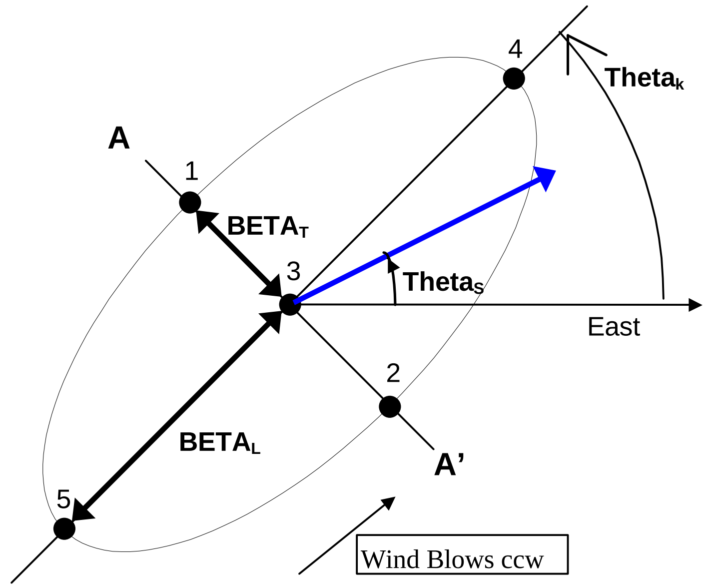
Card Description: Specifies information used to numerically describe storms or frontal passages occurring in the area to be simulated. Each BWS card describes an individual storm. You may simultaneously simulate multiple storm passages (presently dimensioned to 2) to describe complex wind patterns.

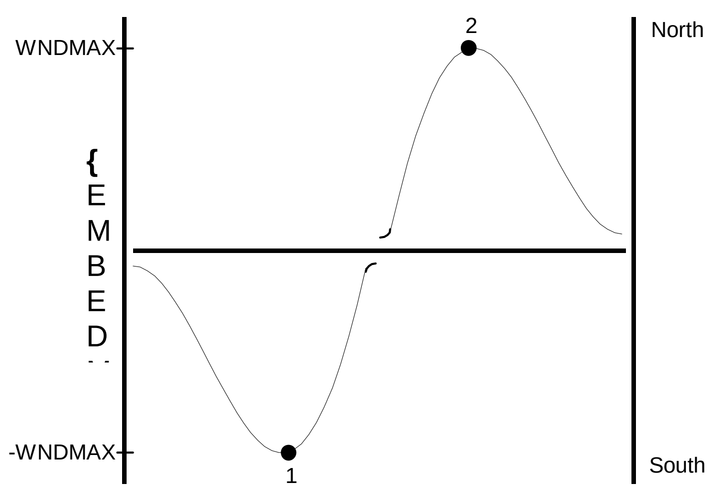
Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
0, C 3	IC3	S	Card type identifier.
1	ISTYPE		Storm reference point (1-5)
		1	The location of maximum winds along the positive minor axis.
		2	The location of maximum winds along the negative minor axis.
		3	The center of the storm.
		4	The location of maximum winds along the positive major axis.
		5	The location of maximum winds along the negative major axis.
2	NREF	+	Reference node for storm track. (for ISTYPE)
3	TREF	+	Reference RMA2 simulation time for storm to arrive at node NREF.
4	SSPD	+	Storm tracking speed, mph.
5	WNDMAX	+	Maximum wind speed, mph (the speed at reference points 1, 2, 4, and 5).
6	WNDMIN	+	Minimum or base wind level, mph (the speed at reference point 3).
7	THETAS	+	Direction of storm track, degrees counterclockwise from positive x-axis toward which the storm is moving.
8	THETAK	+	Orientation of storm (long axis of ellipse) from (0, 180 deg) relative to x-axis in degrees counterclockwise.
9	BETAL	+	Standard deviation of spatial distribution function in the direction of the storm path.
10	BETAT	+	Standard deviation of spatial distribution function in the direction transverse to the storm path.
11	DECAY	+	Exponential temporal decay for wind speed (hr).

Tip: If BETAL is set much larger than the dimension of your mesh, and BETAT small, then a frontal passage can be simulated. Set BETAL >> BETAT, and BETAL >> mesh size.

See also: “Simulating With Storms” page 114, “Applying Wind Friction” page 113, and the BW card.

Storm variable explanation.





Storm Cross Section A to A’

CA Card: Special Calculation Variables

Optional

Card Description: Used to specify modifications to default RMA2 calculation variables.

Field	Variable	Value	Description
0, C 1-2	IC1	CA	Card group identifier
1	USERCA1		Variable to calculate temporal derivative.
		-, 0	1.6 = default used if USERCA1 ≤ 0 or if no CA card is supplied.
		+	Code values between 1.0 and 1.6. **Note:** A value of 1.5 for USERCA1 will specify a second order Taylor Series expansion. **Tip:** Generally, values of USERCA1 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 of USERCA1 has little or no effect. With larger time steps, try values closer to 1.0 if model stability is a problem.

See also: "Adjusting RMA2 Calculations" page 118.

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.

See also: "Any Comments?" on page 52.

DE Card: Wet/Dry by Elemental Elimination

Optional

Card Description: Used to specify information for the Elemental Elimination wetting and drying option. The original wetting and drying method for RMA2, this simplistic technique dries the *entire* element if any one node on that element has a computed depth less than the value specified with the DSET parameter, and rewets the element when all nodes have a computed depth greater than or equal to DSETD.

Field	Variable	Value	Description
0, C 1-2	IC1	DE	Card group identifier.
1	DSET		The depth below which nodes are dry.
			Note: If the values specified on the TI card for the steady or unsteady state convergence criterion are less than zero, then DSET has a secondary purpose. It is involved in the criterion for sufficient depth convergence to allow the code to advance to the next dynamic time step *before* the iteration counter variable (NITN) on the TI card is satisfied. If the maximum depth change is less than ½ of DSET, the code advances to the next dynamic time step.
		0	Default value: 0.275 ft (or 0.084 m).
		+	User defined depth.
2	DSETD		The depth above which nodes become active when re-wetting.
		0	Default value: 0.60 ft (or 0.183 m).
		+	User defined depth.
3	LI		Iteration frequency of testing for wetting and drying.
		0	(Default) Prevents wet/dry testing.
		+	Wet/dry checking will be done at iterations on the specified interval.
			Typically, LI is a positive integer which is a multiple of the iteration counter (See NITI and NITN on TI card).
			Note: LI should never = 1.
			Tip: To permit a wet/dry check only at end of a time step, or revision of a time step (see REV card), set LI to a number larger than the iteration counter specified on the TI card.

Note: Default values are appropriately converted for SI units. If no DE Card is present, the default values are used.

Example:

```
TI        12    4    0.0001  0 .001    Checking is multiple of #iterations requested
DE      .275      .6         4         Wet/Dry checking will be done every 4 iterations.
 .
```

See also: “Wetting and Drying” page 102, “Elemental Elimination” page 105, and the DM card.

DM Card: Wet/Dry by Marsh Porosity

Version 4.25 or higher.

Optional

Card Description: Used to specify information for the Marsh Porosity wetting and drying option. The Marsh Porosity method makes a more realistic and *gradual* transition when wetting and drying than does the method of Elemental Elimination (DE card). All nodes on the element must be flagged dry before the element is considered dry.

Field	Variable	Value	Description
0, C 1-2	IC1	DM	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Identified Node Option (IDNOPT).
		N	Specifies **Option 2:** Set wet/dry option for *node* J.
		E	Specifies **Option 3:** Set wet/dry option for *element* J.
		T	Specifies **Option 4:** Set wet/dry option for *material type (IMAT)* J.
1	J		Code J according to which option was chosen above with IC3 in data field 0, column 3...
			Option 1: Code J as one of the following (IDNOPT = J):
		0	Turns off the Marsh Porosity option. All DM cards are ignored.
		-1	RMA2 uses default values for data fields 2 through 5 for *all* nodes (user specified values are ignored).
		-2	User specifies the values in data fields 2 through 5 for *all* nodes.
		+	User specifies the values in data fields 2 through 5 for all nodes ≥ J.
		+	**Option 2:** Code J as the *node* number. The values in data fields 2 through 5 will be used for this node.
		+	**Option 3:** Code J as the *element* number. The values in data fields 2 through 5 will be used for all nodes comprising this element.
		+	**Option 4:** Code J as the *material type (IMAT)* number. The values in data fields 2 through 5 will be used for all nodes comprising each element of this material type.
2	AC1 *	+	Distance from A0 to minimum regional bed elevation (Default = 3.0 ft or 0.91 m).
			Note: A0 = average regional (nodal area) bed elevation (the '**z**' value on GFGEN GNN card). If AC1 is ≤ 0, see AC4 below.
3	AC2 *	+	Transition range of the distribution (Default = 2.0 ft

			or 0.61 m).
4	AC3 *	+	Minimum wetted surface area factor (Default = .02).
5	AC4 *	+	Minimum regional bed elevation. If AC1 ≤ 0, AC4 = MIN(AC4, A0-ABS(AC1)). If AC4 = 0, AC1 is used.

** See the figures below for definitions of parameters for fields 2 through 5.*

The order in which DM cards should appear is: DMb̸ (DM blank), DMT, DME, and finally, DMN. If a DMT card, for example, appears after a DME card, the DME card is potentially overruled by the DMT card..

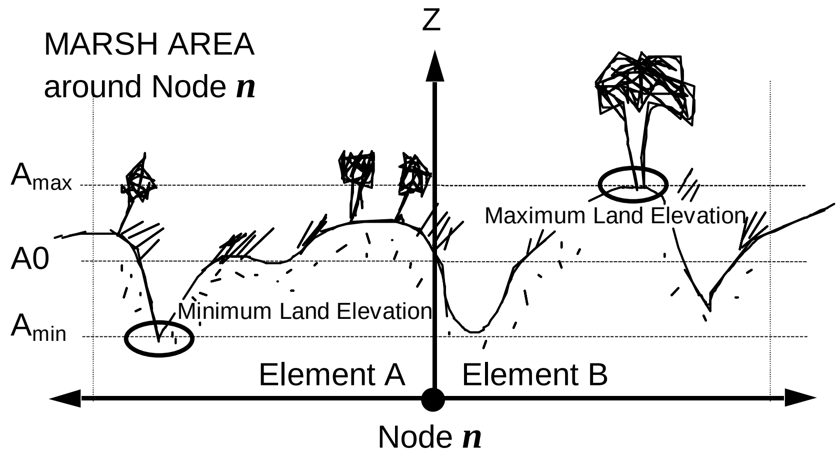
Note: To include computations for the Marsh Porosity option, at least one DMb̸ (DM blank) card is required, then optionally followed by DMT, DME, or DMN cards (in that order). If a node receives multiple assignments, the last assignment is processed.

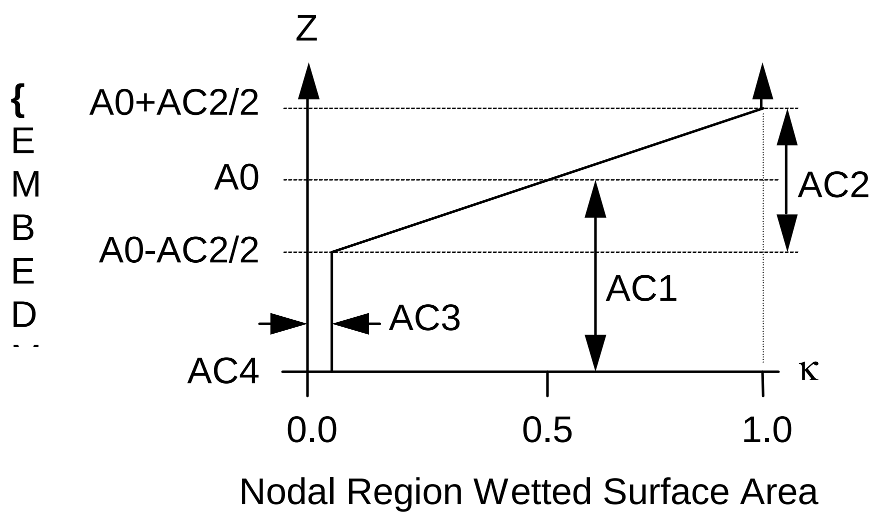
The values for AC1 through AC4 are used as an "initial guess" for RMA2.

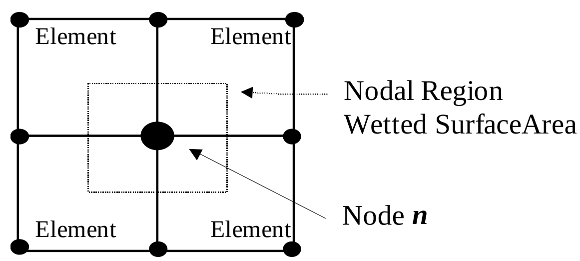
If an element becomes completely wet, DM card information for all nodes comprising that element is ignored until the water depth of the element falls within the transitional range and begins to dry.

Default values are automatically converted to metric if the SI card indicates System International units.

See also: "Wetting and Drying" page 102, and the DE card.







Where:

A0 = average nodal area bed elevation (‘**z**’ value from GFGEN GNN Card). **A0** is the mean land elevation in the vicinity of node ***n***.

AC1 = distance from **A0** to minimum regional bed elevation.

AC2 = transition range of the distribution.

AC3 = minimum wetted area of the distribution.

AC4 = minimum regional bed elevation.

*“Regional” refers to the “nodal area”; the area in the immediate vicinity of node **n**.*

END Card: End Card

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.

See also: "Specifying Boundary Conditions" page 38.

EV Card: Turbulent Exchange Coefficients and Roughness

Required if a PE card, SM card, RD card or HN card, or, an EX card and EY card combination is not used

Card Description: Used to manually supply turbulence in the *x* and *y* directions, as well as a roughness value for the specified material type.

Field	Variable	Value	Description
0, C 1-2	IC1	EV	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,1)	+	Exx = the *x*-momentum of turbulent exchange in the *x*-direction (lb-sec/ft^2 or Pascal-sec for SI-units)*.
3	ORT(J,2)	+	Exy = the *x*-momentum of turbulent exchange in the *y*-direction (lb-sec/ft^2 or Pascal-sec for SI-units)*.
4	ORT(J,3)	+	Eyx = the *y*-momentum of turbulent exchange in the *x*-direction (lb-sec/ft^2 or Pascal-sec for SI-units)*.
5	ORT(J,4)	+	Eyy = the *y*-momentum of turbulent exchange in the *y*-direction (lb-sec/ft^2 or Pascal-sec for SI-units)*.
6	ORT(J,5)	+	Manning's n roughness coefficient (or Chezy if ≥ 3.0).

* *1 lb-sec/ft^2 = 47.879 Pascal-sec.*

Note: Turbulent exchange coefficients (eddy viscosity) and roughness should be coded for every element material type < 900 represented in the mesh.

Recall that Eddy Viscosity (E) and Peclet number are inversely related. The formula for Peclet numbers (P), where P is recommended to range between 15 and 40.

$$P = \frac{\rho u dx}{E}$$ where ρ = fluid density, u= velocity magnitude, dx=element length

Tip: In many cases it is required to Coldstart the RMA2 model with "large" values of eddy viscosity (i.e., about 500 lb-sec/ft) to achieve convergence. The eddy viscosity may be incrementally decreased by using REV cards or a series of Hotstart runs.

See also: "Modeling Turbulence" page 46, "How Methods of Assigning Turbulence Interact with One Another" page 100, EX, EY, HN, PE, and RD card.

EX Card: Turbulent Exchange Coefficient, X-Velocity

Required if EV card or PE card is not used

Card Description: Used to manually supply the momentum of turbulence in the *x* direction. These values are specified according to material type.

Field	Variable	Value	Description
0, C 1-2	IC1	EX	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,1)	+	Exx = the *x*-momentum of turbulent exchange in the *x*-direction (lb-sec/ft^2 or Pascal-sec for SI-units).
3	ORT(J,2)	+	Exy = the *x*-momentum of turbulent exchange in the *y*-direction (lb-sec/ft^2 or Pascal-sec for SI-units).

Note: Values for Eyx and Eyy are coded on the EY card.

See also: "Modeling Turbulence" page 46

EY Card: Turbulent Exchange Coefficient, Y-Velocity

Required if EV card or PE card not used

Card Description: Used to manually supply the momentum of turbulence in the *y* direction. These values are specified according to material type.

Field	Variable	Value	Description
0, C1-2	IC1	EY	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,3)	+	Eyx = the *y*-momentum of turbulent exchange in the *x*-direction (lb-sec/ft^2 or Pascal-sec for SI-units).
3	ORT(J,4)	+	Eyy = the *y*-momentum of turbulent exchange in the *y*-direction (lb-sec/ft^2 or Pascal-sec for SI-units).

Note: Values for Exx and Exy are coded on the EX card.

See also: “Modeling Turbulence” page 46

FC Card: Flow Control Structures for 1D/2D

1D available in Version 4.20 or higher, 2D available in Version 4.40 or higher

Optional

Card Description: Used to specify parameters to simulate flow control structures such as dams, weirs, culverts, etc.

Field	Variable	Value	Description
0, C 1-2	IC1	FC	Card group identifier.
1	NJN	+	Flow controller identifier (IMATs ≥ 904).
2	NJT		Flow controller type...
		1	Point source of flow, i.e., **pump, storm drain**, etc. Conditions are flow out equals flow in plus source (FLZ3), and equal total head at each node of the control element.
		2	Flow is a *reversible* function of head loss across the structure, i.e., **reversible weir, open lock, open culvert**, etc. Conditions are flow out equals flow in, and flow Q = FLZ3 + FLZ4 * (HN1 - HN2 - FLZ5)**FLZ6 in the direction FLZ7. **Note:** If HN1 - HN2 < 0 , the sign of the flow direction is reversed.
		3	*An irreversible type 2 flow*. Flow is an *irreversible* function of head loss across the structure, i.e., a **flapped culvert**. Conditions are flow out equals flow in, and flow Q = FLZ3 + FLZ4 * (HN1 - HN2 - FLZ5)**FLZ6 in the direction FLZ7. **Note:** If HN1 - HN2 - FLZ5 < 0 then Q = 0.
		4	Flow is a function of water surface elevation, i.e., a **dam, weir**, etc. Conditions are flow out equals flow in, and flow Q = FLZ3 + FLZ4 * (HN1 - FLZ5)**FLZ6 in the direction FLZ7.
		5	An optional way of specifying a type 2 flow controller above. *Head loss* is a function of flow Q. Conditions are flow out equals flow in, and head loss HN1 - HN2 = FLZ3 + FLZ4 * (Q)**FLZ6 in the direction FLZ7.
		6	*An irreversible type 5 flow*. Flow structure is irreversible. The head loss is a function of flow Q when the head loss is positive. If the head difference across the structure is negative, the flow is set to zero. HN1 - HN2 = FLZ3 + FLZ4 * (Q)**FLZ6 when (HN1 - HN2) > 0. Q = 0 when (HN1 - HN2) ≤ 0.
		7	*Undocumented*
		8	*Undocumented*
		9 *	*Pump a designated flow rate when the elevation is above a specified criterion.* Q = FLZ4 if water surface elevation > FLZ5
3	FLZ3		Base flow Q in cubic feet per second (cfs or cms).

4	FLZ4	Relational coefficient.
5	FLZ5	Reference elevation or head difference (ft or m).
6	FLZ6	Exponent.
7	FLZ7	Direction of flow (radians, counter-clockwise from the positive *x*-axis).

HN1 and HN2 are the water surface elevations at the first and second nodes of the control structure element.

Tip: When multiple flow control structures are defined in the same mesh, it is generally good practice to have at least one of these structures defined such that the flow, Q, is a function of head loss, ΔH (type 2 or 3), and one such that ΔH is a function of Q (type 5 or 6).

For cases where extremely small head losses are encountered, the types 5 and 6 flow controllers may be numerically more stable.

*Type 9 is available for 1D elements only.
Example of a pump capacity of 30 and trigger elvation of 4:
FC 904 9 0.0 30.0 4.0 0 1.5708

See also: “Using Control Structures” page 75.

FD Card: Fluid Density

Version 4.00 or higher.

Optional

Card Description: Used to supply a fluid density for all nodes or a specified node.

Field	Variable	Value	Description
0, C 1-2	IC1	FD	Card group identifier.
0, C 3	IC3		Card type identifier.
		b̸ (blank)	Specifies **Option 1:** Universal assignment for all nodes ≥ J.
		N	Specifies **Option 2:** Assignment by individual node.
1	J		The node or starting node number as specified by IC3 above.
		+	**Option 1:** The *starting* node number.
		+	**Option 2:** The node number.
2	RON(J)	+	Fluid density at node J (slugs/ft^3 or kg/m^3 for SI units).

Densities for Fresh Water and Sea Water.

Type of Water	English Units slugs/ft^3	Metric Units kg/m^3
Fresh Water	1.935	998.46
Sea Water	1.990	1026.84

Note: If no FD card is present, the density of fresh water is used.

Example:

```
.
.
FD        1.99        Use the density of seawater.
.
.
```

See also: "Fluid Density", page 99

FT Card: Water Temperature

Version 4.20 or higher.

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.

See also: “Fluid Temperature” page 99.

~~G1 Card~~: Geometry, General Geometry Parameters

Optional (No longer supported)

Card Description: This card has been replaced by the GS card, LA card, and BCC card. The G1 card replacement method is summarized below:

- Use LA card to supply an average Latitude used for Coriolis force calculations
- Use GS card to specify an *x* and *y* scaling factor for the mesh
- Use BCC card to specify the number of boundary conditions per time step to be found in the input

Field	Variable	Value	Description
0, C 1-2	IC1	G1	Card group identifier.
1	OMEGA	+	Average Latitude in degrees, local to the study area, used in calculating the Coriolis forces. Default is OMEGA = 0 to turn off Coriolis. **Note:** LA cards with the T option provide for Coriolis by IMAT
2	XSCALE	+	Scale factor for X-coordinates. Default is 1.0.
3	YSCALE	+	Scale factor for Y-coordinates. Default is 1.0.
4	NBX	0, +	Specifies the number of nodes with boundary conditions specified with BCN cards. Default is NBX=0 and program will self-count the BCN cards. **Note:** This data field is no longer necessary for normal operation. It is only necessary when some old style input format. It is recommended to use the default value.

Note: If no G1 card or replacement cards are present then:

- Coriolis is not applied (unless an LA card is used)
- Coordinate scale factors are 1.0 (unless a GS card is used)
- BCN cards are self counted

See also: BCC card, GS card, LA card.

~~GC Card~~: Geometry, Continuity Check Line

Optional

This card is no longer supported. It was replaced by the GCL card.

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

Field	Variable	Value	Description
0, C 1-2	IC1	GC	Card group identifier.
1	LMT(K)	+	Total number of nodes to be listed for the continuity check line described by this GC card.
2	LINE(J,K)	+	List the node numbers 1, 2, ..., total number of nodes specifying the check line. **Note:** If there are more node numbers specifying a flow continuity check line than will fit in the data fields remaining on the current GC card, continue coding the remaining node numbers starting in data field 1 of the next GC card.

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

Note: Specify the dominant boundary line (inflow or outflow) first since that line is used in calculating the percent of the total flow at all subsequent lines. Code corner nodes only.

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, flow rate results may be mis-interpreted since some flow rates may be reported in the printout as positive and some as negative.

Example: *old way* using GC Card

```
CO  Line#1   contains 11 Corner Node numbers...
CO  Line#2   contains  6 Corner Node numbers...

GC    11     10   11   12   13   14   15
GC           16   17   18   19   20
GC     6    100  101  102  109  107  99
```

See also: "Checking For Continuity" on page 54, GCL card.

GCL Card: Geometry, Continuity Check Line

Version 4.30 or higher.

Optional

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: GCL Card

```
The GCL Card permits the line # to be assigned, and the end of
the line is marked with a negative number.

CO  Line#    Corner Node numbers...Negative marks end of line.
GCL    1     10   11   12   13   14   15
GCL          16   17   18   19   20   -1
GCL    2     100  101   102  109  107  99  -1
```

Note: In general, code the <u>corner nodes</u> making up the continuity check line from right to left when facing downstream. Specify the dominant boundary line (inflow or outflow) first since that line is used in calculating the percent of the total flow at all subsequent lines. Code corner nodes only.Code all lines in the same direction (i.e., right to left across the flow), otherwise, flow rate results may be mis-interpreted since some flow rates may be reported in the printout as positive and some as negative.

Tip: When running a steady state problem, continuity check lines can be used to help verify that the model is providing acceptable results by comparing the flow rate results at the points specified with the corresponding flow rates from field data. The flow rate across the line will be reported in the full results listing file.

See also: "Checking For Continuity" on page 54.

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	TH(J)	-, 0, +	Direction of eddy viscosity tensor in **radians**, counterclockwise from the 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 RMA2 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 RMA2 run control file to make these types of changes.

See also: "Modifying Elements" page 88, GN card, GT card, GV card, and GZ card.

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.
3	CORD(J,2)	-, 0, +	*y*-coordinate at node J.
4	AO(J)	+	Bottom elevation.
5 (1D)*	WIDTH	+	Channel width at zero depth for node J.
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. **Note:** This feature is primarily used for model verification purposes.
9	WSCRIT	-,+	Water surface elevation to activate Off-Channel storage
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 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 Note: As of this writing, Neither FastTABS nor SMS support the one-dimensional parameters on the GN card. Use GWN.

GS Card: Geometry, Scale Factors

Version 4.30 or higher.

Optional

Card Description: This card applies scale factors for the *x*, *y*, and/or *z* 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.
3	ZSCALE	+	Scale factor for Z-coordinates (bottom elevation).

Note: If no GS card is present then all scale factors are 1.0.

See also: "Resizing The Mesh" on page 91

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: "Changing An Element's Material Type" on page 89, "Removing Elements" on page 89.

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	TH(J)	-, 0, +	Direction of eddy viscosity tensor (Radians, counterclockwise from the *x*-axis)

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

Note: RMA2 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: "Modeling Turbulence" on page 46, GE card.

GW Card: Geometry, Channel Width Attributes

Version 4.20 or higher.

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.
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.
6	WSCRIT	-,+	Water surface elevation to activate Off-Channel storage
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 1D channel width attributes or 2D off-channel attributes as opposed to the GN card.

FastTABS will read and interpret, but will not update GW card data. Be aware that if the grid is renumbered, you will have to update the GW cards manually.

See also: "Using Special Elements" on page 71, "Off-Channel Storage" on page 103, GN card, figure and description beginningon page 6.

GZ Card: Geometry, Nodal Point Elevation

Version 4.30 or higher.

Optional

Card Description: Used to specify or modify the bottom elevation of the specified nodes.

Field	Variable	Value	Description
0, C 1-2	IC1	GZ	Card group identifier.
0, C 3	IC3		Card type identifier
		ƀ (blank)	Specifies **Option 1:** A constant bottom elevation will be used for all nodes >= J.
		T	Specifies **Option 2:** Bottom elevations by element material type (IMAT).
		N	Specifies **Option 3:** Bottom elevations by nodal point will be used.
1	J		The starting node or node number as specified by IC3 above. **Note:** The nodes should be corner nodes.
		+	**Option 1:** The *starting* node number.
		+	**Option 2:** The element material type number.
		+	**Option 3:** The node number.
2	AO(J)	+	Bottom elevation.
3-10	etc.	+	Continue entering node number/elevation sets.

Note: Code only corner node numbers. The bottom elevations are multiplied by the scale factor ZSCALE on the GS card. If desired, you may fill the GZ card in complete node/elevation sets, or continue on another GZ card.

Tip: You can use GZ cards to create a dredged channel in your mesh. This technique is useful for making base/plan comparisons because you can leave the original geometry data intact. To remove the dredged channel, you only have to remove the GZ cards which define the channel bottom elevation from the RMA2 run control file.

Tip: If you are experiencing problems diagnosing instabilities in an RMA2 simulation, you can use GZ cards to specify that all or part of the mesh has a flat bottom elevation. This technique may help pinpoint the trouble spot.

See also: "Changing Bottom Elevations" on page 91

HN Card: Roughness, Manning n-Value

Required if an EV card or RD card is not used

Card Description: Used to assign a Manning's n roughness value to the entire grid, or to an individual element material type.

Field	Variable	Value	Description
0, C 1-2	IC1	HN	Card group identifier.
0, C 3	IC3		Card type identifier.
		b̸ (blank)	Specifies **Option 1:** Use a constant n-value for all *elements* ≥ J.
		E	Specifies **Option 2:** Use a constant n-value for element J.
		T	Specifies **Option 3:** The n-value is specified by element *material type* (IMAT).
1	J		Code the element or material type as specified by IC3 above.
		+	**Option 1 & 2:** The *element number*.
		+	**Option 3:** The element *material type*.
2	ORT	+	Manning's n-value. If (ORT > 3) then the Chezy coefficient is used.

Note: It is advisable to always use an HN-*blank* card to set the n-value for all elements in the mesh. An EV card and/or RD card may then be used to override this n-value.

Tip: The table below suggests some Manning's n-values for various types of surfaces.

Manning's n-value Guidelines.

Manning's n-value	Condition
.015 - .020	Smooth earth with no weeds.
.020 - .025	Sand channel.
$.034\ (d_{50})^{1/6}$	Rip-rap channels (d_{50} = the particle size of which 50% of the mixture is finer; i.e., 50% of the mixture is finer than this particle size).
.075 - .150	Very winding/overgrown.

Reference US Army Corps of Engineers publication EM 1110-2-1601 "Hydraulic Design of Flood Control Channels", Appendix J: "Methods for Predicting n-values for the Manning Equation

See also: "Bed Friction And Resistance To Flow" on page 45, EV card, RD card.

HW Card: Hydrodynamic Wave Stress

Required if variable INWAVE on the $L card is activated

Version 4.42 or higher.

Card Description: Used to assign the radiation wave stress options to the entire mesh, or to an individual element material type.

Field	Variable	Value	Description
0, C 1-2	IC1	HW	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Set wave parameters for the entire mesh.
1	IWAV		Flag for performing adjustments to wave model input
		1	Perform indicated adjustments
		0	No adjustments are made, Default
2	IWCUR		Flag to activate/deactivate wave/current interaction
		1	Activate wave/current interaction
		0	Turn off wave/current interaction, Default
3	PWCORR	+	Time shift for starting time in wave model input file (hr)
4	HSFACT	+	Factor applied to the wave height
5	HSLIM	+	Maximum allowable wave height (meters)
			Where: $\max = \overline{H} + 2\sigma$
6	HSVAR	+	Scaling factor for the variance about the mean wave height. A value of 0.0 would return a constant mean wave height over the entire model.
7	HPFACT	+	Factor applied to the wave period.
8	WPLIM	+	Maximum allowable wave period (seconds)
9	WPVAR	+	Scaling factor for the variance about the mean wave period. A value of 0.0 would return a constant mean wave period over the entire model.

Note: Only the global mesh assignments for radiation wave stress option is available.

This is an experimental feature. For more information, see the section entitled "Applying Wave Radiation Stress" on page 111.

IC Card: Initial Conditions

Required

Card Description: Used to provide initial conditions to start a cold run.

Field	Variable	Value	Description
0, C 1-2	IC1	IC	Card group identifier.
1	ELEV	+	Average initial water-surface elevation (feet or meters). **Tip:** The most stable way to start a model run is to make sure that all nodes in the entire mesh are wet. In general, a *minimum* depth of about 2 feet on all nodes should be acceptable.
2*	HMIN	+	Minimum depth used for one-dimensional nodes at startup. **Note:** Eliminates possible negative depth on sloping river systems.
3*	UNOM	+	Nominal velocity for one-dimensional nodes. Used as the initial guess if not restarting. Suggested value = .25 fps.

* *Used for one-dimensional elements only.*

Note: If HMIN and UNOM are specified with the elemental wetting and drying option activated (DE card), then no wet/dry checks will occur in the first iteration.

HMIN and UNOM are unused for fully two-dimensional networks.

See also: "Specifying Initial Conditions" on page 31

LA Card: Local Latitude

Version 4.25 or higher.

Optional

Card Description: Used to specify the latitudinal position, north or south, of the area of interest, measured in degrees from the Earth's equator. This card provides information for the Coriolis Force correction calculation, which is used to account for angular velocity effects on bodies of water due to the Earth's rotation.

Field	Variable	Value	Description
0, C 1-2	IC1	LA	Card group identifier.
0, C 3	IC3	b̸ (blank)	Specifies **Option 1:** Global assignment of average Latitude of the mesh by element.
		T	Specifies **Option 2:** Assigns specific Latitude by material type IMAT.
1	ISTART	+	**Option 1:** Code the first *element* number for which global assignment is to begin.
	IMAT	+	**Option 2:** Code the element *material type.*
2	OMEGA	+	Latitude in degrees. (Set OMEGA=0 to turn off Coriolis)

If desired you may fill the card with (IMAT, OMEGA) sets, or use more LAT cards. Latitude (Coriolis) may be *globally* assigned on the G1 card.

Note: Generally, the Coriolis Force correction should not be necessary unless you have a wide water surface (possibly > 100 miles) in the area of interest.

Warning: Instabilities on the southwest quadrant of the solution have been observed when using Coriolis Force correction. If this is a concern, a workaround is to apply graduated values of latitude (Coriolis). This is done by setting the latitude value for elements near the area of instability to zero, and increment the value of latitude in rows or layers of elements toward the point in the mesh where you want the value of latitude to be the true latitude. An increment of about 25% of the true latitude should be sufficient.

See also: "Compensating For The Rotation Of The Earth" on page 111

PE Card: Peclet Method for Assigning Automatic Turbulence

Version 4.28 or higher.

Optional

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

Field	Variable	Value	Description
0, C 1-2	IC1	PE	Card group identifier.
0, C 3	IC3	ƀ (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 eddy viscosity by Peclet number (Zero turns the option off).
	IMAT	+	**Option 2:** *Material type* for the Peclet parameter assignment.
2	GPEC	+	Peclet number. **Note:** A smaller Peclet number will result in a larger value for eddy viscosity, 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 (ft/sec, or meters/sec) to be used to compute the automatic eddy viscosity equation. **Note:** Remember that if the SI card indicates Metric units, the velocity must be input in Metric units.
4	EPSXX	+	Scaling factor for computed eddy viscosity in the *xx* direction from the Peclet calculation (default = 1).
5	EPSXY	+	Scaling factor for computed eddy viscosity in the *xy* direction from the Peclet calculation (default = 1).
6	EPSYX	+	Scaling factor for computed eddy viscosity in the *yx* direction from the Peclet calculation (default = 1).
7	EPSYY	+	Scaling factor for computed eddy viscosity in the *yy* direction from the Peclet calculation (default = 1).

Tip: In many cases it is required to Coldstart the RMA2 model with "large" values of eddy viscosity (i.e., about 500) to achieve convergence. The eddy viscosity may be incrementally decreased by using REV cards or a series of Hotstart runs.

Example:

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

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), where P is recommended to be between 15 and 40.

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

where:

Coefficient			English Units	Metric Units
ρ	=	fluid density	1.94 slugs/ft^3	998.46 kg/m^3
u	=	average elemental velocity	fps	mps
dx	=	length of element in streamwise direction	ft	m
E	=	eddy viscosity	lb-sec/ft^2	Pascal-sec

See also: "Automatic Turbulence Closure Assignment" on page 100, EV Card, and SM Card.

RA Card: Rainfall And Evaporation

Version 4.297 or higher.

Optional

Card Description: Used to assign rainfall or evaporation at the specified elemental location or via material type association.

Field	Variable	Value	Description
0, C 1-2	IC1	RA	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	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.
1	J		The element or material type number as specified by the value of IC3 above in data field 0, column 3.
		+	**Option 1:** The *starting element* number.
		+	**Option 2:** The *element* number.
		+	**Option 3:** The element *material type* number.
2	SIDF	-, 0, +	The elemental *inflow* in inches/hour (or cm/hr) Positive values represent rainfall (inflow), negative values represent evaporation (outflow).

Note : A positive value for SIDF represents rainfall, while a negative value represents evaporation.

Junction elements and other elements with material types greater than 900 cannot be assigned rainfall or evaporation.

See also: "Specifying Boundary Conditions" on page 38, "Adding Rainfall And Evaporation" on page 110

RD Card: Automatic Roughness Coefficient Assignment by Depth

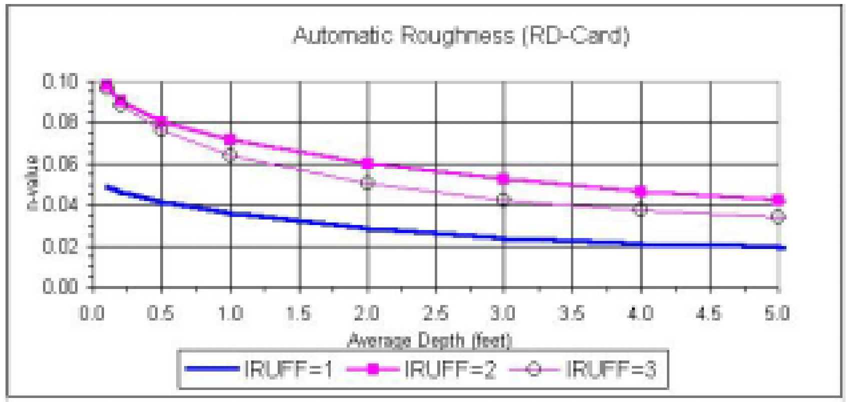
Version 4.28 or higher.

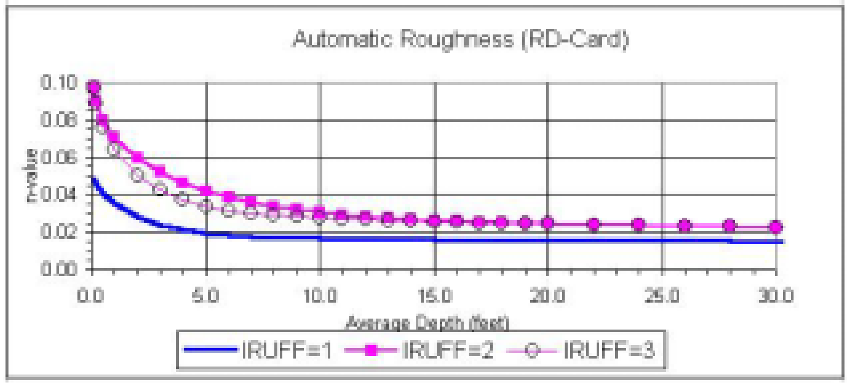
Optional

Card Description: Provides for real time adjustment of the bottom roughness coefficient. Automatically changes the Manning's n-value of an element depending upon the depth at each Gauss point. Generally, the deeper the water, the lower the roughness value.

Field	Variable	Value	Description
0, C 1-2	IC1	RD	Card group identifier.
0, C 3	IC3		Card type identifier.
		ƀ (blank)	Specifies **Option 1:** Global assignment for element numbers ≥ ISTART.
		T	Specifies **Option 2:** Set n-value by element *material type* (IMAT).
		E	Specifies **Option 3:** Set n-value by element *number*.
1	ISTART	+	If **Option 1:** Code the *first* element number for global assignment to begin.
			If **Option 2:** Code the element *material type*.
			If **Option 3:** Code the element *number*.
2	IRUFF	-, 0, +	On/Off switch for auto Manning's n-value calculations based upon an average depth (AVEDEP in the equation) of the element. A positive value turns the option on and sets defaults as in the table on the next page. A negative value or zero will turn the option off.
3	RDR0	0, +	Maximum Manning's n-value for non-vegetated water (a positive value will override the default).
4	RDD0	0, +	Depth at which vegetation effects roughness (a positive value will override the default).
5	RDRM	0, +	Manning's n-value for vegetated water (a positive value will override the default).
6	RDCOEF	0, +	Roughness by depth coefficient (a positive value will override the default).

Note: Default parameters are set as in the table on the next page depending upon the value of IRUFF.





$$NVALUE = RDR0 / (AVEDEP^{RDCOEF}) + (RDRM * EXP(-AVEDEP/RDD0))$$

Default values for n-value parameters on RD card.

IRUFF	Project used for	RDR0	RDD0	RDRM	RDCOEF
1	Miss River Delta project	.02	2.0	.026	.08
2	S-shaped River example test case	.04	4.0	.040	.166667
3	San Francisco Bay Estuary project	.04	2.0	.040	.166667

See also: "Varying Discharge During A Simulation (Rating Curve)" on page 92, EV card, and HN card.

REV Card: Revise the Current Time Step

Version 4.25 or higher.

Optional

Card Description: Used to signal the RMA2 model to solve for the current time step. RMA2 will read data cards until it sees a REV card. At that point it will perform calculations and obtain an intermediate solution that will be used as *initial conditions* for the next series of data cards.

This card can be used as an alternative to Hotstarting. It can be used to gradually alter parameters to obtain the desired values.

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

Example: REVise Eddy Viscosity and discharge during steady state.

```
T3
$M
$L
 . . .
EV      1    500   500   500   500    0.03
TI      4      0    .01
 . . .
BQL     6  50000   -1.2   0.5
BHL     7    100
REV     Now lower coefficients and increase Q for 3 iterations
TI      3      0    .01
EV      1    300   300   300   300  0.027
BQL     6  75000   -1.2   0.6
REV     Now lower coefficients more and increase Q
EV      1    150   150   150   150  0.025
BQL     6 100000  -1.2   0.7
END      Marks the end of the time step
```

Solve for Hour 0.0 (at first REV)

Solve for Hour 0.0 given new data (at second REV)

Solves for Hour 0.0 and writes the solution using EV = 150 and Q = 100000 and flow distribution by depth = 0.7. (at END)

See also: "Revising Boundary Conditions During A Simulation" on page 92

.

RSC Card: Restart/Hotstart Read Control

Version 4.35 dated Feb 1998 or higher.

Optional

Card Description: Used to specify the exact time step and/or iteration to retrieve from the restart input file for the initial condition specification.

Field	Variable	Value	Description
0, C 1-2	IC1	RS	Card group identifier.
C3	IC3	C	
1	HOT_TET		Simulation hour to retrieve from the Hotstart file
2	IT_UVH		Hydraulic iteration to retrieve (uvh)
3	IT_VOR		Vorticity iteration to retrieve

Note: If this card is omitted, the first restart/hotstart record on the input file is used as the initial condition.

See also: "Resuming A Stopped Simulation" on page 80

SI Card: System International

Optional Units

Card Description: Used to control the type of units RMA2 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 are applied (default).
		1	Metric units are expected as input and used for output.

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, English units are used.

See also: "Specifying Units" on page 84

SM Card: Smagorinski Method for Automatic Turbulence Assignment

Optional

Card Description: Used to control the horizontal eddy viscosity and diffusion by Smagorinski.

Field	Variable	Value	Description
0, C 1-2	IC1	SM	Card group identifier.
0, C 3	IC3	ƀ (blank)	Specifies **Option 1:** Set Smagorinski parameters for the entire mesh.
		T	Specifies **Option 2:** Set Smagorinski parameters for the element *material type* (IMAT).
1	JSMAG	+	Material type for the Smagorinsky parameter assignment.
2	TBFACT	-,0,+	Smagorinsky coefficient for turbulent exchange. A negative value applies the default coefficient= 0.1
3	TBFACTS	-,0,+	Smagorinsky coefficient for diffusion (Vorticity). A negative value applies the default coefficient= 0.1
4	TBMINF	-,0,+	Smagorinsky minimum turbulent exchange factor. A negative value applies the default coefficient= 1.0
5	TBMINFS	-,0,+	Smagorinsky minimum diffusion (Vorticity) factor. A negative value applies the default coefficient= 1.0

Note: The SM card assignment for a given material type will override any EV and/or PE card parameters for that same material type. See the section entitled "Automatic Turbulence Closure Assignment" on page 100.

See also: EV-Card and PE-Card

STO(P) Card: Stop the RMA2 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.

See also: "Stopping The Simulation" on page 50, END card.

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 RMA2 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 FastTABS - John Doe
T3    Mississippi River Gulf Outlet -- mean tide
  .
  .
```

See also: "Using Titles" on page 28, T3 card.

T3 Card: Job Title

Required

Card Description: Used to tell RMA2 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 RMA2 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. RMA2 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 RMA2 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 FastTABS – John Doe, 24 Jan 1991
T3    Mississippi River Gulf Outlet -- mean tide
  .
  .
```

See also: "Using Titles" on page 28, T1-T2 cards.

TI Card: Number of Iterations

Version 4.25 or higher.

Required

Card Description: Used to specify the iteration and convergence criterion for satisfactorily completing a time step.

Field	Variable	Value	Description
0, C 1-2	IC1	TI	Card group identifier.
1	NITI	0, +	The maximum number of iterations for the initial solution (use for steady state Coldstart/Hotstart).
2	NITN	0, +	The maximum number of iterations for each dynamic time step(s).
3	SSDCRT	-	Uses DSET * .5 (from DE Card).
		0, +	User specified criterion for satisfactory depth convergence during steady state simulation (Default= 0).
4	USDCRT	-	Uses DSET * .5 (from DE card).
		0, +	User specified criterion for satisfactory depth convergence during an unsteady (dynamic) simulation (Default = 0).

Note: If you want the depth criterion to be based on ‘DSET’, the DE card must precede the TI card.

Tip: When Hotstarting, parameter NITI should be coded as zero only if no more iterations are desired for the initial steady state conditions.

See also: "Iteration Control" on page 32, TZ card and DE card.

TO Card: Time for Saving the Binary Results Solution File

Optional

Card Description: Used to prevent RMA2 from writing to the solution file until the simulated hour matches a time specified.

Field	Variable	Value	Description
0, C 1-2	IC1	TO	Card group identifier.
1-10	TBINRY(J)	0, +	Time in decimal hours at which you want to save the final results to the binary file (Requires NOPT on the $L card to be active). You may continue to list times until the card is full. If necessary, continue the list with another TO card.

Note: The decimal hour specified on this card must comply with the actual simulated hour (to five places past the decimal point).

If no TO card is specified, all time steps are written to the solution file.

Tip: Suppose you only need to save results at every simulated hour, but you want to make calculations at every simulated quarter-hour. You can use TO cards to specify the specific simulation hours you want to save in the solution file.

Example:

```
   .
   .
TO    25      26      27      28      29      30
TO    31      32      33
   .
   .
```

In this example, results are written to the solution file starting at simulated hour 25, then continue to be written at every simulated hour until simulated hour 33.

See also: "Customizing The Solution File" on page 84

TR Card: Full Results Listing Control

Required

Card Description: Used to specify what information will be contained in the full results listing file.

Field	Variable	Value	Description
0, C 1-2	IC1	TR	Card group identifier.
1	IPRT		Print control pertaining to startup conditions.
		0	Node and element input data is suppressed.
		1	Write all input data except initial conditions.
			Note: Suggested when Coldstarting.
		2	Write all input data including initial conditions obtained from the results of the previous run (Hotstart).
			Note: Suggested when Hotstarting.
2	ITSI		Control pertaining to nodal results. The number of iterations/time steps between successive writes to the full results listing file.
		0	Do not print nodal results.
		1	All *iterations* are written.
		2	Every other *iteration*.
		3	Every third *iteration*.
		...	Etc...
		-1	Every *time step* (last iteration only).
		-2	Every other *time step* (last iteration).
		-3	Every third *time step* (last iteration).
		...	Etc...
3	IECHO		Switch to control HEC type card input data echo to screen (a.k.a. terminal or log file).
		0	No input data echo (default).
		1	Run control input is echoed.
4 *	ITRACE		Trace subroutine calls and controller diagnostics will be written to the screen (a.k.a. terminal or log file).
		0	No trace.
		1	Trace report for each subroutine call…
		2-3	Degree of detail increases with the choice.

* *Used for source code debugging purposes only.*

See also: "Customizing The Full Results Listing File" on page 86

TRN Card: Summary Results Listing Control

Optional

Card Description: Used to provide hydrographic information (velocities, depths, etc.) in a summary results listing file for the nodes specified.

Field	Variable	Value	Description
0, C 1-2	IC1	TR	Card group identifier.
C3	IC3	N	Card type identifier.
1-10	NSPLPT(J)	+	List of node numbers for the results listing summary. You may list nodes until the card is full. If necessary, continue the list with another TRN card.

Tip: Summary results listing data can be easily imported into a spreadsheet program to create charts and graphs, and to perform further analysis.

See also: "Requesting A Summary Of Nodal Results" on page 87

TS Card: Timing For Binary Solution Output

Version 4.35 dated Feb 1998 or higher.

Optional

Card Description: Used to control the time interval for writing time steps or iterations to the binary solution file.

Field	Variable	Value	Description
0, C 1-2	IC1	TS	Card group identifier.
1	IBHO		Scheme for saving RMA2 binary (u, v, h) solution (Default = -1).
		0	Never save the solution.
		+n	Save every n^{th} *iteration.*
		...	etc.
		-1	Save results at the end of each *time step*.
		-n	Save results at the end of every n^{th} *time step*.
		...	etc.
2	IBVO		Scheme for saving RMA2 binary vorticity solution (Default = 0).
		0	Never save the vorticity solution.
		+n	Save every n^{th} *iteration*.
		...	etc.
		-1	Save results at the end of each *time step*.
		-n	Save results at the end of every n^{th} *time step*.
		...	etc.
3	IBRO		Scheme for saving RMA2 binary hotstart/restart. (Default=0)
		0	Only the last converged hour/iteration of the hotstart/restart results are saved
		+n	Save every n^{th} *iteration.*
			(the last iteration of a time step is saved)
		-1	Save at the end of every *revision* and every *time step*
		-n	Save at the end of every n^{th} *time step*.

Note: It is advised to save only time steps in the solution file, not iterations. The ability to save iterations is intended for source code debugging and diagnostics.

See also: "Solution File" on page 28, "Vorticity Solution File" on page 28, TO card.

TV Card: Iteration Control For Vorticity Calculations

Version 4.30 or higher.

Required if IVOR on the VO card is 1.

Card Description: Used to control the number of iterations for vorticity calculations.

Field	Variable	Value	Description
0, C 1-2	IC1	TV	Card group identifier.
1	NPASS1	+	The maximum number of passes between the steady state (u, v, depth) and vorticity iterations.
2	NVITI	+	The maximum number of steady state vorticity iterations.
3	NPASS2	+	The maximum number of passes between the (u, v, depth) and vorticity iterations per dynamic time step.
4	NVITN		The maximum number of dynamic vorticity iterations.
5	NGOODMAX	+	The number of good passes through a series of (u, v, depth) and vorticity calculations before the solution has converged for the time step. "Good passes" means the solution has converged before NVITI or NVITN is met.

Note: All RMA2 calculations will stop if NGOODMAX is achieved before NPASS1 or NPASS2 is reached.

See also: "Bendway Correction (Vorticity)" page 93, BV card, VO card.

TZ Card: Computation Time

Required

Card Description: Used to control simulation time.

Field	Variable	Value	Description
0, C 1-2	IC1	TZ	Card group identifier.
1	DELT	+	Specifies the length of the computation time interval for a dynamic run, in decimal hours.
		0	Specifies that steady-state conditions will be used.
2	TMAX	0, +	The total run time in decimal hours.
3	NCYC	+	The total number of time steps (cycles).
4	NSTART		Specifies how many END cards to skip in the RMA2 run control input file before starting.
		0	Coldstart/Hotstart run. If REV cards are present in the *first time step*, then they will be processed.
		1	Coldstart/Hotstart run. Ignores any REV cards in the first time step.
		1+	Hotstart run. Skips through the RMA2 run control input file, keying on END cards, until it reaches the NSTART set. RMA2 then reads those boundary condition values to resume computations. For a Hotstart run, this should equal the last time-step of the prior run plus 1 minus MBAND.
5	MBAND		MBAND is the logic control variable that directs the program to either skip to the next time-step (MBAND=0) or remain at the last time-step of the previous run (MBAND=1) for the first computations of the Hotstart run
		0	Initial run or restarting on the first iteration in the convergence scheme
		1	Restart at an intermediate dynamic iteration in the convergence scheme. (Activates the Time terms in the equations)

Note: RMA2 execution will stop whenever either TMAX *or* NCYC have been satisfied.

See also: "Total Simulation Time" on page 37, BCC card, TI card.

VO Card: Vorticity (Bendway Correction)

Version 4.30 or higher.

Optional

Card Description: Used to supply values associated with the calculation of vorticity.

Field	Variable	Value	Description
0, C 1-2	IC1	VO	Card group identifier.
1	IVOR	0, 1	On/Off switch for the vorticity (bendway correction) option. Vorticity is active if IVOR = 1. Default = 0.
2	SSVCRT	0, +	Steady state vorticity convergence criterion (refers to vorticity iterations only). Recommend: 0.00001. Default is .5 * DSET on the DE card.
3	USDVCC	0, +	Dynamic vorticity convergence criterion (refers to vorticity iterations only). Recommend: 0.00001. Default is .5 * DSET on the DE card.
4	ASEC	+	ASEC coefficient for the vorticity equation. Recommend: 5.0
5	DSEC	+	DSEC coefficient for the vorticity equation. Recommend: 0.5
6	RCMIN	+	Minimum radius of curvature that will be allowed. Recommend: 6 feet (or 2 meters).

See also: "Bendway Correction (Vorticity)" page 93, BV card, TV card.

Performance Enhancements

If you have the RMA2 source code and a FORTRAN 77 or 90 compiler, you can customize RMA2 to better conform to your system. With a little experimentation and manipulation of array dimensions, you may be able to realize considerable performance improvements.

Why Is RMA2 So Slow?

RMA2 is a complex numerical model which performs many calculations per time step to obtain a solution. To run, 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 RMA2 will be able to solve your problem. Unless you do not need results in a hurry, it is not recommended to run an RMA2 simulation of any significant size on a computer with less power than a 400 Mhz PC. In fact, a mainframe computer system or a workstation environment is the preferred setting.

Temporary Files

RMA2 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 RMA2 is running.

The process of writing to and reading from temporary files places a significant strain on the performance of RMA2, 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 "Reducing The Number Of Temporary Files" on page 200.

Disk Performance

The speed at which your disk can transfer data will have some influence on the time that RMA2 will take to complete your simulation. Especially if RMA2 is writing temporary files, a slow disk will kill 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.

Reducing The Number Of Temporary Files

If your system has a large amount of RAM, say 32 Megabytes or more, it is likely you can improve performance by dimensioning RMA2 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. To accomplish this, change the variable NBS in the RMA2 include file to a value that 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.

If you wish to customize the dimensioning to your problem, the memory size required to keep the solution matrix completely within memory can be determined from the RMA2 output. Locate on the key words **"BUFFER BLOCKS"**or **"FINAL LQ SIZE",** which are located just prior to the convergence parameters**.** The required dimensioning size of the parameter variable NBS to fit the problem fully in memory may be calculated as:

new_NBS = old_NBS * (# BUFFER BLOCKS written) + Final LQ SIZE

If NBS is set larger than this number, then on subsequent model runs the reported number of buffer blocks written will be zero.

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, RMA2 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 signs this is happening are that RMA2 is 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 while running entirely in RAM.

Redimensioning RMA2

RMA2 is written in standard FORTRAN 77 syntax.

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 RMA2. These array dimensions can be changed in the source code by editing the RMA2 include file and recompiling. The array dimension variables are as listed in the table below.

Variable	Description
MEL	Maximum number of elements in the mesh.

MND	Maximum number of nodes in the mesh.
MR1	Maximum number of equations. This is normally MND * 2.
MCC	Maximum number of continuity check lines.
MCCN	Maximum number of nodes per continuity check line.
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). **SMS Note:** The Mesh Information tool in SMS will provide a good estimate for the front width of your mesh.
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.
MAXSTRM	Maximum number of storms simulated with BWS cards.
NBS	Maximum solution matrix buffer size (in RAM). This dimension influences the number of temporary files written.[*]

[*] See the section entitled "Reducing The Number Of Temporary Files" on page 200

Common Problems

Common Problems and Remedies

The Simulation Stops Prematurely

Cause 1

The Depth convergence parameter has exceeded a critical value of **25.0**.

Remedy

Examine the convergence parameters in the full results listing file to determine the first time step where divergence is noticed, then check for:

- negative depth at a node
- high velocities over shallow zones
- the Froude number is beginning to approach one
  Froude # = velocity / SQRT(gravity * depth)
- large depth gradients around the zone
- performing a check for wetting and drying before the model has converged to a solution.
- irregular wet/dry patterns
- incompatible boundary conditions
- large boundary break angle
- abnormal Manning's n-value roughness, or eddy viscosity assignments
- non-smooth boundary condition signal
- delta time step assignment, etc.

If the problem isn't obvious, graphical tools are available.

Cause 2

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 RMA2.
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 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 RMA2 run. If there is adequate RAM available, recompile with a larger buffer size. See "Reducing The Number Of Temporary Files" on page 200.

An Element Has a Negative Area

Cause

The element connection was made clockwise rather than counter clock wise.

Remedy

Correct the corresponding GE card(s) in the GFGEN geometry file, or in the RMA2 run control file if the element is defined there.

Parameters for 1D Width and/or Off-Channel Storage Are Wrong

Cause 1

The 1D parameters of width, side slope, and off-channel storage, as well as the 2D off-channel storage capability are advanced concepts and therefore are infrequently used features of RMA2 modeling. The graphical user interfaces have historically had difficulty keeping these parameters intact, especially during mesh modification and renumbering. It is advisable to check these parameters after processing the mesh with FastTABS and/or SMS.

Remedy

These described inconveniences while simulating complex RMA2 applications (San Francisco 1D-2D for example), resulted in a new utility program, **GET_1D**, to check and restore all of these parameters.

These errors have been reported and are expected to be fixed in the year 2000.

An Element Has a Zero Area

Cause 1

Determine if the element is a special element. All control structures and junction elements will have zero area by definition.

Remedy

No remedial action is required.

Cause 2

An element with *zero* area may indicate that no eddy viscosity has been specified for the material type associated with the element.

Note: Junction and control structure elements have zero area by definition.

Remedy 1

Include an EV card (or use automatic turbulence methods) for all material types <900 which are used in the mesh.

Cause 3

Either the element does not exist in the network or the element material type has been set to zero (usually not fatal).

Note: A material type can be intentionally set to zero to remove the element or elements from the mesh. This technique is acceptable.

Remedy

Renumber the mesh.

Inconsistency In Units

Cause

The SI card metric flag does not match the GFGEN geometry.

Remedy

Verify which type of units you need to use and set the SI card appropriately in both the GFGEN geometry file and the RMA2 run control file.

Hotstart Difficulties

Cause 1

Improper settings for Hotstarting

Remedy

Double check these hotstart control parameters:

- IHOTN > 0 on the $L card
- IPRT = 2 on the TR card
- NITI = desired value on TI card
- NSTART and MBAND variables on the TZ card
- Verify the time steps/iterations made available for re-starting, TS Card.

- Verify the above with what was requested on the RSC Card.

Note: The program counts END cards, not REV cards, to determine how many boundary condition time steps to skip for a Hotstart run.

A message is printed to the screen or logfile indicating when a Hotstart file is saved. The file named "*r2hot.dat*" summarizes which records were written to the Hotstart file. It is created when ITRACE=1 on the TR Card.

Hotstarting with an *old-style* alternate boundary condition file is not supported.

Cause 2

Geometry changes were made between Hotstart runs. The Hotstart immediately diverges.

Remedy

If any bottom elevation or other mesh editing has occurred between Hotstart runs, or some other abrupt geometry related change was added to the run control file, the old Hotstart file will no longer match the geometry and is useless. The simulation must be restarted from the beginning. NOTE: The Hotstart file may be "repaired" (if only minor geometry changes were made) with the utility program, "R2_HOTFIX". See the utilities section for more details.

Cause 3

The initial conditions read from a Hotstart file generated from a previous RMA2 run do not match the first set of active boundary conditions. A diverged solution is the result.

Remedy

The NSTART value on the TZ card is wrong, or the wrong hotstart record was requested on the RSC 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. Set variable ITSI to a large negative value to limit full print output (TR card).
3. Only save critical time steps (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 or core dumps were deleted.
7. If there is enough RAM available, set the buffer size dimension very large so the problem will run in memory rather than create temporary files.
8. Purchase more disk space.

Array Bounds Underflow

Cause

A node or element number is zero or negative.

Remedy

Examine the GFGEN geometry and/or RMA2 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.

Execution Terminated Because of 1D Nodes Lacking a Width Assignment.

Cause

No GWN card are present.

Remedy

Either re-run GFGEN with GN cards or GW cards with the N option to specify width, or add these cards to the RMA2 run control file.

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

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 RMA2 input files.

Note: If using an alternate boundary file, the first character of the file must be a 'B' for boundary or a 'C' for comment.

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 "Moving RMA2 Binary Solution Across Platforms" on page 231.

Execution Running Slower Than Expected

Cause

There can be many reasons for RMA2 to run slow. Below are a few suggestions for improving performance. See "Performance Enhancements" on page 199 for more information.

Remedy

1. If there is ample RAM available, try increasing the dimension of the parameter statement in the RMA2 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 at a location 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.

See Also: "Warning And Error Messages" on page 209

Warning And Error Messages

Warning Messages

AMW is Zero or Negative

Cause

This results from poor element shape or aspect ratio. Typically this is not fatal ... unless an element is flagged multiple times (16 maximum/element) and velocities become large.

Remedy

Edit the associated elements in the geometry to meet the suggested shapes. See the GFGEN Users Guide.

ARRAY Overrun In CRACK

Cause

Input parameters on a card go beyond column 80, or possibly a real or character value was read in an integer data field.

Remedy

Use a continuation card to code more data.

Auto Parameter LU Turned Off Via $L

Cause

The variable controlling a report of automatic parameter setting, IWR33, was set to zero on the $L card.

Remedy

Ignore, or stop and turn this feature on if desired.

BCN Card For Node= '*NODE*' illegal

Cause

A new boundary condition assignment was attempted with a BCN card after a REV card or an END card, but this node number was not initially assigned with a BCN card for steady state.

Remedy

All boundary conditions must be initially defined for the steady state portion of the simulation. New boundary conditions cannot be added in mid simulation. Correct the run control file.

COEFS ASC (variable) near 0

Cause

Subroutine COEFS detects that the alignment of the boundary rating curve is near zero, and a division by this number could cause problems.

Remedy

Check the parameters on the BRC card.

Fear Of Dividing By Near Zero In COEFS

Cause

The water density value, ROAVG, is near or equal to zero.

Remedy

Be sure the water density value is specified properly on the FD card.

Final Binary Is Turned Off Via $L

Cause

The $L card NOPT variable is set to not save the final solution to a file.

Remedy

This is typically not done, since what you generally want from RMA2 is the solution. Ignore this warning, or if you need a solution, correct the NOPT variable on the $L card.

Full Print Is Turned Off Via $L

Cause

The $L card IOUT variable is set to not save the full results listing file.

Remedy

Ignore this warning, or if you need a full results listing, correct the IOUT variable on the $L card.

Geometry Was Not Defined Prior To *xxx* Card

Cause 1

GE cards were not read prior to the card in question in the RMA2 run control file.

Remedy

Change the card order RMA2 run control file by placing the $L card and GE cards before the card in question.

Cause 2

The $L card is set to not read the binary geometry.

Remedy

Change the IFILE variable on the $L card so that the binary geometry is read.

Infinite Loop Error

Cause

The GCL card did not have the “-1” terminator flag to designate the end-of-line.

Remedy

Check the RMA2 run control file and correct the GCL card(s).

Insufficient Information On *xxx* Card

Cause

All parameters were not completed on the card in question.

Remedy

Check the RMA2 run control file and correct the card.

IVRSID (Machine ID) = ?? Is Not Permitted

Cause

An invalid value for machine identifier was specified. The default value of 4 will be substituted

Remedy

Check the $M card in the RMA2 run control file and provide a valid value which is correct for your machine type.

Negative Depth Calculated during Simulation

How to Identify if you have Negative Depths

The traditional approach to finding occurrences of negative depth was a tedious look through the full results listing or by examining the depths contours (in FastTABS or SMS) from the interval of 0.0-1.0.

A relatively new feature (RMA2 version 4.51 or higher) will echo to the full results listing and to the “screen” a warning message only if a negative depth occurs for that time step. The message appears immediately following the statistical results for a given time step/iteration:

NEGATIVE DEPTH WARNING: For entire mesh= 1 For active mesh= 1

---> Suggestion: Set ITRACE=1, TR Card. This will save a file = r2negdepth.dat

The file r2negdepth.dat is actually in a SCAT2D format that can be readily modified on the first line to include the total number of points. It can then be read into SMS as a two-dimensional scatter data file and graphically illustrate all nodes that experienced a negative depth within the simulation. The ###, must be manually edited to respresent the total number of (x,y,z) values in the SCAT2D file prior to reading it into SMS.

Example: Diagnosing negative depth with Scatter 2D Data .

```
SCAT2D
XYD ###   NEG_DEPTH_SCATTER  5  XVEL YVEL DEPTH WSELV NDRY   time/iter
3750.9   343.0  21.60   0.0 0.0   21.600  -0.186   2 Time/Iter= 2.0   1
3623.4   353.6  21.49   0.0 0.0   21.490  -0.078   2 Time/Iter= 2.0   1
3761.8   424.3  21.47   0.0 0.0   21.470  -0.065   2 Time/Iter= 2.0   1

4520.4  2069.4  20.30   0.0 0.0   20.300  -0.029   2 Time/Iter= 6.0   4
```

Cause

This is predominately a wetting and drying issue. The most likely cause is a jagged wet/dry edge(s). A wise modeler will not ignore a negative depth warning.

Remedy

One or more of the following steps will be required to eliminate this problem: Modify the mesh using the guidelines for good element and mesh properties. The use of marsh porosity may solve the problem. Other considerations would include: reducing the time step, not allowing any of the boundary condition specifications to "dry"

Node xxx Formed Dead End Without BC Specified

Cause

There is not a boundary condition assignment at the node specified.

Remedy

Specify a valid boundary condition for the specified node.

Reordering List Overruled If GFGEN Is Read

Cause

If a GO card is used in RMA2 (not recommended), it will overrule the reordering list that was read in by GFGEN.

Remedy

The use of a GO card is not recommended in RMA2. You are advised to reorder using GFGEN.

Storm Array Over-Run

Cause

You have requested or assigned more storms than the RMA2 is dimensioned to handle.

Remedy

Decrease the number of storms (BWS cards), or change the value of the MXSTRM variable to increase the number of storms RMA2 can simultaneously simulate.

Unused Elements

Cause

The element numbering is not sequential.

Remedy

Renumber the mesh, or proceed with caution.

Value of ISWTCH Invalid in CONVRT

Cause

Valid values are 1, 2, and 3.

Remedy

This is an internal program error. Please report via e-mail to **tabs@hl.wes.army.mil**.

You Just Turned Off *xxxxxx*

Cause 1

The $L card is set to not read or write the file in question.

Remedy

Ignore the message, or change the $L card.

Cause 2

The text “**null**” was entered at the interactive filename prompt.

Remedy

If you want to use this file, rerun RMA2 and enter the filename.

See Also: "Common Problems" on page 203.

Error Messages

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

Array Over-Run In Sub Check

Cause

Too many nodes were assigned on the stated continuity check line after the midside nodes were automatically inserted.

Remedy 1

Attempt to move the continuity check line to another location with less resolution.

Remedy 2

Increase the variable MCCN in the source code and recompile.

Cannot Modify Control Structure Not Defined At Beginning

Cause

An attempt was made to revise a control structure, which has not been initially defined.

Remedy

Define the control structure at the beginning of the run.

Cannot Rain In Junction Or Control Structure

Cause

Rainfall or evaporation was specified via RA card for a junction element or control structure element. These element types have zero area.

Remedy

Remove all specifications of rainfall and evaporation from the RMA2 run control file for any elements with material types greater than 900.

See Also: "Adding Rainfall And Evaporation" on page 110

Card Input Complete With xxx Errors

Cause

If any errors have occurred, the RMA2 simulation will stop after the initial run through and report the number of errors encountered.

Remedy

Tend to the errors and rerun.

Card xxx Affects A 1D Element That Lacks A Width

Cause

An RA card was applied to a one-dimensional element which lacks a width assignment.

Remedy

Use a $L card to read the geometry prior to the RA card, or use a GW card with the N option to assign a width to at the one-dimensional node.

Card xxx Continuity Line Number xxx Has A 1st Node xxx

with an undefined width xxx or bogus bottom elevation

Cause

This error occurs when using Boundary Reflection and Absorption. The BRA type card works on one-dimensional elements, and uses a continuity line assignment.

Remedy

You must define the one-dimensional element prior to using the BRA card. Use a GC card, GN card with the N option, and/or GW card with the N option to define the element.

Card xxx Out Of Bounds Continuity Check Line

Cause

A continuity check line on the indicated card was determined to be negative, zero, or larger than the maximum number of continuity check lines RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the number of continuity lines, or increase the MCC parameter (maximum number of continuity check lines) in the source code and recompile.

Card xxx References An Out Of Bounds Element

Cause

An element on the indicated card was determined to be negative, zero, or larger than the maximum number of elements RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MEL parameter (maximum number of nodes) in the source code and recompile.

Card xxx References An Out Of Bounds Material Type

Cause

An element material type on the indicated card was determined to be negative or larger than the maximum number of elements RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MEL parameter (maximum number of nodes) in the source code and recompile.

Card xxx References An Out Of Bounds Node

Cause

A node on the indicated card was determined to be negative, zero, or larger than the maximum number of nodes RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MND parameter (maximum number of nodes) in the source code and recompile.

Depth Convergence Exceeds 25.0

Cause

The program stops and no solution is saved for the time step.

Remedy

See "The Simulation Stops Prematurely" on page 203

If the problem isn’t obvious, graphical tools are available.

File Not Found

Cause

RMA2 cannot locate the specified file.

Remedy

Check to see that the filename is not misspelled, or that a wrong path was entered. Also, be sure the file is located where you believe it to be.

GC Card For Continuation Of Previous GC Was Expected

Cause

The continuity check line was not continued correctly. RMA2 expected a continuation GC card as a result of the corner node counter specified on the first GC card for the line.

Remedy

When using the GC(blank) Card (no longer supported):

For a given continuity line, the GC card contains a corner node counter as the first parameter, which tells RMA2 the number of corner nodes making up the continuity line. When the number of nodes is more than can fit on one GC card, a continuation card is required to continue the set. These continuation cards contain only the corner nodes and no counter value. There must be as many corner nodes coded as defined by the corner node counter on the first GC card for the line.

When using the GCL Card:

The GCL card contains only corner nodes. A negative number marks the end of the node list. The GCL card differs from its predecessor GC(blank), in that no counter variable is required. All continuation lines of for type of assignment must begin with GCL.

GFGEN Banner Shows Metric (*or English*) Units

(but SI card indicates English (*or metric*))

Cause

The SI card metric flag in RMA2 does not match the GFGEN geometry.

Remedy

Verify which type of units you need to use and set the SI card appropriately in both the GFGEN geometry file and the RMA2 run control file.

GFGEN Geometry Exceeds Program Dimension

Cause

The total number of node and/or elements in the geometry has exceeded RMA2 dimensions.

Remedy

Edit the source code and increase the variables MND for nodes and MEL for variables to appropriate sizes and recompile. Be aware that you must have enough RAM to load RMA2 after this change. See "Redimensioning RMA2" on page 200 for more information.

Illegal Card Type

Cause

The values contained in columns 1 through 3 in the RMA2 run control file did not reference a valid RMA2 data card.

Remedy

Check the typing and be sure you are trying to use a valid card type.

Length & Size Are Inconsistent With Dimension Capabilities

Cause

Subroutine CONVERT contains an error converting between integer and character variables.

Remedy

Internal program error. Please report via e-mail to **tabs@hl.wes.army.mil**.

Logical Unit # is Zero

Cause

Attempt to read non-defined alternate boundary condition input file. If the run control file is prematurely short, or contains no END card, RMA2 will switch to an alternate input file for more information.

Remedy

Specify that an alternate boundary condition input file is to be read with the IBUP variable on the $L card, or put all boundary condition data in the initial run control file.

Next Input Record Is Too Long

Cause

The information on a data card exceeds 80 characters. The maximum amount of data allowed on a data card is 80 characters, including any spaces at the end of the line.

Remedy

Check the data card indicated and eliminate any unnecessary spaces.

NMAX is Not Sufficiently Large

(to permit the assembly of the next element ... Increase MFW parameter ...)

Cause

The mesh was not efficiently reordered in GFGEN

Remedies

1. Check the GFGEN GO card, and try other locations for ordering starting points.
2. Increase program dimensions of the front width.
3. Reduce the size of the mesh.

RMA2 Stops in Subroutine CHECKDATE ... Fatal Error

Cause

The expiration date of the executable has expired. Typically the expiration date is 30-days beyond the end of the fiscal year (30 Oct).

Remedy

Contact the vendor from whom you obtained RMA2 model. If eligible, you may apply for an update or newer release of the model.

RMA2 Stops in Subroutine REVHYD ... Fatal Error

Cause

The continuity check line stated was not previously assigned by the stated card type at the beginning of the run.

Remedy

Correct the run control file by defining the continuity line.

RMA2 Has A Revise *xxx* Error

Cause

RMA2 cannot revise the card in question because it has not been initially defined.

Remedy

Correct the run control file by incorporating the card in question somewhere at the beginning of the run.

Side Slopes At Node xxx Non-zero

Cause

A one-dimensional node was assigned a side slope, but RMA2 would not allow it here. For instance, a one-dimensional transition node cannot have a left or right side slope.

Remedy

Remove any side slope specifications at the specified node.

Size Larger Than 1200

Cause

Array problem in subroutine CONVERT. Please report via e-mail to **tabs@hl.wes.army.mil**.

Remedy

Internal program error. Please report via e-mail to **tabs@hl.wes.army.mil**.

"T" Card Expected, Run Terminated

Cause

The RMA2 run control file does not contain a valid Title card at the beginning of the file.

Remedy

Put a valid Title card at the beginning of the run control file. A blank line at the top of the file is not allowed.

The User Must Put The SI Card Before Specifying The Density

Cause

Misplaced SI card.

Remedy

The SI card should immediately follow the $L card.

This File Is Mandatory

Cause

You cannot eliminate this file because it is required for the RMA2 simulation.

Remedy

Provide a valid file name when prompted, and do not attempt to "null".

Too Many Continuity Check Lines

Cause

The number of continuity check lines specified via GC cards exceeds the RMA2 dimension for the maximum continuity check lines.

Remedy

Reduce the number of continuity check lines for your simulation, or increase the MCC parameter (maximum continuity check lines) in the source code and recompile.

Too Many Nodes In The Above Continuity Check Line

Cause

The number of nodes specified via GC cards per continuity check lines, plus the midside nodes which are automatically included, exceeds the RMA2 dimension for the maximum number of nodes per continuity check lines.

Remedy

If possible, break the indicated continuity check line into two individual lines using two GC cards, or increase the MCCN (maximum nodes per continuity check line) parameter in the source code and recompile.

Unsatisfied Elimination Error

Cause 1

There are LCOL number of problem locations.

The contents of LHED are EQUATION Numbers...

Make note of the equation number(s)

NODE number ... NBC ARRAY IS ....

node # VelX-eqn#1 VelY-eqn#2 Head-eqn#3

Find the node causing the problem in the listed table, and check for:

1. Element connection problem.
   For example, two nodes having an identical (*x*, *y*) coordinate that are not one-dimensional Junction or Control Structure elements.
2. Erroneous boundary conditions.
   Example: specifying all water surface elevations with no inflows.
3. Fragmented mesh domain.
   Part of the mesh has dried and caused ‘puddles’.
4. Not correctly specifying the FC card parameters for a control structure.

Remedy 1

Search the NBC ARRAY to match the EQN# echoed to determine which node number(s) are involved. Then closely examine the mesh in that area.

Cause 2

Subroutine FRONT cannot eliminate the element because its solution matrix pivot value is less than PIVOT_CRIT (default 1E-4). This may happen when all velocities are very near zero.

Remedy 2

Edit the RMA2 source code. Decrease the value of variable PIVOT_CRIT inside subroutine FRONT and recompile.

See Also: "Common Problems and Remedies" on page 203.

RMA2 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 (120-129 for RMA2 results) (135-139 for RMA2 Hotstart file)
IREC	Version number of the RMA2 program
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 221.

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

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 221.

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.

RMA2 Binary Hotstart File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 221.

Records 5 (u-v-h arrays)

TET, NP, NE, NITSV, ((VEL(j,k),J=1,3),k =1,NP),

((VDOT(j,k), j=1,3), k = 1, NP),

((VOLD(j,k), j=1,3), k = 1, NP),

((VDOTO(j,k), j=1,3), k = 1, NP),

(NDRY(k), k = 1, NP), (HEL(k), HOL(k), HDET(k), HDOT(k), k = 1, NP),

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

Records 6 (vorticity arrays)

NITSV, (VOR(k), VOROLD(k), VORDOT(k), VORDOTO(k), k=1,NP)

where

TET	Simulation time, in decimal hours
NP	Number of nodes in the mesh
NE	Number of elements in the mesh
NITSV	u-v-h Iteration number for this record
VEL	Array containing --> X-velocity, Y-velocity, and Depth at time TET
VDOT	Array containing derivative of velocities and depth for each node
VDOLD	Same as VDOT, except for the previous time step?
VDOTO	Same as VDOT, except for the previous time step?
NDRY	Array containing wet/dry status for each node (1 = wet, 2 = dry, -1 = About to become re-wet)
HEL	Array containing water surface elevation for each node
HOL	Array containing the derivative of WSELV for each node
HDET	Same as HOL, except for the previous time step?
HDOT	Array
IMAT	Array containing each elements material type assignment
NITSV	Vorticity iteration number for this record
VOR	Voticity array at current time time step
VOROLD	Voticity array at previous time time step
VORDOT	Time derivative of vorticity at current time step

Records 7-8 (through the *Last record, in records of two*)

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

RMA2 Binary Vorticity File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 221.

Records 5 (vorticity arrays)

TETO, NQUAL, NP, NE, (VOR(k), k=1,NP), (IMAT(k),k=1,NE)

where

TETO	Simulation time, in decimal hours
NQUAL	Number of constituents transported (NQUAL=1 for vorticity)
NP	Number of nodes in the mesh
NE	Number of elements in the mesh
VOR	Array containing vorticity results at each node
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.

Note: The vorticity binary solution file is primarily used by developers. The file format mimics the RMA4 solution file format to permit easy visulalization by the SMS graphical user interface.

RMA2 Binary Wave File Format

Records 1-4 are the same as the header records in "TABS Binary File Header Format" on page 221.

Records 5 (vorticity arrays)

TETO, NQUAL, NPPW2, (WAVEHIT(k), k=1,NPPW2),

(WAVEDIR(k), k=1,NPPW2),(WAVEPER(k), k=1,NPPW2),

NEPW2_VOID, (IMAT_VOID(k),k=1,NE)

where

WAVE_HR	Wave time, in decimal hours
NVALUE	Number of nodal arrays to follow (NVALUE=3 for wave)
NPPW2	Number of nodes represented in the wave file
WAVEHIT	Array containing wave height results at each node
WAVEDIR	Array containing wave direction results at each node
WAVEPER	Array containing wave period results at each node
NEPW2_void	Number of elements represented (NEPW2=0 for wave)
IMAT_void	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

Note: The wave binary file is primarily used by advanced users. The file format mimics the RMA4 solution file format to permit easy visualization by the SMS graphical user interface. SMS will label the contents as constituents 1-3, but they are actually the wave height, direction, period, respectively.

RMA2 Alternate Dynamic BC File Format

An alternate file containing dynamic boundary information can be used in conjunction with the primary run control file (bc file). This file is useful when the dynamic boundary conditions do not change between several RMA2 runs. All steady state boundary data is supplied in the primary run control file.

To use an alternate file, be sure to

1. Active the alternate boundary condition file ($L card, IBUP)
2. Do not put a STOP card in the primary run control file. The last card in the primary run control file should be an END card, not a blank line.
3. Create an alternate dynamic boundary condition file, which may be in card style format or *old style* format (described below)

The *old* style fixed format file (prior to version 4.28) is composed of line types A through I for each dynamic time step. The new style follows the same rules as the dynamic section of the primary run control file (bc file).

Note: The use of an *old style* alternate dynamic boundary condition file is available but not supported. The card format style is supported, but is awkward to use in SMS.

Old Style Alt-BC File Format (Not Supported)

The *old* style fixed format file is composed of line types A through I for each dynamic time step.

Line type A

1 line

Integer values end in columns 5, 10, 20, 25, 30, 35, 40, 45, 50.

NBX NSID DELT IQGEN IHGEN ISTGEN NCFLW IDEN IWIND

Line type B

IDEN lines, skip if IDEN = 0

Values end in columns 10 and 20.

NODE RON(NODE)

Line type C

NBX lines, skip if NBX = 0.

Values end in columns 10, 20, 30, 40, 50.

NODE NFIX(NODE) SPEC(NODE, 1) SPEC(NODE, 2) SPEC(NODE, 3)

Line type D

NSID lines, skip if NSID = 0.

Values end in columns 10 and 20.

IELEM   SIDF(IELEM)

Line type E

IQGEN lines, skip if IQGEN = 0.

Values end in columns 10, 20, and 30.

ICONT   QTOT   THETA

Line type F

IHGEN lines, skip if IHGEN = 0.

Values end in columns 10 and 20.

ICONT   HFX

Line type G

ISTGEN lines, skip if ISTGEN = 0.

Values end in columns 10, 20, 30, 40, 50, and 60.

ICONT   AC1X   AC2X   AC3X   AC4X   THETA

Line type H

If IWIND = +, then provide NP lines, if IWIND = -, then provide 1 line, skip if IWIND = 0.

NODE   TWX   TAX

Line type I

NCFLW lines updating earlier controller data, skip if NCFLW = 0.

Values end in columns 10, 20, 30, ..., 70.

NJN   NJTI   FLZ3   FLZ4   FLZ5   FLZ6   FLZ7

Card Style Alt-BC File Format

The example provided below, should illustrate the alternate boundary condition file concept. The example is a RMA2 simulation for a tidal inlet with a constant discharge inflow.

Example:

	T1 Primary RMA2 Run control set to use an Alternate BC File
	T2 Boundary condition file created by NGrid/FastTABS/SMS
	T3 NOYO (Metric Units) PECLET=10 with min velocity=.003m
	SI 1
	CO hotn hoto geo sol **altbc** fp sp auto-par vorticity
Alternate-BC File requested	$L 0 1 60 64 **1** 3 59 33 0
	$M 1
	TR 1 -99 1 0
	TRN 23 330 547 636
	CO ... Request a hotstart to be saved every other time step
Multi-Record Hotstart	TS -1 0 -2
	GC 4 631 633 636 639
	GC 8 1 7 12 17 22 27 32 37
	CO ... Set time controls for a dynamic run
	TZ 0.500 12.00 100 0 0
	TI 3 3 0.0001 0.0001
	FT 17.00
	IC 4.88 0.20 0.250
	EV 1 4787.00 4787.00 4787.00 4787.00 0.0150
	EV 2 4787.00 4787.00 4787.00 4787.00 0.0500
	EV 3 2394.00 2394.00 2394.00 2394.00 0.0300
	PE 1 10 0.0030 1 1 1 1
	PET 2 5 0.0030 1 1 1 1
	BQL 1 28.3168 3.14 .7
	BHL 2 4.8768 1
The last card is an "END"	**END** Finish the time step for hour = 0

The Dynamic boundary condition data is in a separate file.

1st line of alt-bc file	BHL 2 4.8478
	END Simulation at time = 0.50
	BHL 2 4.7622
	END Simulation at time = 1.00
	BHL 2 4.6259
	END Simulation at time = 1.50
	BHL 2 4.4473
skip forward several time steps	...
	Note: other cards legitimate for the transient
	run control section are also permitted in an alternate-bc file.
	…
	BHL 2 4.6509
	END Simulation at time = 11.00
Simulation Stops after 12 hours	BHL 2 4.7796
	END Simulation at time = 11.50
	BHL 2 4.8564
	END Simulation at time = 12.00
	STOP

Utilities

Although it is not necessary to use utility programs to accomplish a study, there are occasions when they have been very useful.

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.

FastTABS is a proprietary product of Brigham Young University.

SMS is a proprietary product of Brigham Young University

Moving RMA2 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 RMA2 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.

Subtracting Two RMA2 Solution Files

The utility program **R2DIFF** reads two RMA2 solutions created from the same geometry and subtracts the second from the first. The newly created RMA2 “look-alike” binary solution file can then be examined to view the differences between them. A graphical user interface, such as FastTABS or SMS, provides a convenient way of viewing these differences. The areas with the most difference will be the most emphasized.

The *data calculator* option within SMS eliminates the need for a post processing differencing utility program. SMS permits a wide range of arithmetic operations to be performed to one or more data sets in memory.

Merging RMA2 Solution Files

The utility program **MERGAVG** using the “merge” option, reads two or more RMA2 solutions created from the same geometry and merges the requested hours of simulation number 1 with simulation number 2, etc. The newly created RMA2 “look-alike” binary solution file can then be conveniently used for post-processing or as input to a TABS transport numerical model. 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.

Averaging RMA2 Velocities

The utility program **MERGAVG** using the “average” option, reads one RMA2 solution and averages the velocities over the requested hours of the simulation. The newly created RMA2 “look-alike” binary solution file will be composed of only one time step with a designated hour of -99.00. It can then be conveniently used for post-processing or as input to a TABS transport numerical model.

Tip: In order to examine “net” or “residual” effects of velocity in an estuary, request the utility program to average over a tidal cycle from your simulation.

Obtaining Guideline Values Of Eddy Viscosity

The utility program **MAKE_EV_DF** will read an ASCII geometry file and will, for each material type, determine the average size of the elements composed of that material type. The user is asked for a Peclet number he wishes to maintain, and, for each material type, a table is generated containing a range of velocities and associated eddy viscosity values which should maintain the given Peclet value (if the fluid is moving **S** fast, use the value for eddy viscosity associated with **S**).

This utility program is also used to help derive the appropriate values for the diffusion coefficients required in the RMA4 water quality transport model.

Obtaining A Summary Listing After RMA2 Has Run

The utility program **R2_2_SUM** will create a summary results listing file from your solution after the RMA2 model simulation has run.

Repairing a RMA2 HOTSTART File

The utility program **R2_HOTFIX** will aid in fixing a faulty hotstart file after the RMA2 model simulation has run. Minor oversites, such as bathymetry errors, which make the original hotstart file unusable, may be repaired with the help of this program.

The utility has proven successful for changes that will generallyr educe the velocities. For example, deepening a channel, placing a barrier or dike will reduce the local velocities. These changes do not create hot spots and RMA2 seems to be able to handle the shock and proceed. However, changes that cause increases in the velocities may make the model too unstable to recover. In those cases, the model must be restarted from a steady-state run and the dynamic spin-up repeated.

Determining the Records Available on a RMA2 HOTSTART File

The utility program **R2_HOTFIX** may be used to determine the records that were saved to the hotstart file during a RMA2 simulation. An example follows:

Example of using R2_HOTFIX to diagnose the contents of a RMA2 hotstart file.

```
r2_hotfix.exe

  Program R2_HOTFIX, Last Modified 4 May 2000
  Author: Joe Letter, 1998,  US ERDC at WES-CHL
  Reads a multi-record HOTStart file from RMA2
  Repairs and Recreates the HOTStart
        based upon a new geometry mandate

  Enter the original geometry binary file name (binary):
  It is the one used to create the broken hotstart.
     NOTE: enter "null" to only examine a hotstart file.
null
   The geometry file was voided.
   Determine all available hotstart records.

   Enter a RMA2 hotstart output file (binary):
myproject.hot
        ... Ok, Let us find out
            what records are available
            on this HOTSTART file

 Begin to read the HOTSTART file (Sub ... HOTREAD)
 Seeking to find the record corresponding to:
    Simulation Time     = 9876.000 if -999 -> get 1st hour
    UVH_Iteration       =       99 if    0 -> get 1st iter
    Vorticity_Iteration  =       99 if    0 -> get 1st iter

 HOTSTART Rec # 1  Time(hr)= 7.00 UVH_Iter= 5 VOR_Iter= 0  Node/Ele= 640  200
 HOTSTART Rec # 2  Time(hr)= 7.50 UVH_Iter= 3 VOR_Iter= 0  Node/Ele= 640  200
 HOTSTART Rec # 3  Time(hr)= 8.50 UVH_Iter= 3 VOR_Iter= 0  Node/Ele= 640  200
 HOTSTART Rec # 4  Time(hr)= 9.50 UVH_Iter= 3 VOR_Iter= 0  Node/Ele= 640  200
```

```
HOTSTART Rec # 5  Time(hr)= 10.5 UVH_Iter= 2 VOR_Iter= 0  Node/Ele= 640  200
HOTSTART Rec # 6  Time(hr)= 11.5 UVH_Iter= 2 VOR_Iter= 0  Node/Ele= 640  200

→ EOF on hotstart input
```

Exterior Curving and Eliminating Bad Boundary Break Angles

The utility program **SLOPEFIX** will aid in curving all exterior element edges. The program will not curve edges involved with boundary condition assignments. It cycles through the entire mesh multiple times in an attempt to best fit the curves to eliminate bad boundary break angles. Joe Letter of ERDC at WES wrote this utility program.

The SLOPEFIX program is not able handle every possible condition successfully. The user should inspect the shoreline of the mesh after performing a slope-fix. On occasion, a local "twist" will occur that will need to be adjusted manually using a graphical interface such as SMS.

PROGRAM SLOPEFIX Version 6a, Created Dec 1998, Last Mod: May 2000

Enter the GFGEN ascii input geometry file
tweed2gate.geo

Enter file name for new geometry created by SLOPEFIX
tweed2gate.geo_sf

Enter the file containing the GC-Cards that represent the open water boundaries.
These GC-lines will not be slope fixed.
tweed2gate.bc

Enter the maximum acceptable shoreline angle break ...
before trying to move the midside node. Input degrees (1 degree recommended):
1

Enter the percentage of nodal movement relative to the elemental dimensions.
Input percent (40 percent recommended)
40

The program iterates 6 times before saving a “slope fixed” geometry file.

Repairing a mesh with 1D Width and Off-Channel Parameters

The utility program **GET_1D** will repair a mesh that has lost the 1D width and off-channel parameters. The criterion to run this program is an older version of the mesh that had the 1D section correct. Joe Letter of ERDC at WES wrote this utility program.

This program was written at a time when all of the 1D cross section informationwas included in the BC file. Many users would forget to read in the BC file prior to renumbering the mesh. This utility program was a means of recovering that lost connection.

Technical Support

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.

On-line Support

Technical support is available on the Internet 24 hours a day, 7 days a week.

Option 1: World Wide Web

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

Option 2: E-mail

If you cannot find a solution to your problem yourself, you can send us E-mail at **tabs@hl.wes.army.mil**. We will try to obtain a solution to your problem and reply with our findings.

TABS Hotline

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:

- The GFGEN Geometry file
- The RMA2 run control file (.rc2 file)
- Computer information: type, memory capacity, etc.
- Any error messages
- Your E-mail and FTP addresses if applicable

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

REFERENCES

Bernard, R. S., and Schneider, M. L (1992) .“Depth-Averaged Numerical Modeling for Curved Channels”. Technical report HL-92-9. US Army Engineers Waterways Experiment Station, Vicksburg, MS.

Norton, W. R., King, I. P. , "Operating Instructions for the Computer Program RMA-2V", February 1997, Resource Management Associates, Lafayette, CA

Donnell, Barbara P., Letter, Joseph V., McAnally, W. H., Roig, Lisa C., and others, "Users Guide for RMA2 Version 4.3", August 3,1998, http://chl.wes.army.mil/software/tabs/docs.htp

Finnie, J., Donnell, B. P., Letter, J. V., Bernard, R. S., "A Secondary Flow Correction for Depth-Averaged Flow Calculations",Proceeding of the 11th Conference Engineering Mechanics Division, ASCE, Held May 19-22, 1996, Fort Lauderdale, FL

Finnie, John I., Donnell, Barbara P. , Letter, Joseph V., and Bernard, Robert S., “Secondary Flow Correction for Depth-Averaged Flow Calculations”, Journal of Engineering Mechanics, Vol 125, No. 7, page 848-863, July 1999.

Roig, Lisa C., (1995) "Mathematical Theory and Numerical Methods for the Modeling of Wetland Hydraulics", in Water Resources Engineering, Proceedings of the 1995 First International Conference San Antonio, Texas, August 14 - 18, 1995, ASCE, New York, pp. 249-253.

Smagorinski, J., "General Circulation Experiments with the Primitive Equations", Monthly Weather Review, Vol. 93, 1963, pp. 99-165

Speziale, C. G., "Turbulence Modeling for Time Dependent RANS and VLES: A Review", AIAA Journal, Vol. 36, No. 2, February 1998

Thomas, W. A., and McAnally, W. H., 1991(Revised), "User's Manual for the Generalized Computer Program System, Open-Channel Flow and Sedimentation, TABS-MD," Instructional Report HL-85-1, USAE Waterways Experiment Station, Vicksburg, MS. Original publication date: July 1985.

Thomas, T. G., Williams, J. J. R., "Large Eddy Simulation of a Symmetric Trapezoidal Channel at a Reynolds Number of 430,000", Journal of Hydraulic Research, Vol. 33, No. 6, 1995.

Users Guide to GFGEN Version 4.27, US Army Corps of Engineers Waterways Experiment Station - Hydraulics Laboratory, August 1997

Users Guide to RMA4 WES Version 4.5, US Army, Engineer Research and Development Center, Waterways Experiment Station, Coastal and Hydraulics Laboratory, July 2000, Draft

Users Guide to SED2D WES Version 4.5, US Army, Engineer Research and Development Center, Waterways Experiment Station, Coastal and Hydraulics Laboratory, July 2000, Draft

----

Brigham Young University, 1994, "FastTABS 3.0 Hydrodynamic Modeling Reference Manual," Engineering Graphics Laboratory, Provo, Utah.

Brigham Young University, 1995, "SMS Hydrodynamic Modeling Reference Manual," Engineering Graphics Laboratory, Provo, Utah.

SMS 6.0, Reference Manual, Environmental Modeling Research Laboratory, Brigham Young University, January 28, 1999

SMS 6.0, Tutorial, Environmental Modeling Research Laboratory, Brigham Young University, October 27, 1998

SMS 7.0, Tutorials, Environmental Modeling Research Laboratory, Brigham Young University, June 1, 2000

Glossary of Terms

Alternate boundary condition file

(Supplemental run control file)

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

Anemometer

A device used to measure wind speed and direction.

See Also:

BWC card.

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.

See Also:"Maintain Good Elemental Properties" on page 22

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.

See Also: "Verifying The " on page 53.

Batch mode

The opposite of Interactive mode. 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.

See Also: "Running In Batch Mode" on page 19.

Bathymetry

Measured depths in water bodies.

Binary

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

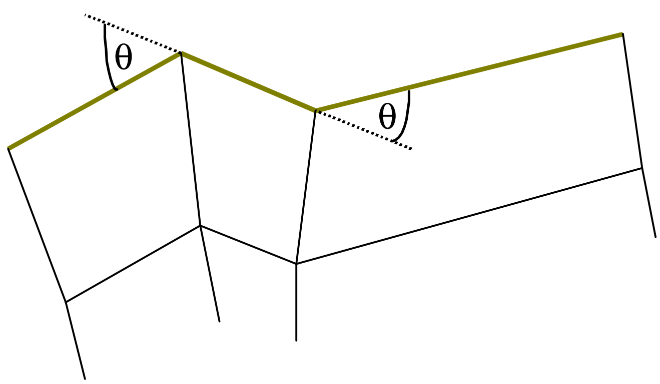
The TABS system uses some binary files. A binary file permits an efficient means to store numerical results.

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 unformatted FORTRAN WRITE statement.

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 tailwater elevation and incoming upstream discharge are typical boundary conditions.

See Also: "Specifying Boundary Conditions" on page 38

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.

See Also: "Specifying Boundary Conditions" on page 38

Boundary node

Any node which lies along an *exterior element* edge, or demarcates the wet/dry interface.

Buffer blocks

Temporary files created by RMA2, 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 RMA2 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.

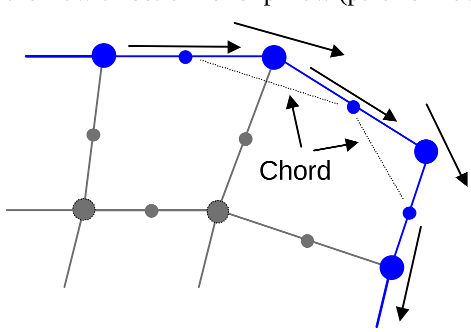
The TABS programs use cards in the same way. The difference is that the card data are stored in a disk file and not in a filing cabinet.

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, training, and support for the TABS modeling system, contact the ERDC at WES Coastal and Hydraulics Laboratory via e-mail at **tabs@hl.wes.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.

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 used herein is a check of the water mass flux at locations for use in a test of the satisfaction of the continuity equation. The continuity check lines are typically used to estimate the flow rates and serve as an error indicator.

Continuity line

(or Continuity Check 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 2-node element with an IMAT $\geq$ 904. The first node in the element connection table should be the side of the structure with the typical higher elevation.

See Also: "Control Structure Elements" on page 9

Convergence

The process of obtaining a solution by way of an iterative solution technique, such as the Newton-Raphson method.

See Also: Diverge, and Iteration

Convergence parameters

Related to the three RMA2 independent variables: *x* and *y* velocity components, and depth. The difference between previous and present values should approach zero with each successive iteration. If they do not, the solution is said to diverge.

For information on setting the convergence criterion, see "Providing A Convergence Criterion" on page 34.

Coriolis

Coriolis effect [for Gaspard Coriolis], is the tendency for any moving body on or above the earth's surface to drift sideways from its course because of the earth's rotational direction (west to east) and speed, which is greater for a surface point near the equator than toward the poles. In the Northern Hemisphere the drift is to the right of the motion; in the Southern Hemisphere, to the left. In most human-operated vehicles, continuous course adjustments mask the Coriolis effect. The Coriolis effect must be considered, however, when plotting ocean currents and wind patterns as well as trajectories of free-moving projectiles through air or water.

See Also: “Compensating For The Rotation Of The Earth” on page 111.

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 can help conserve mass in the transport models.

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.

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 by the iteration technique.

See Also: "What Is An Iteration?" on page 32

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.

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 Exx (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 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.

Exponential underflow or overflow

An attempt by the computer to calculate a real number outside the recision of the machine.

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

This was the predecessor to SMS. 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 (WES) and Brigham Young University (BYU).

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.

Floating point exception

Usually implies that a negative number is being raised to some expontial power. This computation error results from the use of logarithms, which are not defined for numbers less than or equal to zero.

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 0.6.

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 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.

See Also: "Hotstart File" on page 26 and "Resuming A Stopped Simulation" on page 80

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.

Hydraulic radius

The hydraulic radius, R, is the ratio of the cross sectional area to the wetted perimeter. R = A/P.

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

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.

See also: "What Is An Iteration?" on page 32, and "How RMA2 Finds A Solution" on page 32.

Junction element

A special element, consisting of 3 to 8 nodes, 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 pass filter

This is one form of a digital filter. It eliminates noise at low frequencies. For example, an ideal low pass filter would be a square wave in the frequency domain, with a value of 1 for frequencies less than 1/35 cph and zero for frequencies greater than 1/36 cph. This would allow oscillations of periods longer than 36-hrs to pass while those shorter than 36-hrs are removed from the record. A cutoff period of 36-hours was selected to remove tide effects from the record.

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 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.

Null point

A location in a network where there is no net fluid transport (no flow).

Off-channel storage

A one-dimensional element feature. The storage width associated with the node at zero depth, as specified on GN or 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.

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

Defines the relationship between element properties, velocity, and eddy viscosity for the Peclet Method of assigning automatic turbulence.

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

See Also: "Modeling Turbulence" on page 46, PE card.

PREHYD

The subroutine within RMA2 that reads the TABS formatted run control file..

Proclv

The user friendly interactive procedure on mainframe WES Cray computers which permits users to access the library versions of the TABS system. Proclv is no longer supported.

Progressive wave

A wave that moves relative to a fixed coordinate system in a fluid. The direction in which it moves is termed the direction of wave propagation.

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.

Reflected wave

That part of an incident wave that is returned seaward when a wave impinges on a steep beach, barrier, or other reflecting surface.

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.

Ringing convergence

If the convergence statistics show a maxium change after several iterations at the same node for each successive iteration with approximately the same magnitude, alternating in sign, then "*ringing*" is occuring.

Rip tides

A current of water disturbed by an opposing current. A strong, narrow current that flows rapidly away from the shore, returning water carried landward by waves.

Rip-rap

Stones, chunks of concrete, or other debris on an embankment slope, or stream side slope, generally used to prevent erosion.

RMA10

Formally named RMA10. TABS-MDS 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 US Army ERDC-WES-CHL version is based upon the work of Dr. Ian King of Resource Management Associates. Many enhancements have been made to the WES version. Dr Charlie R. Berger is the ERDC at WES point of contact.

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.

Roughness

In a river or stream bed, the material on the side slopes or the bottom, such as stones, etc., which inhibit the flow.

See Also: "Bed Friction And Resistance To Flow" on page 45.

Run control file

An input data file which provides parameters that control the RMA2 simulation run.

SED2D

Originally known as STUDH, SED2D is a two-dimensional depth averaged sediment transport Finite Element numerical model within TABS. Joe Letter is the WES point of contact for SED2D theory.

Semidiurnal tide

A tide with two high and two low waters in a tidal day with comparatively little diurnal tide inequality.

Ship simulator (WES)

The Waterways Experiment Station Navigation Simulator started out in 1984 as a single ship simulator with one view screen, a radar screen, a ship console, and a tow console running on a PDP-11 computer. The visual scene at that time was comparable to early computer games in color and detail. In the twelve years since then, the view screen has changed to three large screen monitors for a total viewing angle of 140 degrees. A second simulator has been added along with the capability of two-way traffic.

In the two-way traffic mode of operation, the two pilots interact vocally through an intercom, visually through the view screens, and physically through the hydrodynamic model controlling the reaction of the ship due to currents, wind, channel geometry, and traffic interaction.

In 1995 a new facility was built within the Hydraulics Laboratory at WES to house the two simulators and the new computer system that will run both simultaneously. The most recent addition to the Navigation Simulator was visual scene software incorporating hardware texturing and many other features to give the visual scene a more “realistic” look.

Smagorinski Coefficient

Defines the relationship between element properties, gradient of velocity, and eddy viscosity for the Smagorinski Method of assigning automatic turbulence

SMS

SMS officially replaced *FastTABS* in 1996. 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. Corps of Engineers employees may contact the Waterways Experiment Station for more information via e-mail at **tabs@hl.wes.army.mil**, or call (601) 634-3339.

Source code

The US Army Corps of Engineers, Waterways Experiment Station, Hydraulics Laboratory, 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 Coastal and Hydraulics Laboratory via e-mail at **tabs@hl.wes.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 gravity wave to propagate across the longest axis of the mesh and return.

The speed of the gravity wave is calculated as

$$S = \sqrt{gh}$$

where

S = wave speed
g = gravity
h = representative 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 US Army ERDC-WES-CHL.

STWAVE

A steady state, irregular wave model maintained by US Army ERDC-WES-CHL. Dr. Jane Smith is the point of contact.

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. Maintained by US Army ERDC-WES-CHL

Tailwater

The water surface elevation at the exit boundary.

Thalweg

The line representing the deepest part of a river or channel.

Tidal bore

A high and often dangerous wave caused by the surge of a flood tide upstream in a narrowing estuary or by colliding tidal currents.

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.

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. These adjustments are made interactively until the model agrees satisfactorily with field (prototype) observations.

Vertically homogeneous fluid

Refers to the similarity of the fluid within the water column.

Viscosity

The degree to which a fluid resists flow under an applied force.

Vorticity

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.

See Also: "Bendway Correction (Vorticity)" on page 93

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.

Geo-Structures Laboratory (Official new Lab title not finalized)

> 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).

Well-confined channel

A channel in which the water cannot escape the banks.

Wetted perimeter

The wetted perimeter, P, is the length of wetted cross section normal to the direction of flow.

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

Notes