UPDATE DOCUMENTATION

RMA2 - A TWO DIMENSIONAL
FINITE ELEMENT MODEL
FOR FLOW IN ESTUARIES AND STREAMS

Version 6.4

by

Ian P. King

Dept of Civil Engineering
University of California
Davis, Ca

June 1997

1. INTRODUCTION

This update to the RMA-2 documentation is designed to the provide the user with information relevant to the latest version of RMA-2. Section 2 presents the current status of the model including changes since the last full documentation that described version **4.0.** Section 3 describes the run time instructions, section **4,** the data input formats required to execute this latest version and section 5 provides guidelines for execution of the model. Appendices present:

(A) the governing equations and finite element derivatives used in the model.
(B) the formulation of the Smagorinksky closure
(C) a description of the marsh porosity formulation as applied in RMA-2

2. CURRENT MODEL STATUS

2.1 Update history from Version 4.0:

Version 6.4 June 1997

Changes to the model include:
(a) optional addition of greater roughness when the water surface is in the porous zone when marsh elements are activated (see section 2.2.9).
(b) changes to the output file structure to create a single file with all the convergence information for each iteration.
(c) improved computation of elevation changes as elements dry into the porous zone.
(d) optional input of under-relaxation factors for each time step (see line type DT input instructions.
(e) addition of options to reset the clock time when restarting from a file.
(9) correction of format errors in the wind routines,
(g) addition of a set of options to control output of the binary results file and of extra restart files.

*Version 6.3 January - **April** 1997*

This version incorporates a number of additional capabilities designed to improve the utility of RMA-2. Changes include interpolation of wind fields, new flow boundary conditions, additional hydraulic structure elements, revisions to wetting and drying options and a more convenient method for computing initial conditions for sloping bed environments. Additional file names have been added for compatibility between compilers.

*Version **6.2** May. 1996*

Version 6.2 incorporates the addition of the Smagorinsky (1963) turbulence closure as an option in the model. Appendix B presents the formulation used.

Version 6.1 Feb. 1996

Version 6.1 represents a significant change to the input structure. The objective is to date stamp all hydrograph input files and to permit execution from a given date starting time. Changes have been made in numerous routines to archive this end. The main changes are in GETH, GETQ, GETEQ, GTIDE, SBGEN and OUTPUT. A second change has been to move all of the response files, except the main input file, into a new labeled format list at the head of the input data file. This is designed both to simplify input and to provide a record of what files were used for a given input set. Finally, when a time step is input the user may now set a time until which the accompanying set of boundary condition data will be applied. This allows time step size to altered at specific times. Note that the various graph files are still interpolated.

Version 6.0(D)   Jan. 1996

Version 6.0(d) modifies SBGEN to permit input of hydrographs for the element inflow values. GETEQ has been created to carry out this operation. An additional input option on the EFE and EFA triggers this operation.

*Version 6.0(C)   **Nov.** 1995*

Version 6.0(C) Modifies a number of routines to eliminate IABS. FORTRAN 90 does not allow this form for INTEGER*2. Also eliminate blank COMMON from INPUT and COEFI. Correction to COEFI to update method by which the surface width is calculated for cases with side slope

Version 6.0(B)   Oct. 1995

This version fixes bug in OUTPUT associated with line specification of velocity distribution along elevation boundary. Also fixed is typo in GETQ that leads to errors when reading hydrographs.
Fix error in BLINE associated with 2-d control structures.

Version 6.0(A)   May 1995

This revision fixes a bug in output (a limiting dimension of TT in routine OUTPUT) when running many time steps. In the process a test has been added in input for potential violation of the new parameter. The new parameter has been placed in BLKIA.COM

'Version 6.0   March 1995

This revision introduces a totally revised set of input formats for the main input file of RMA-2. These changes have been designed to smooth introduction of graphic data editor RMA2PR which is now available. RMA2PR is a FORTRAN based program that will present to the user each line of the input data to RMA-2 together with descriptive information. An on-line help screen is also available. This process will largely eliminate the need to look in the manual for formats of individual lines.
All lines of the main and alternate input files now have a character identifier in the first 2 or 3 columns and the data is divided into fields of ***8.***

'Version 5.1   Sept 1994

This revision introduces the ability to dropout equations during the solution process. Decisions to drop out are based on convergence to a tighter tolerance than the basic convergence. Additional data on line D1 define that capability. A bug associated with flow angles has been fixed for two-dimensional control structures.

'Version 5.0   Oct 1993

This new version introduces the capability to use 9 node Lagrangian basis function quadrilaterals. These elements reputedly perform "better" that the current **8** node isoparametric elements. The geometric input is unchanged, the ninth node is internally generated for all quadrilaterals by a switch in the input data file.

'Version 4.5   Oct 1993

Two features have been added to RMA-2. Control structure elements have been expanded to include two-dimensional elements. **Section 2.2.1** discusses their implementation. Data input of hydrographs and tidal boundary conditions have been made much more flexible. **Section 2.2.2** discusses these changes.

*Version 4.4 **May** 1993*

Bank friction has been added to the model. Input is through a change to the element property lines. The print options have been modified so that when IPRT = 2 all nodal, element and initial condition data is printed. Treatment of input geometry files has been modified to allow the model to recognize the type in input file at the first prompt.

Version 4.3g March 1993

A modification to the wetting and drying analysis was made to adjust for the marsh depth shift and range parameters. Reading of geometry files was made more flexible. It is no longer required that the alternate geometry file entry be used when widths and side slopes are present.

Version 4.3f Dec 1992

The model was corrected for a bug in the special case when input boundary flows are exactly zero. The angle of flow for the distributed velocity boundary condition was also modified to eliminate a bug.

'Version 4.3e June 1992'

Model updated to improve unit consistency in metric version when using wetting and drying option and control structures. Error corrected in distributed velocity boundary condition introduced with version 4.3d. The error was a special case when velocities were nearly in the y-direction.

'Version 4.3d Feb 1992'

Model updated to permit input of distributed velocities (i.e. scale factor values) along a line where one elevation is specified

'Version 4.3c Nov 1991'

Wind algorithms updated to reflect metric options.

'Version 4.3b Feb 1991'

Error in SUBROUTINE INPUT associated with automatic computation of eddy coefficients corrected problem only appeared when not printing input data, also change in output to ensure saving of alternate print file at each time step.

'Version 4.3a Dec 1990'

Updated to allow alternate geometric file input. This is the alternate file format that can be generated by RMA-1. It contains all the channel width data.

'Version 4.3 Nov 1990'

Updated to insert convergence testing and to permit metric units and to permit automatic scaling of turbulent eddy coefficients.

'Version 4.2d   May 1990'

Updated to improve convergence of marsh elements and modify the wetting and drying algorithm.

'Version 4.2c   March 1990'

Updated to fix bug in check for continuity lines and increase limit on nodes for continuity lines.

'Version 4.2b   February 1990'

Updated to include check on renumbering scheme and fix a minor bug.

'Version 4.2a   August 1989'

Updated to include experimental version of computational algorithm for eddy viscosity coefficients (undocumented and not fully tested. Use recommended only after consultation. Other minor bugs and inconveniences fixed.

'Version 4.2   December 1988'

Updated to include more computer options (for file manipulation) and different scratch file names. Modified wind options and input installed.

'Version 4.1   October 1988'

Updated to include variable density.

*'Version 4.0   **August** 1988'*

Baseline version for full documentation.

2.2   *Recent Program Updates*

2.2.1   *Implementation of two-dimensional control structures*

Two-dimensional control structures are implemented by creation of special **8** node elements with type numbers in excess of 903. The type number is later used to define the properties of the control structure. The are special requirements of these elements. The upstream side of the structure must be defined with sequence number (1), (2), and (3) of the element numbering sequence. The downstream side must be defined with sequence numbers (5), (6) and **(7).** Sequence numbers **(4)** and **(8)** must be left zero. The entire sequence must be numbered in a counterclockwise sequence. Note that the element does not have to have a dimension. So that, for example, sequence number (1) and sequence number **(7)**can have the same location.

2.2.2   *Implementation of alternate boundary condition file input*

The model has now been structured to allow the user to define alternate files that contain (1) hydrographs relating flow with time, **(2)** tidal-graphs that relate elevation with time and (3) tidal harmonics for computation of tidal elevations. The program automatically prompts for and uses these files when code values are input on the steady state boundary condition data lines. The exact file structure for each of these options is presented following the main input file data instructions.

When this type of input is used the user may use the alternate boundary condition file to define the controls for each time step. It should be remembered that this file will be rewound and reread when an end of file is encountered. Thus a user can put one set of controls in this file and have it reread repeatedly.

Note that when file interpolation is defined the user should not enter any data after the initial steady state values for that continuity line. Other continuity lines can be entered if desired.

2.2.3   *Implementation of equation dropout feature*

The model has been modified to allow the user to define a second tighter level of convergence. When a node passes this test then it is a candidate for elimination from the next iteration. A transition region is created around areas that pass the test and only nodes that are three elements away from the boundary between the converged and still active nodes are dropped for the next iteration.

2.2.4 *Addition of interpolation of wind field data to nodal values*

An option has been added to the model that permits input of time histories of wind velocity data at a number of discrete stations which are then interpolated to all nodes of the system for each time step.

2.2.5 *Computation of initial conditions for sloping environments*

When modeling river basins, it is often very difficult to develop initial steady flow conditions given a flat initial water surface. The conventional method has been to carry out a number of steady state solutions gradually lowering the water surface. This can be very time consuming when flows are shallow and marsh porosity comes into play. The new option of the model allows the user to apply a tilt to the entire basin to bring the net slope to zero and then gradually reduce a factor applied to this correction. This method should be capable of faster development of appropriate initial conditions. Note that the tilt must be applied as a planar tilt about a fixed point, and that users must be careful to ensure that the downstream boundary is only minimally affected by the tilt.

2.2.6 *Modification to element dropout options*

An additional option has been provided that allows the user to specify which element type numbers are candidates for dropout when the marsh porosity option is active. This will allow the user to prevent dropout of elements in shallow main channels but still use the dropout feature.

2.2.7 Addition of new inflow boundary condition

Until this version boundary inflows were required to be setup so that flow was normal to boundary along the line, but parallel to boundary at the end points. This required a nozzle like mesh design (see section 5.9.2). The new option permits inflow to cross a limited number of nodes across a straight boundary. Section 5.9.3 provides details of the revised option.

2.2.8 Additional one and two-dimensional structures incorporated

Two additional types of flow control structure have been incorporated. One type acts as a weir, with no flow when water levels are below the crest, overflow when one water level is above the crest and submerged weir flow when both water levels are above the crest.

2.2.9 Computation of additional friction as elements dry.

It has long been noted that velocities have been too high in the very shallow zone created as the water surface elevation falls below the bed level for the marsh porosity option. With version 6.4 a new option has been added. A scaling factor is applied to the effective Manning or Chezy coefficient.

When the water surface *is* at or below the level at which the minimum porosity is active, the factor is computed using the following formula

$$F_c = 1.0 + [F_{max} - 1.0][1. - \frac{(W_{sel} - B_{el})}{S_{dep}}]$$

where:

F_c	=	Factor applied to the friction coefficient
F_{max}	=	Input adjustment factor (note that if the input factor, F = 0.0 then F_{max} = 1.0)
S_{dep}	=	Overall depth of sub bed (depth shift)
W_{sel}	=	Water surface elevation
B_{el}	=	Elevation at the base of the sub-bed

2.3   *Program Structure*

The list below presents a brief description of the function of each subroutine.

MAIN   Main program that calls all the primary routines, governs the overall flow and writes restart and save files.

INPUT   Principal subroutine controlling inflow of data.

BFORM   The subroutine (called from INPUT) that installs the input boundary conditions into the appropriate arrays.

SBGEN   Subroutine that controls generation of boundary conditions from convenience input values.

HGEN   Subroutine that generates fixed elevation boundary conditions.

QGEN   Subroutine for generation of boundary conditions for specified flow across a line.

STGEN	Subroutine to generate a given stage-flow boundary condition.
AGEN	Subroutine to generate cross sectional area of a continuity line at a given water stage.
LOAD	Subroutine for generation of equivalence between node number and degree of freedom from equation number. Also establishes lists indicating compilation of equation formulation.
FRONT	Subroutine for solution of simultaneous equations.
COEFS	Subroutine for generation of element coefficient matrices for two dimensional elements.
XRED,XWRT	Subroutines for reading and writing of blocks of data at intermediate stages of the equation solution process.
CHECK	Subroutine for computation of continuity checks at specified sections.
COEFI	Subroutine for generation of element coefficient matrices for one dimensional elements
CSTRC	Subroutine for generation of element coefficient matrices for control structure elements
CSTRC2D	Subroutine for generation of element coefficient matrices for two-dimensional control structure elements
AMF	Subroutine called evaluate parameters under the marsh option.
ANGLEN	Subroutine for computation of eddy viscosities as functions of element dimensions.
BDATA	Subroutine called to initialize data values.
BLINE	Subroutine to compute angles and parallel flow boundary conditions.
DEL	Subroutine for establishment of network in the dry node configuration, generating nodes as required
DELM	Subroutine for establishment of network, under the marsh element option, dropping elements as required
FILE	Subroutine that is called to interactively open and close required files
FLDIR	Subroutine to establish directions of flow for one dimensional elements.
GETQ	Subroutine to read and interpolate hydrograph file as required to form boundary conditions.
GETEQ	Subroutine to read and interpolate hydrograph file of element inflows as required..
GETH	Subroutine to read and interpolate tidal graph file as required to form boundary conditions.
GTIDE	Subroutine to read and interpolate harmonic components from a file and to compute tidal stage as required to form boundary conditions

OUTPUT   Subroutine for printing of latest velocities and depths.

QNORM   Subroutine to compute normal flow boundary conditions.

REPLY   Subroutine used to decode user responses

REWET   Subroutine for restoration of original grid with apparent negative depths, using the dry element option.

REWETM   Subroutine for restoration of original grid, under the march element option, with apparent negative depths to check whether an area has re-flooded

SB   Subroutine called to store values of the basis functions at the Gauss integration points.

WINDF   Subroutine called when wind stresses are used in the model.

WINDSCLE   Subroutine used to read data from a file and interpolate wind velocity and direction data from values from several stations.

ZVRS   Subroutine used to set identification parameter for computer type. This allows user to define methods used for scratch file utilization.

XN,DNX,DNY Double precision functions for evaluation of basis functions.

2.4   *Model Limitations*

The program is written in standard FORTRAN77 and has been executed on IBM-PC's under DOS, PRIME 550 and Vax 780 computers with virtual memory operating systems, a MAC-2 using MAC-FORTRAN and on UNIX systems such as SUN, SGI and DEC. It is possible that in particular applications the size limitations listed below will have to be adjusted by recompilation with changed dimensions to reflect different size networks. All the limits listed below except fluid types are maintained in PARAMETER statements.

It has been found that differing computer systems use different OPEN statement and have limitations on file sizes and numbers of files. SUBROUTINE **ZVRS** has been created with the variable IVRSID designed to be set by the user with the integer appropriate for his system. The default number supplied is IVRSID=4 which will create a sequential file for each time the buffer in the equation solving routine is filled. This setting checks and generates file names that have the structure Hxxx.001 etc.

The latest version of RMA-2 is designated as release 6.0. Descriptions of recent updates to the code are includes as comments at the beginning of the FORTRAN code. **In** this version it is presently compiled to allow:

6,000	max. nodes
3,000	max. elements
12,000	max. number of equations
225	max. front width
250,000	max. buffer size

The above dimensions may be adjusted provided sufficient memory space is available.

The following file units are required in execution of the model.

2/ Input data
3/ Output results
9/ Scratch unit for data during equation solution (not required if buffer size is large
    enough

Units (60) (61) (62) (63) (64) and (59) may be required if the options that permit geometric file input, alternate input file for boundary conditions, saving of a restart file, using a previously saved restart file, saving a binary results file and generation of an alternate output file are executed. When boundary conditions are input as hydrograph, tidalgraph, tidal harmonic, or element inflow hydrograph files (11) (12) (13) or (14) may also be used. File (75) may also be written as "messrm2.dat" giving information on inflow hydrographs, element flow hydrographs, tidal graphs and tidal harmonics and errors.

3.   RUN TIME INSTRUCTIONS

The model operates interactively with the user requesting the file name for data input. All other files are now (post Version 6.0) named in the data input file depending on the functions desired by the user. The basic input data is entered on this interactively named file and is described in section **4.**

The following lists and describes files that may be required:

a) Input file name.

b) Output file name. This file is used for tabulation of all simulation results. From version **6.4** the output file name before any suffix is used to name a file with suffix "**.itr**" that is used for storage of convergence data from all iterations

c) Input geometry file name (from **RMA-1** or RMAGEN). Note that the alternate geometry file is not required at version 4.3(f). It can be input as a regular geometry file

d) Boundary condition input file name (used for boundary conditions. File is read when regular input file is at end of file and may be reread for repeating boundary input.

e) Input restart file name, for restarting from a previous run.

9 Output restart file name for saving results with a view to restarting.

g) Output save file name for storing all velocities and depths computed during each solution step. This is a binary file that may be used for post processing.

h) Alternate print file for saving of results at selected nodes. This file name is only required if the input data sets the appropriate switch.

i) File containing element inflow hydrograph data for interpolation.

j) File containing boundary hydrograph data for interpolation.

k) File containing tidal-graph data for interpolation.

l) File containing tidal harmonic data for later computation.

m) File containing wind data for later interpolation.

4.   DATA INPUT

This chapter presents the data format instructions for assemblage of the input files required to execute RMA-2. The model is designed to operate in English and metric units, units are selected based on the value IGRAV input on line type D. In these instructions actual units are only given where they are needed for clarity and for the wind stress input where they must conform. The optional alternate input data file may start with any new time step of the boundary condition data. The sample problems represent two typical problems that are available with the program files.

4.1   Main Input file formats

The data input file has the following structure:

The list below contains the file names that are to be used in any given simulation. If any given file type is not required it may be omitted from the list. The only required file is the OUTFIL line. Note that the filenames may be presented in any order. The order listed here is not significant. The list of files should be terminated with an **ENDFIL** data line.

Line type **OUTFIL OUTPUT FILE NAME**   1 line (required)

01-08	ID	A	"OUTFIL "
09-40	FNAM	A	Name of ASCII file that will contain the results

Line type **OUTSPL SELECTED NODE OUTPUT FILE NAME**   1 line (rewired)

01-08	ID	A	"OUTSPL "
09-40	FNAM	A	Name of ASCII file that will contain the results printed in hydrograph form for selected nodes.

Line type **BCFIL   BOUNDARY CONDITION FILE NAME** 1 line (optional)

01-08	ID	A	"BCFIL "
09-40	FNAM	A	Name of ASCII file that contains boundary conditions. This is an optional file that will continue when the main input file is exhausted. Note that when an end of file is reached on this file it will be rewound and reread from the start, so that repeated boundary conditions can be read from a single set on this file.

Line type **INGEO   INPUT GEOMETRY FILE NAME   1 line (optional)**

Use when the GEO file has been generated with a standard UNFORMATTED structure from a compatibly compiled version of RMAGEN or RMA-I   For example, Lahev to Lahev versions

01-08   ID   A   "INGEO "

09-40   FNAM   A   Name of unformatted input geometry (generated by RMA-1 or RMAGEN). This is an optional file. All data in this file can be input in the main data file.

Line type **INBNGEO INPUT PURE BINARY GEOMETRY FILE NAME   1 line (optional)**

Use when a TRANSPARENT or BINARY file has been created using special versions of RMAGEN or RMA-1

01-08   ID   A   "INBNGEO"

09-40   FNAM   A   Name of pure binary (no headers or trailers) input geometry (generated by RMA-1 or RMAGEN). This is an optional file. All data in this file can be input in the main data file.

Line type **INRST INPUT RESTART FILE NAME 1 line (optional)**

01-08   ID   A   "INRST "

09-40   FNAM   A   Name of binary input restart file generated by a previous RMA-2 simulation that will be used as the initial condition for this simulation.

Line type **INBNRST INPUT PURE BINARY RESTART FILE NAME   1 line (optional)**

01-08   ID   A   "INBNRST"

09-40   FNAM   A   Name of pure binary (no headers or trailers) input restart file generated by a previous RMA-2 simulation that will be used as the initial condition for this simulation.

Line type **OUTRST OUTPUT RESTART FILE NAME   1 line (optional)**

01-08   ID   A   "OUTRST "

09-40   FNAM   A   Name of binary output restart file to be generated for use by a later RMA-2 simulation.

Line type **OUTBNRST OUTPUT PURE BINARY RESTART FILE NAME 1 line (optional)**

01-08   ID   A   "OUBNTRST"

09-40   FNAM   A   Name of pure binary (no headers or trailers) output restart file to be generated for use by a later RMA-2 simulation.

Line type **OUTRES OUTPUT RESULTS FILE NAME   1 line (optional)**

01-08	ID	A	"OUTRES "
09-40	FNAM	A	Name of UNFORMATTED output file that will be generated to contain all results from this simulation. This file may be read by standard versions of RMAPLT.

Line type **OUTBNRES OUTPUT PURE BINARY RESULTS FILE NAME** 1 line (optional)

01-08	ID	A	"OUTBNRES"
09-40	FNAM	A	Name of pure binary (no headers or trailers) output file that will be generated to contain all results from this simulation. This transparent unformatted file will require the modified version of RMAPLT to read it.

Line type **INELTFL INPUT ELEMENT INFLOW FILE NAME**   1 line (optional)

01-08	ID	A	"INELTFL "
09-40	FNAM	A	Name of input element inflow file for use in this simulation. For file format see section 4.2

Line type **INHYD INPUT BOUNDARY FLOW FILE NAME**   I line (optional)

01-08	ID	A	"INHYD "
09-40	FNAM	A	Name of input boundary hydrograph file for use in this simulation. For file format see section 4.3

Line type **INELEV INPUT BOUNDARY ELEVATION FILE NAME**   1 line (optional)

01-08	ID	A	"INELEV "
09-40	FNAM	A	Name of input elevation boundary condition file, the tidal graph file. For file format see section 4.4

Line type **INHARM INPUT TIDAL HARNONICS FILE NAME**   1 line (optional)

01-08	ID	A	"INHARM "
09-40	FNAM	A	Name of input elevation boundary harmonics file for use in this simulation. For file format see section 4.5

Line type **WINDSCL INPUT WIND DATA FILE NAME**   1 line (optional)

01-08	ID	A	"WINDSCL "
09-40	FNAM	A	Name of input wind data (location, time, velocity and direction for later interpolation to nodal values. For file format see section **4.6**

Line type **ENDFIL END OF LIST OF FILE NAME LIST INDICATOR**   1 line (reauired)

01-08	ID	A	"ENDFIL "

Line type **TI TITLE LINE**   1 line (required)

01-02	ID	A	"TI"

| 72-80 | TITLE | A | Any heading comment |

Line type **C1 CONTROL LINE 1** 1 line (rewired)

01-02	ID	A	"C1"
09-16	IPRT	I	Control for output printing: = 0 node and element input data suppressed = 1 all input data printed except initial conditions = 2 all input data printed
17-24	IDNOPT	I	Marsh option switch 0 = Marsh option inoperative -1 = Use default marsh settings at all nodes and optionally individual nodal vales (see line type **MP1**) -2 = Input overall default setting (see line type **MP**) and optionally individual nodal vales (see line type **MP1**)
25-32	IYRR	I	Year to start simulation
33-40	IDAT	I	Julian day to start simulation
41-48	TET	R	Hour to start simulation
49-56	IEDSW	I	Switch controlling turbulence closure 0 = Input constant values on line type ED 1 = Input multipliers of element dimensions on line type ED 2 = Use Smagorinsky closure.
57-64	TBFACT	R	Alfa factor applied in Smagorinsky closure
65-72	TBMIN	R	Minimum eddy viscosity using the Smagorinsky closure

Line type **C2 CONTROL LINE 2** 1 line (rewired)

01-02	ID	A	"C2"
09-16	OMEGA	R	Local latitude average, (degrees)
17-24	ELEV	R	Average initial water surface elevation. Used when no restart conditions are input
25-32	XSCALE	R	Scale factor for X coordinate inputs
33-40	ZSCALE	R	Scale factor for Y coordinate inputs
41-48	DSET	R	Depth below which nodes are dry, when drying. Active only if LI > 0 (see line type C3).
49-56	DSETD	R	Depth above which nodes are active when wetting. Active only if LI > 0 (see line type C3).

57-64	UNOM	R	Nominal velocity, used as initial guess for one dimensional nodes if not restarting. Default equals 0.25 .
65-72	HMIN	R	Minimum depth used for one dimensional nodes at startup. Eliminates possible negative depth on sloping river systems.

Line type **C3 CONTROL LINE 3 1** line (required)

01-02	ID	A	"C3"
09-16	NITI	I	Number of iterations for initial steady state solution
17-24	NITN	I	Number of iterations per dynamic solution step
25-32	NSTART	I	Starting time step (used to skip through B.C. data for restart)
33-40	NCYC	I	Number of time steps to be simulated
41-48	LI	I	Frequency of testing for dry node modification. Set = 0 to eliminate dry node operation.
49-56	ITSI	I	Iteration frequency for printing full solution if ITSI<0. This number is time step frequency for printing full solution.
57-64	IHOE	I	Switch to force higher order integration on all elements = 0 use regular integration = 1 use 16 Gauss points for all elements
65-72	IGRAV	I	Acceleration due to gravity, system of units 0 = use English units (g = 32.2) 1 = use metric units (SI) (g = 9.81)
73-80	I9NOD	I	Nine quadrilateral switch. 0 = use 8 node elements 1 = use 9 node elements

Line type **C4 FILE OUTPUT CONTROL LINE 4 1** line (optional)

01-02	ID	A	"C4"
09-16	ISAVSTRT	I	Solution cycle number to start output of the binary results file (default = 1).
17-24	ISAVFRQ	I	Solution cycle frequency of savings binary results file (default=1)
25-32	IRSTFRQ	I	Solution cycle frequency for extra restart' files (default=1)

[1]   When selected these options cause extra files named RESTART.000, RESTART.OO1 etc. to be generated at the end of solution steps. The suffix number cycles through the NUMRST files, retarting at .000. IRSTFRQ defines frequency with which these files are saved

33-40   NUMRST   I   **Number of extra** restart **files (default** =0)

Line type **SLP   CONTROLS FOR TILTING BASIN FOR INITIAL CONDITIONS 1**   line
(optional)

01-03	ID	A	"SLP"
09-16	CXREF	R	X coordinate of origin of tilt[2]
17-24	CYREF	R	Y coordinate of origin of tilt
25-32	**SLX**	R	Slope of tilt in the x-direction.
33-40	SLY	R	Slope of tilt in the y-direction.
33-40	SLFCT	R	Scale factor applied to the tilted bed.
33-40	SLFCTP	R	Former scale factor applied to the tilted bed. Required when restarting from a solution when tilting was applied.

Line type **CV CONVERGENCE TOLERANCES** 1 line (optional)

01-02	ID	**A**	"CV"
09-16	CONVX	R	Convergence tolerance for maximum change per iteration for the X-direction velocity component. This component is also used for one dimensional and parallel boundary velocities.
17-24	CONW	R	Convergence tolerance for maximum change per iteration for the Y-direction velocity component.
25-32	CONVH	R	Convergence tolerance for maximum change per iteration for the depth component.
33-40	IDRPT	I	Switch to control activation of equation dropout during iterations. 0 = No dropout allowed 1 = Activate dropout capability
41-48	DRFACT	R	Multiplier applied to convergence values above that, when satisfied, permit this degree of freedom to be considered for dropping from the next iteration.

Line type **IOV TIME OVERLAY OPTION**   (optional) When this line is entered the input time from any restart file is overlaid with the time from the control line C1

01-03	ID	A	"IOV"

Line type **SP SELECTED PRINT CONTROLS** 9 values/line, as many lines as needed
(optional)

---

[2]   The equation for the transformed bottom for node N **is** given **by:**
$$A_t(N) = AO(N) + SLFCT*[(X(N) - CXREF)*SLX + (Y(N) - CYREF)*SLY]$$

01-02	ID	A	"SP"
09-80 9 @ 8	NSPLPT(J)	I	Node locations for which alternate print file is generated

Line type ED TURBULENCE/FRICTION PARAMETERS 1 line for each element type

01-02	ID	A	"ED"
09-16	J	I	Identification number for a set of turbulent exchange coefficients.
17-24	ORT(J,1)	R	If > 0.0 Turbulent exchange coefficient associated with the X direction and the u velocity (lb-sec/ft2 or Pascal-sec).[3] If < 0.0 or turbulence switch on line type ED =1, the magnitude of this entry is treated as a scale factor applied to a notional element length in the X direction.
25-32	ORT(J,2)	R	If > 0.0 Turbulent exchange coefficient associated with the Y direction and the u velocity (lb-sec/ft2 or Pascal-sec). If < 0.0 or turbulence switch on line type ED =1, the magnitude of this entry is treated as a scale factor applied to a notional element length in the Y direction.
33-40	ORT(J,3)	R	If > 0.0 Turbulent exchange coefficient associated with the X direction and the v velocity (lb-sec/ft2 or Pascal-sec) If < 0.0 or turbulence switch on line type ED =1, the magnitude of this entry is treated as a scale factor applied to a notional element length in the X direction.
41-48	ORT(J,4)	R	If > 0.0 Turbulent exchange coefficient associated with the Y direction and the v velocity (lb-sec/ft2 or Pascal-sec) If < 0.0 or turbulence switch on line type ED =1, the magnitude of this entry is treated as a scale factor applied to a notional element length in the Y direction.
49-56	ORT(J,5)	R	Chezy coefficient for this element type. If ORT(J,5) < 1.0 Manning coefficient is assumed.
57-64	ORT(J,6)	R	Bank friction coefficient. Chezy coefficient for this element type. If ORT(J,6) < 1.0 Manning coefficient is assumed.
65-72	ORT(J,7)	R	Friction enhancement coefficient. This factor is applied to increase the bed friction when the marsh porosity is active and the water surface elevation is such that flow is in the porous zone. For further information see section 2.2.9 above

Line types CC1 and CC2 CONTINUITY LINES One set of lines for each line desired

---

[3] Note that when IEDSW = 2 (Smagorinsky closure), values must be entered for use during the initial steady state iterations if no restart file is used. Recommended values are 0.1 for ORT(J,1), ORT(J,2), ORT(J,3) and ORT(J,4).

01-03	ID	A	"CC1"
09-80	LINE(J,K)	I	Lists of node numbers which define line segments across which total flow is to be computed for
(9 @ 8)			continuity checking. Up to 32 values may be specified. Data should be input with CC1 on the first line and CC2 on the continuation line. Please note that the continuity calculation assumes the sequence of the input is positive, and thus the sign of computed flow will depend on the sequence of input values. . Only corner node numbers should be specified. These lines are also used for automatic generation of boundary conditions.

Line type **EL ELEMENT CONNECTION DATA** As many lines as required. These lines may be used to edit an input geometry file or as the entire element connection input These lines are not required

01-02	ID	A	"EL"
09-12	J	I	Element number
13-44	NOP(J,K)	I	Up to eight node numbers for element J, listed counterclockwise around the element STARTING FROM ANY CORNER
45-48	IMAT(J)	I	Element type (the number entered here corresponds to the parameters specified in the ORT array, defined above)
49-56	TH(J)	I	Element principal direction (overrides any computed value). Measured in radians counterclockwise from the x-axis.

Line type **ND NODAL LOCATION DATA** As many lines as required. These lines may be used to edit an input geometry file or as the entire nodal location input These lines are not required

01-02	ID	A	"ND"
09-16	J	I	Node number
17-24	CORD(J,1)	R	X-coordinate
25-32	CORD(J,2)	R	Y-coordinate
33-40	AO(J)	R	Bottom elevation at node J (feet)

NOTE: The coordinate values read from the above lines are multiplied by the appropriate scale factors, XSCALE and ZSCALE, and should result in the proper X and Y coordinates (feet or metres) after transformation

<u>Line type **MP**   **MARSH CONTROLS TYPE 1**</u> 1 line skip if IDNOPT' equals **-1** or 0

01-03   ID   A   "MP"

09-16   J   **I**   Unused

17-24   AC1   **R**   Bottom elevation shift for AO.
                                    Default value = 4.5. (= 1.5 metric units)

25-32   AC2   **R**   Depth range over which section reduces.
                                    Default value = 2.0. (= 0.67 metric units)

33–40 AC3   **R**   Kappa = Minimum active fraction over lower section
                                    Default value = 0.04

41-48   AC4   **R**   Absolute bottom elevation that overrides the bottom
                                    elevation shift defined with AC1 (optional)

+   Note that when IDNOPT > 0 or = -2 line type MP is required to define default
values when individual nodal values are input. Lines type MP1 do not have to include
all nodes.

Line type **MP1   MARSH CONTROLS TYPE 2**   As many lines as required, 1 line skip if
IDNOPT' equals -1 or 0 (optional otherwise)

01-03	ID	**A**	"MP1"
09-16	J	I	Corner node number
17-24	AC1	R	Bottom elevation shift for A0 at node J
25-32	AC2	R	Depth range at node J over which section reduces
33-40	AC3	**R**	Kappa = Minimum active fraction for node J over lower section
41-48	AC4	**R**	Absolute bottom elevation at node J that overrides the bottom elevation shift defined with ACI (optional)

Line type **MD   MARSH DROPOUT TYPES**   List of element types that may be considered for
dropout when the marsh porosity option is active. As many
lines as required (optional).   Note that if no lines type MD
are input all types will be considered.

01-02	ID	A	"MD"
09-80 9 @ 8	MDP	I	**A** list element type numbers which will be used to define elements eligible for dropout when the marsh porosity option is on.

Line type **WD   WIDTH DATA** As many lines as required. These lines may be used to edit an
input geometry file or as the entire nodal width input These
lines are not required

01-02	ID	A	"WD"
09-16	J	I	Node number
17-24	WIDTH(J)	R	Node J channel width at zero depth
25-32	SS1(J)	R	Left side slope at node J
33-40	SS2(J)	R	Right side slope at node J
41-48	WIDS(J)	R	Storage width associated with node J at zero depth
49-56	WIDBS(J)	R	Bottom elevation for storage width. (Note that when WIDBS = 0. bottom elevation is set to bed elevation). Values for this parameter can only be input from line data.

[+]   Note that when IDNOPT > 0 or = -2 line type MP is required to define default
values when individual nodal values are input. Lines type MPI do not have to include
all nodes.

Line type **RO   REORDERING DATA**   Not required   As many lines as needed to form a
                                                     complete data set

01-02	ID	A	"RO"
09-80	NFIXH	I	A complete list of NE element numbers which will be used to reorder the internal formation of the system
9 @ 8			equations. This feature can be used to achieve more efficient core storage allocation without reentering other existing system data (NE is the maximum element number in the network, list unused elements at the end of the data set).

Line type **SS SPECIAL SLOPE DATA** Not required   As many lines as desired

01-02	ID	A	*"SS"*
09-16	N	I	A node number
17-24	ALFAK(N)	R	Angle of rotation of coordinate system at node **N.** Overrides any internally computed value. Measured in radians counterclockwise from the x-axis.

Line type **DN   DENSITY DEFINITION** Not required   *As* many lines as desired

01-02	ID	A	"DN"
09-16	N	I	Node number
17-24	RON(N)	R	Density at node number N

Line type **ENDGEO**   Required

01-06	ID	A	"ENDGEO"

LINES FOR DEFINITION OF TIME STEPS

The data lines that follow up to and including the "ENDSTEP line should be repeated for each step of the solution. If an initial steady state simulation is to be made then a set of data for this step should be included with the time step set equal to zero.

Line type **DT TIME STEP DATA** 1 line

01-02	ID	A	"DT"
09-16	DELT	R	Time step in hours, use 0.0 for steady state.
17-24	IWIND	I	Control for wind field input: If IWIND = 0 no wind field input If IWIND = 1[4], use the original RMA2 wind formula If IWIND = 2, use the Van Dorn Formula with user supplied coefficients (see line type W02) If IWIND = 3, use the Wu Formula with user supplied coefficients (see line type W03) If IWIND = 4, use the Safaie Formula with user supplied coefficients (see line type W04) If IWIND = 5, use the Ekman Formula If IWIND = 6, use the Generic Formula with user supplied coefficients (see line type W06) If IWIND = 7, use the Van Dorn Formula with default coefficients If IWIND = 8, use the Wu Formula with default coefficients
25-32	IYRE	I	Year to end use of this time step.
33-40	IDYE	I	Julian day to end use of this time step.
41-48	IHRE	R	Hour to end use of this time step.
49-56	RELX	R	Under-relaxation factor to be applied to solution. If = 0.0. then no under-relaxation applied.
57-64	LASTITER	I	Iteration number by which under-relaxation factor has returned to 1.0. The model computes the under-relaxation factor using linear interpolation between the first iteration and LASTITER. If = 0, then LASTITER set = 10

Line type **BN NODAL BOUNDARY CONDITION DATA** As many lines as needed

01-02	ID	A	"BN"
09-16	J	I	Node number
17-24	NFIX(J)	I	Boundary condition specification for this node. Note that parallel slip boundaries are automatically generated for edges and need not be specified unless additional

---

[4]   For compatibility with earlier data sets the WIND option may **be** defined as positive or negative, the effect is the same

conditions such as specified flow, velocity or head are required at that node. To specify flow, enter 31000, except in the special case of zero x-component when 13000 must be entered. To specify velocity, enter 01000 for velocity at a slope ALFA (see above). Enter 1000 when ALFA = . Enter 11000 when both components are specified. To specify elevation, enter XX200 where XX is the appropriate velocity boundary condition or 00. Note that flow or velocity and elevation may not both be defined at a node except at corners in the network.

| 25-32 | SPEC(J,1) | R | The specified X-direction flow or velocity as appropriate (ft3/sec/ft or ft/sec or in metric units m3/sec/m or m/sec) |

| 33-40 | SPEC(J,2) | R | The specified Y-direction flow or velocity as appropriate (ft3/sec/ft or ft/sec or in metric units m3/sec/m or m/sec) |

| 41-48 | SPEC(J,3) | R | The specified water surface elevation. |

Line type **EFA GLOBAL ELEMENT INFLOW DATA** 1 line if appropriate

| 01-03 | ID | A | "EFA" |

| 09-16 | SIDF(J) | R | Element inflow per unit area or length applied to all elements as appropriate to the element type. |

17-24	NEST	I	Switch defining units of inflow
			0 = Units are flow per unit length or area
			1 = Units are total flow over the element.

| 25-32 | LABL | I | = 1 if during dynamic simulation the user wishes to interpolate values from the values on the element inflow hydrograph file, Selection of this value triggers a request for the element inflow hydrograph file at the beginning of dynamic simulation. |
| | | | = 0 or leave blank if flow values will be read from this file. |

Line type **EFE ELEMENT INFLOW DATA** As many lines as needed

| 01-03 | ID | A | "EFE" |

| 09-16 | J | I | Element number |

| 17-24 | SIDF(J) | R | Element inflow per unit area or length as appropriate to the element type. |

25-32	NEST	I	Switch defining units of inflow
			0 = Units are flow per unit length or area
			1 = Units are total flow over the element.

| 33-40 | LABL | I | = 1 if during dynamic simulation the user wishes to interpolate values from the values on the element inflow hydrograph file. Selection of this value triggers a request for the element inflow hydrograph file at the beginning of dynamic simulation. |
| | | | = 0 or leave blank if flow values will be read from this file. |

Line type **QC   CONTINUITY LINE INFLOW DATA (TYPE 1)**   Note that parallel flow is maintained at the end of the line. As many lines as needed

01-02	ID	A	"QC"
09-16	J	I	Continuity line number
16-24	QTOT	R	Total flow crossing the continuity line
25-32	THETA	R	Direction of flow (radians measured anti-clockwise). Note that the boundary directions are adjusted to maintain parallel flow.
33-40	LABL	I	= 1 if during dynamic simulation the user wishes to interpolate values from the input hydrograph, values on the hydrograph file. Selection of this value triggers a request for the boundary inflow hydrograph file at the beginning of dynamic simulation. = 0 or leave blank if flow values will be read from this file

Line type **QF   CONTINUITY LINE INFLOW DATA (TYPE 2)**   Flow applied across a specified line. Tapered to zero at the each end. As many lines as needed

01-02	ID	A	"QF"
09-16	J	I	Continuity line number
16-24	QTOT	R	Total flow crossing the continuity line
25-32	THETA	R	Direction of flow (radians measured anti-clockwise).
33-40	LABL	I	= 1 if during dynamic simulation the user wishes to interpolate values from the input hydrograph, values on the hydrograph file. Selection of this value triggers a request for the boundary inflow hydrograph file at the beginning of dynamic simulation. = 0 or leave blank if flow values will be read from this file

Line type **HC   ELEVATION SPECIFICATION DATA**   As many lines as needed each line paired with lines type HCN as required

01-02	ID	A	"HC
09-16	J	I	Continuity line number.
17-24	ELEVS	R	Specified elevation for all nodes on this line.
25-32	LABL	I	= 0 or leave blank if elevation values will be read from this file = 1 if during dynamic simulation the user wishes to interpolate values from the input tidalgraph values on the tidalgraph file. Selection of this value triggers a request for the tidal hydrograph file at the beginning of dynamic simulation. = 2 if during dynamic simulation the user wishes to compute elevation values from the input harmonic values on the tidal

harmonic file. <u>Selection of this value triaaers a reauest for the tidal harmonic file at the beainnina of dvnamic simulation.</u>

<u>Line tvpe</u> **HCN   FLOW DISTRIBUTION SPECIFICATION FOR ELEVATION BOUNDARIES**
<u>If needed, repeat until all nodes are accounted for</u>

01-03	**ID**	A	"HCN"
09-80 9 @ 8	WTS	R	Scalar multipliers for velocity distribution applied to each node (including mid-sides) Note that an input value of 0.0 is equivalent to 1.0. This input slightly relaxes the elevation specification along the line but allows the user to force a distribution of velocity.

<u>Line tvpe</u> **SQC   ELEVATION - DISCHARGE BOUNDARY DATA** *As* <u>manv lines as reauired</u>

01-03	ID	A	"SQC"
09-16	J	I	Continuity line number
17-24	A I	R	Coefficients in the elevation flow
25-32	*A2*	R	Relationship for line J given by
33-40	EO	R	$Q = A1 + A2 * (ELEV - E0)$ ** C
41-48	C	R	
49-56	THETA	R	Direction of flow (radians measured anti-clockwise). Note that the boundary directions are adjusted to maintain parallel flow.

<u>Line tvpe</u> **WVA   GLOBAL WIND DATA**   <u>1 line if needed</u>

01-03	ID	A	"WVA"
09-16	J	I	Leave blank
16-24	SIGMA(J,1)	R	The wind velocity at all nodes.
25-32	SIGMA(J,2)	R	The angle between the wind velocity and the X-axis for all nodes (degrees measured anti-clockwise). NOTE: The wind values read here are used only for the calculation of the wind effect on fluid motion.
33-40	IM	I	Flag to identify units of SIGMA(J,1) If IM = 0, then all subsequent SIGMA(J,1) values are given in miles/hour If IM = 1, then all subsequent SIGMA(J,1) values are given in meters/second

<u>Line tvpe</u> **WVN   NODAL WIND DATA**   <u>As manv lines as needed</u>

01-03	ID	A	"WVN"
09-16	J	I	Node number J

16-24	SIGMA(J,1)	R	The wind velocity at node J.
25-32	SIGMA(J,2)	R	The angle between the wind velocity and the X-axis at node J (degrees measured anti-clockwise). NOTE: The wind values read here are used only for the calculation of the wind effect on fluid motion.
33-40	IM	I	Flag to identify units of SIGMA(J,1) If IM = 0 on the first line type WVA or WVN, then all subsequent SIGMA(J,1) values are given in miles/hour If IM = 1 on the first line type WVA or WVN, then all subsequent SIGMA(J,1) values are given in meters/second

Line type **WVF WIND DATA INTERPOLATED FROM FILE**   As many lines as needed. File name must have been entered earlier.

01-03	ID	A	"WVF"
09-16	IM	I	Flag to identify units of wind velocity. If IM = 0, then values on file are given in miles/hour If IM = 1, then values on file are given in meters/second

Line type **W02 WIND CONTROL DATA TYPE 2** 1 line, use if ABS(IWIND) eaual to 2

01-03	ID	A	"WO2"
09-16	YHT*	F	Anenometer height, meters
17-24	YALPHA	F	Smooth water wind stress coefficient
25-32	YBETA	F	Rough water wind stress coefficient
33-40	WCRIT	F	Critical wind velocity for wave formation, meters/second

Line type **W03 WIND CONTROL DATA TYPE 3** 1 line, use if ABS(IWIND) eaual to 3

01-03	ID	A	"W03"
09-16	YHT*	F	Anenometer height, meters
17-24	CWIND	F	Wind stress coefficient
25-32	RHOAIR	F	Air density If ABS(RHOAIR) > 0.0 then user supplies value in grams/cubic cm If ABS(RHOAIR) <= 0.0 then the default value of 0.001226 grams/cubic cm is used (dry air at 1 atm pressure and 15 degrees Centigrade)

Line type **W04 WIND CONTROL DATA TYPE 4** 1 line, use if ABS(IWIND) eaual to 4

01-03	ID	A	"WO4"

---

* The program presently requires wind velocity at a height of 10 meters. YHT is ignored.

09-16	AWIND	F	Charnock's constant If ABS(AWIND) >0.0 then user supplies value If ABS(AWIND) <= 0.0 then the default value of 0.0332 is used

17-24	ETA	F	Dynamic roughness in cm. No default.

25-32	AGRAV	F	Acceleration due to gravity If ABS(AGRAV) > 0.0, then user supplies the value in cm/(sec*sec) If ABS(AGRAV) <= 0.0 then default value of 979.965 cm/(sec*sec) is used . Note: this value of AGRAV affects wind formula only

33-40	RHOAIR	F	Air density
			If ABS(RHOAIR) > 0.0 then user supplies value
			  in grams/cubic cm
			If ABS(RHOAIR) <= 0.0 then the default
			  value of 0.001226 grams/cubic cm is
			  used (dry air at 1 atm pressure and
			  15 degrees Centigrade)

Line type **W06 WIND CONTROL DATA TYPE 6** 1 line. use if ABS(IWIND) equal to 6

01-03	ID	A	"WO6"

09-16	YHT[*]	F	Anenometer height (meters). No default

17-24	CWIND	F	Wind stress coefficient. No default

25-32	EXP	F	Empirical exponent. No default

33-40	RHOAIR	F	Air density
			If ABS(RHOAIR) > 0.0 then user supplies value
			  in grams/cubic cm
			If ABS(RHOAIR) <= 0.0 then the default
			  value of 0.001226 grams/cubic cm is
			  used (dry air at 1 atm pressure and
			  15 degrees Centigrade)

Line type **FC FLOW CONTROLLER DATA**   As many lines as needed

01-02	ID	A	"FC"

09-16	NJN	I	Flow controller identifier

17-24	NJT1	I	Flow controller type

= 1 Point source of flow, conditions are flow out equals flow in plus source (AJ1), and equal total head at each node of the control element. **Note that this option is not available** for **two-dimensional control structures.**

= 2   Flow is a reversible function of head loss across the structure. Conditions are flow out = flow in, and flow
 $Q = AJ1 + BJ1*(HN1-HN2-CJ1)**GAM1$ in the direction

QD1, where HN1 and HN2 are water surface elevations at the nodes of the control structure element. Note that if HN1-HN2 is negative the sign of the flow direction is reversed.

= 3 Flow is a irreversible function of head loss across the structure. Conditions are flow out = flow in, and flow $Q = AJ1 + BJ1*(HN1-HN2-CJ1)**GAM1$ in the direction QD1, where HN1 and HN2 are water surface elevations at the nodes of the control structure element. Note that if HN1-HN2-CJ1 is negative then the $Q = 0$.

= 4 Flow is a function water surface elevation. Conditions are flow out = flow in, and flow $Q = AJ1 + BJ1*(HN1-CJ1)**GAM1$ in the direction QD1, where HN1 is the water surface elevation at the first node of control structure element

= 5 Head loss is a function of flow. Conditions are flow out = flow in, and head loss $HN1 - HN2 = AJ1 + BJ1*(Q)**GAM1$ in the direction QD1, where HN1 and HN2 are the water surface elevations at the nodes of the control structure element.

= 6 Flow is a function of water surface elevations. Conditions are flow out = flow in. When levels are below BJ1, flow = 0. When one level is above BJ1, flow is in the appropriate direction and given by: $Q = AJ1*(HN1-BJ1)**GAM1$ or $Q = AJ1*(HN2-BJ1)**GAM1$ When both levels are greater than BJ1, flow is in the appropriate direction and given by $Q = AJ2*(HN1-HN2)**CJ1$ HN1 and HN2 are the water surface elevations at the nodes of the control structure element and AJ2 is given by $AJ2 = AJ1*(HN1-HN2)**(GAM1-CJ1)$

25-32	AJ1	R	
33-40	BJ1	R	
41-48	CJ1	R	AJI, BJ1, CJ1, GAM1 and QD1 are defined by
49-56	GAM1	R	the options defined above.
57-64	QD1	R	

Line type **ENDSTEP** Required

01-07	ID	A	"ENDSTEP"

Line type **ENDDATA** Required

01-07	ID	A	"ENDDATA"

4.2   *File format for the element inflow hydrograph file*

Line type **TE   ELEMENT HYDROGRAPH TITLE LINE**   1 line

01-02	ID	A	"TE"
09-80	TITLE	**A**	Any heading comment

Repeat lines type **QEI** and **QE** for each element inflow usina hydrograph data.

Line type **QEI ELEMENT IDENTIFIER**   1 line

01-03	ID	A	"QEI"
09-16	NCL	I	Element number for inflow. To trigger assigment of inflow to all elements set NCL = -9999
17-24	NEST	I	Switch defining units of inflow 0 = Units are flow per unit length or area 1 = Units are total flow over the element.
25-32	IYDAT	I	Starting year for the data set

Line type **QE HYDROGRAPH DATA**   As many lines as required

01-02	ID	A	"QE"
06-08	IDY	**I**	Julian day
09-16	TSS	**R**	Time (hours)
17-24	QSS	R	Flow rate in units defined by MEST above.

Line type **ENDDATA END ELEMENT HYDROGRAPH DATA FILE**   1 line (Required)

01-07	ID	A	"ENDDATA"

4.3   *File format* for **the** *boundary hydrograph file*

Line type **TH HYDROGRAPH TITLE LINE   1 line**

01-02	ID	A	"TH"
09-80	TITLE	A	Any heading comment

Repeat lines type **CLQ** and **QD** for each continuity line usina hydrograph data.

Line type **CLQ CONTINUITY LINE IDENTIFIER   1 line**

01-03	ID	A	"CLQ"
09-16	NCL	I	Continuity line number
16-24	IYDAT	I	Starting year for the data set

Line type **QD HYDROGRAPH DATA   As many lines as** rewired

01-02	ID	A	"QD"
06-08	IDY	I	Julian day
09-16	TSS	R	Time (hours)
17-24	QSS	R	Flow rate

Line type **ENDDATA END HYDROGRAPH DATA FILE   1 line (Required)**

01-07	ID	A	"ENDDATA

4.4   File format for the tidal-graph file

Line twe **TT TIDAL-GRAPH TITLE LINE**   1 line

01-02	ID	A	"TT"
09-80	TITLE	A	Any heading comment

Repeat lines type **CLH** and **HD** for each continuity line usina hydrograph data.

Line type **CLH CONTINUITY LINE IDENTIFIER**   1 line

01-03	ID	A	"CLH"
09-16	NCL	I	Continuity line number
16-24	IYDAT	I	Starting year for the data set

Line type **HD TIDAL GRAPH DATA**   As many lines as reauired

01-02	ID	A	" H D
06-08	IDY	I	Julian day
09-16	TSS	R	Time (hours)
17-24	HSS	R	Elevation

Line type **ENDDATA END TIDAL-GRAPH DATA FILE**   1 line (Required)

01-07	ID	A	"ENDDATA"

4.5   File format for the tidal harmonic file

<u>Line type **A TITLE LINES** 2 lines</u>

 1-80   TITLE   Heading or comments

<u>Line type **B DATE DATA** 1 line</u>

01-03   IYR   Year as defined for the harmonics

03-06   IMO   Month as defined for the harmonics

06-09   IDA   Day as defined for the harmonics

10-14   IHR   Starting hour for the harmonics

15-24   TDUM   Blank

25-34   UTILT   Adjustment to computed tidal stage

<u>Line type **C HARMONICS CONTROL**   1 line</u>

 1-05   NHM   Number of harmonics in file.

<u>Line type **D   BLANK** 1 line</u>

Blank line that may be used as a header for data that follows

<u>Line type **E HARMONIC DATA** NHM lines</u>

01-10   RH   Amplitude of harmonic

11-20   SP   Speed of harmonic (radians/hr)

21-30   EQ   Equilibrium phase advance degrees (date dependant)

31-40   ZP   Phase lag for this location degrees.

41-50   FCT   Dividing factor for amplitude of this constituent (date dependant)

The formula for the tidal elevation is the sum for each component of

$$RH/FCT*\cos(SP*time+(EQ-ZP)/57.296) + UTILT$$

35

4.6 File format for the wind data file

Line type **TI WIND FILE TITLE LINE**   1 line

01-02	ID	A	"TI"
09-80	TITLE	A	Any heading comment

Line type **STL STATION IDENTIFIER** As many lines as rewired

01-03	ID	A	"STL"
09-16	ISTATLB	I	Station number
16-24	STATCOORX	I	x coordinate of station.
25-32	STATCOORY	I	y coordinate of station.

Line type **ENDSTL END STATION DATA** 1 line (Required)

01-06	ID	A	"ENDSTL"

Repeat lines type **WBC** and **WDT** for each time with available data. Terminate each set with an **ENDWBC** line.

Line type **WBC TIME DATA**   1 line per set

01-03	ID	A	"WBC"
09-16	BCENDHRR		Time (hours)
17-24	BCENDDY	I	Julian day
25-32	BCENDYR	I	Year

Line type **WDT WIND DATA**   As many lines as rewired

01-03	ID	A	"WDT"
09-16	IBCLBL	I	Station number
17-24	BCWIND	I	Wind speed (units given by IM)
25-32	BCDIR	I	Wind direction (Radians from the x-axis)

Line type **ENDWBC END WIND DATA** 1 line (Required)

01-06	ID	A	"ENDWBC"

Line type **EOF END WIND DATA FILE**   1 line (Required)

01-03	ID	A	"EOF"

5. DEMONSTRATION PROBLEM

5.1 Introduction

This chapter is designed to describe the process of setting up and running a test case. A description of the steps necessary to complete a simulation will be followed by sections discussing boundary and initial condition data set-up, selection of eddy and diffusion coefficients, model output and interaction with post-processors. In order to give structure to the description that follows a typical estuarial problem is used as a basis. Application of the model to other systems such as lakes, bays or offshore regions can be easily extrapolated from this discussion.

5.2 Outline of Steps

Preparation and carrying out of a simulation using RMA-2 can be broken down into the following steps

(a)   Define study area
(b)   Evaluate data availability
(c)   Select levels of approximation for study area.
(d)   Prepare preliminary finite element mesh
(e)   Test network.
(f)   Calibrate and verify data refining approximations as required
(g)   Carry out production runs and post process final results.

The sections that follow will describe each of the above steps in more detail.

5.2.1   Define study area

The selection of the study area can be one of the most difficult steps in the overall process. A compromise must be reached between the computational capabilities of the model and the need to ensure that an appropriate study area is defined. The study area clearly must cover the area of interest. The question is how much more area is required. Broadly speaking the boundaries should be far enough away that either their location does not influence the simulation results or that a boundary condition is known from available data and will not depend on velocities, water surface elevations or concentrations computed in the simulation. An example of the former would be moving an estuary upstream boundary far enough upstream that it is not influenced by tidal activity. An example of the latter would be imposition of a tidal stage boundary at the estuary mouth. Note that in cases where a pollutant fresh water might be transported through an estuary mouth it might be necessary to move the tidal stage boundary off-shore. Shoreline boundaries present a different problem. A decision must be made as to whether a fixed boundary is to be used or whether it will be allowed to move as the stage varies. If the former approach is selected then a maximum bottom elevation must be established that ensures that the location will be submerged at all times. If the latter method is to be used the study area must be defined to cover the inundated area at high tide. Note that a decision to use the variable area approach can increase run time significantly. Unless shoreline currents or concentrations are critical to the analysis it is generally desirable to use the fixed shoreline approach.

5.2.2   Evaluate data availability

In order for a successful simulation to be undertaken RMA-2 requires several types of data. The following information is required:

(a)   Map of study area showing contours of bathymetry or at least spot values.
(b)   Data describing boundary conditions and system constants. Specifically:
          Any inflows into the system. Hydrographs are required for unsteady flow problems

Locations and magnitude of specified water surface elevations, hyetographs for unsteady flow cases.

Wind magnitude and direction if appropriate

Estimates of Manning or Chezy coefficients for the entire bed area. They may vary from element to element, as desired

(c)   Verification data. Specifically:

Values for current magnitude and direction at points within the system.

Water surface elevations at points within the system.

When data is unavailable in any of these categories the user must either plan data measurement exercises or be prepared to make reasonable approximations in order to continue with the simulation process.

5.2.3   Select levels of approximation for the study area

Simulation of the study area defined in the step above may not require that the complete area be modelled using two-dimensional elements. RMA-2 is structured to allow the user to select onedimensional, or two-dimensional depth averaged. Examples of applications where use one-dimensional elements can simplify approximations include:

(a)   Upstream along rivers that can be considered fully mixed laterally , where the important factor is inclusion of all the tidal area to ensure correct simulation of the full tidal prism.

(b)   In rivers or straits within the system that are not close to area of modelling interest.

On the basis of available data and the study area the user must also decide whether to include the baroclinic term in the simulation. In RMA-2 density distributions are not currently simulated and if variable values are desired the user must enter these through data.

5.2.4   Prepare preliminary finite element mesh

Finite element meshes may be constructed in any type of irregular form. However model accuracy is sensitive to the form and size of the mesh. A general rule is that areas where large variations in a parameter are expected should have a higher level of resolution. The network can consist of assemblages of one-dimensional line elements and two-dimensional triangular and quadrilateral elements. Line elements are defined with two corner nodes and one mid-side node. Triangular and quadrilateral elements are defined with three corner nodes and three mid-side nodes and four corner nodes and four mid-side elements respectively. With the exception of transitions from one-dimensional elements to two-dimensional elements that will be discussed later, a node is either uniquely corner or mid-side for all elements to which it is connected. In other words: *A corner node for one element cannot be a midi-side node for another*. Nodes are defined by giving the **x** and y coordinates and the bottom elevation in a cartesian axis system. For nodes associated with one dimensional elements the channel is assumed to have a trapezoidal cross section and additional data is required defining cross-section bottom width and side slopes. Elements are defined by listing the nodes numbers in a counterclockwise direction around an element (or along a line for one-dimensional elements), starting at a comer node. An element type number which must be defined for every element allows the user to identify classes of element that can have the same properties such as Chezy or Manning coefficient, eddy viscosity or diffusivity.

As a general guideline, elements are best constructed when they follow the line of the bathymetry and the expected direction of flow. One potential area of difficulty is matching the geometric outline of the system, particularly in areas where the shoreline is very irregular. If this area is not close to the study area it is usually more cost effective to smooth the boundary outline so that the resolution does not have to be greatly increased. Note that, as pointed out in King (1991), boundaries with sharp curves require considerable element definition in the vicinity in order to capture the local influence on flow. Insertion of curved boundary elements that just match the geometric form may be inadequate for that purpose. Where possible anticipated geometric design alternatives should be incorporated into the original network

configuration so that elements can be dropped or added to the system without major reconfiguration during the production runs.

The use of junctions of one dimensional elements and combinations of one and two-dimensional elements is documented in King (1987). Implementation of transition is achieved by defining a one dimensional element with the usual three nodes followed by the two additional nodes that define the entire transition. Figure 5-1 illustrates and lists the connections.
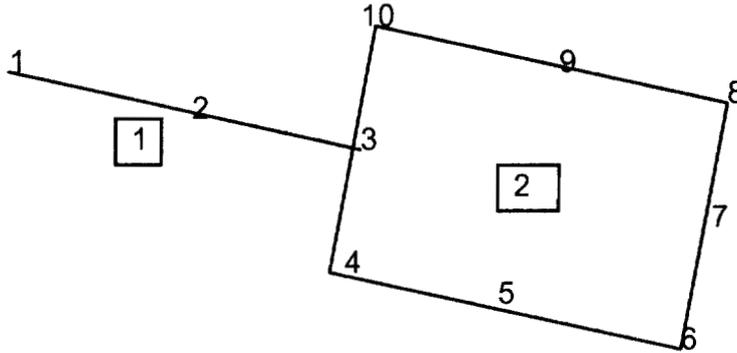
Connections for element # 1   1 **2** **3**

Connections for element # 2   4 *5* **6** **7** *8* **9** 10 **3**

Figure 5-1 Connection of 1-D to 2-D element

Note that node number 3 is a corner node for the one dimensional element and a mid-side node for the quadrilateral element. *This is the only circumstance when a node may function both as a mid-side node and a corner node.*

At a junction where more than three one-dimensional elements meet, each element entering that junction must be terminated by a node with a different number. Each of these nodes may have their own individual properties, such as bottom elevation or width. An additional - special purpose- element must be defined listing the node numbers that form the junction together with a special element type number that defines the junction type.

       Type 901 junction   enforces flow continuity and equal water surface elevation at the junction.

       Type 902 junction   enforces flow continuity and equal total head at the junction.

       Type 903 junction   enforces flow continuity and momentum conservation for the first two nodes listed. Other nodes in the junction are constrained to a water surface elevation equal to that of the average water surface elevation of the first two momentum conserved nodes

Network construction may be undertaken manually using tracings of the system configuration to layout elements and nodes. When this method is used, RMA-1 may be used to create the interface geometric file. **As** an alternative, an interactive computer graphics approach may be undertaken using RMAGEN to form the grid and to create the interface file. Note that during the network definition the user is not required to define the coordinates of a mid-side node number. If no location is provided, both RMA-I and RMAGEN place the node at the exact middle of the side. Similarly the element listing may omit the node number for all regular mid-side nodes (that is, they are set to zero) and the pre-processors will automatically insert a node number and locate it at the exact middle of the side.

RMA-1 is a line input based pre-processor for RMA-2 and other RMA models. It is documented in King and Rachiele (1990). It is used to generate graphics files, to verify consistency of the network, and to pre-process the element ordering system to minimise front width during execution of programs such as RMA-2. This element reordering is internal to the data system and actual element numbers are not changed. The method used is presented in King (1970). For this algorithm the user must supply the initial element or elements, the

program then develops an estimate of the best element sequence for the equation solver built into RMA-2. RMA-1 has two other subsidiary capabilities.

1. For very regular problems the finite element network may be generated from a series of rectangles. The user specifies the vertices of rectangles and the resolution (which may vary from rectangle to rectangle). The program then automatically assembles a complete network, interpolating as necessary.
2. In early versions of the RMA hydrodynamic codes it was a requirement that along boundaries the slope of the two external sides entering a comer node be identical. This was to ensure a unique definition of the velocity components. This requirement is now relaxed, although it is still desirable the boundaries be as smooth as reasonably possible. An option of RMA-I permits automatic generation of mid-side coordinates to force this parallel condition.

The final output from RMA-1 is a binary file containing the geometric structure and the "best" re-ordering sequence.

RMAGEN is a graphics based pre-processor for RMA-2 and other RMA models. It is documented in King (1992). It may be used to interactively generate and refine finite element networks. As input it may use a previously generated ASCII or binary file. Note that the ASCII file is compatible with the format used in RMA-1 and the programs may be used interchangeably. Once an element system has been created RMAGEN may be directed to carry out consistency checks of the network, to pre-process the element ordering system and save the binary and ASCII files.

5.2.5   Test network

The first tests are designed first to investigate whether the network is correctly and consistently constructed and then to study the dynamic response of the system.

Steady state tests of consistency can be constructed by establishing boundary conditions that should induce no circulation, for example a fixed elevation along one boundary and no flow into the system. If a condition of zero flow into the system is specified for all boundaries it does not have a unique solution for water surface elevation and should not be used with the steady state option. A second set of steady state tests can be used to evaluate network detail and general model performance. In these studies, upstream flows should be introduced to the system and continuity checks undertaken at various sections in the system. In RMA-2 continuity lines are constructed by entering data describing a sequence of *corner nodes* that form a line along element boundaries. The model automatically computes flow across each of these lines and reports the magnitudes in the output file. Levels of consistency in continuity are a useful measure of model performance. This finite element formulation ensures that overall flow continuity is maintained, that is, total flow in equals total flow out at steady state. However at individual locations the continuity equation is only satisfied in the same weighted sense that the momentum equations are satisfied. It has been found that satisfactory overall performance is usually achieved when flow continuity across section lines is within five per cent of the nominal. When this level of accuracy is not achieved refinement of the grid is recommended. Note that to carry out this test the user can often increase the upstream level of inflow above that which is physically realistic to ensure reasonable velocities (that is, the levels expected in tidal simulation) in other sections.

When the network has been shown to be correctly constructed dynamic simulations can be used to evaluate overall model response to time varying boundary conditions. These initial tests can be used to confirm the appropriateness of the level of detail in the network and that the time step captures the features of the system response.

5.2.6   Calibrate and verify data, refining approximations as rewired

Initial calibration should be based on adjustment of the horizontal eddy and diffusion coefficients and the bottom friction coefficients for the network. Simulations for this purpose will require definition of appropriate initial and boundary conditions. Frequently there is not sufficient data to establish initial conditions at all nodes and these values must be generated. Section 5.3 below describes an approach

As calibration is attempted it may be necessary to refine the network in certain areas to improve the performance. the initial objective is to get good comparisons for the water surface elevations and their phase and timing. Calibration to match velocities is more sensitive and should represent the final steps of the calibration process.

5.2.7   Carry out production runs and post process final results.

Production runs represent the evaluation of potential design alternatives, creation of flow data bases for subsequent water quality simulations or evaluation of various hydraulic inputs. In so far as is possible they should be executed with a minimum of changes to the calibrated system. As previously mentioned design alternatives should be incorporated into the original network configuration. Section 5.9 discusses the interaction of results from RMA-2 with available post-processors.

5.3   Data Set-up for Initial Conditions and Boundaries

5.3.1   Initial Conditions

Generation of initial conditions is critically important for RMA-2 simulations. Rarely is there sufficient data to define values of velocities and depths at all the nodes.

Several approaches may be used to initiate simulation. All require preliminary simulations using RMA-2. The restart file that may be generated during an RMA-2 simulation is a powerful tool that can be used to transfer conditions from one simulation to be initial conditions for a later simulation. The only requirement is that the geometry be consistent from simulation to simulation. In the paragraphs that follow some of these alternatives will be described.

*1.   Dynamic spin **up** to obtain initial conditions.*
In this approach a dynamic simulation is executed for a long enough period that the initial conditions have been washed out of the system. The assumed starting condition is usually a level water surface with zero velocities. This method is most suitable for hydraulic characteristics such as water surface elevations and currents that are required for an RMA-2 simulation. For complex systems however it can take a very long time to evolve into realistic conditions.

2.   Artificial smooth redistribution using steady state simulation.
In this approach a limited amount of known initial data values are input as specified values at nodal locations both internal to the system and on the boundaries. A steady state simulation is then undertaken. The resulting solution is then saved as a restart file. This simulation has the effect of defining non zero values for velocities and depths at all locations in the system. This restart file may then be used with the dynamic spin up (shortening it) to create a consistent starting condition.

3.   Manipulation and interpolation of available data.
When sufficient data is available data interpolation techniques may be used to create restart files in the correct format for input to RMA-2.

5.3.2   Selection of Boundary Conditions

Boundary conditions may be entered into RMA-2 in a number of ways depending on the type of condition required. Values may be entered for individual nodes or for lines of nodes.

Individual node conditions are generated by setting up data containing a node number followed by a key number defining the boundary condition type and then the desired specified value.

The lists of nodes defined as flow continuity checks may also be used to specify boundary conditions along lines of nodes. Note that although mid-side nodes are not defined in the

input data for continuity lines they are internally added to the list and function in any boundary condition constructed along a continuity line.

Definition of boundary conditions is not required for parallel flow boundaries along a shoreline. If the boundary tangents are not exactly equal for elements adjacent to a boundary node, angles for flow direction are automatically computed to ensure that "leakage in" equals "leakage out".

5.4   Selection of Eddy Coefficients

Two dimensional simulation requires selection of both horizontal eddy coefficients. **RMA-2** uses an empirical approach to selection of these coefficients. The user may input either:
(a)   fixed constant values for groups of elements by type
(b)   if the turbulent exchange coefficients are entered as a negative numbers they receive special treatment. The magnitude of each entry is treated as a scale factor applied to an appropriate turbulence scale for this element. The element turbulence length scales are computed from the notional characteristic dimensions of the element.   These dimensions have been selected because elements size controls the resolution of eddies in the system. That is, smaller elements resolve smaller scale turbulence effects. Conceptually the units of the scale factor are those of density times velocity and its size can reflect expected current magnitude. This method also resets the principal direction for these coefficients based on the characteristic directions for each element.

The methodology for this latter approach is presented below.

5.4.1   Computation of element characteristic directions and lengths

Figure **5-2** shows three generalised element shapes' that are represent typical element shaped that will be analysed in this section. Element type (a) is **a** quadrilateral that has two the longest sides opposite one another(ij and ik in the figure). For this element the principal x direction is defined as along a line drawn through the mid points of the shorter sides. The y direction is perpendicular to this direction. Element type (b) is a quadrilateral that has the two longest sides adjacent to one another (ij and jk in the figure).   The principal x direction is computed as along a line drawn between the node between the two longest sides (j) and the midpoint of a line joining the nodes at the other end of these two longest sides (ie., from **i** to k). Once again the y direction is perpendicular to this computed direction. Element type (c) is a triangle.   For this element type the principal x direction is computed as along a line drawn between the node between the two longest sides (j) and the midpoint of the element side joining the nodes at the other end of these two longest sides (ie., from i to **k).**

The characteristic length in the x direction for element type (a) is computed as the length of this line (ie., a-a'). The characteristic length in the y direction is computed as the element area divided by the length a-a'. For element type (b) the process is repeated using the length **j-b** for the x-direction and the same computation for **y** direction characteristic length as for element type (a).   For element type (c) the length in the x-direction is computed as the distance (j-c) and the y direction length is computed as twice element area divided by the **x** direction length.

---

[5]Note that curved elements are treated as straight sided in this analysis.

Element type(a)   Element type **(b)**   Element type **(c)**
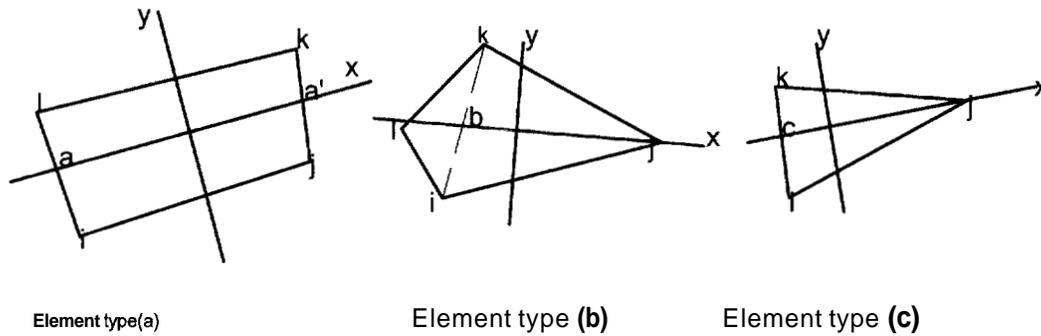
Figure **5-2** Element types for Computation of
Characteristic Lengths and Directions

5.5 Selection **of** Friction Formulation

As implemented in RMA-2 two friction formulations are permitted. The traditional forms for
depth averaged flow are given by:

(a)   Manning's Equation $S_f \quad = \quad \dfrac{n^2 . u_m^2}{h^{1.33}}$ in metric units   5.1

(b)   Chezy Equation $S_f \quad = \quad \dfrac{u_m^2}{C^2 h}$ 5.2

where u_m is the mean velocity over the water depth

These formulations represent frictional shear applied as average forces over the entire depth
(h). For use as boundary shears they must be multiplied by the depth (h). Thus:

(a)   Manning's equation $t_f \quad - \quad \dfrac{n^2 . u_m^2}{h^{0.33}}$ in metric units   5.3

(b)   Chezy's equation $t_f \quad = \quad \dfrac{u_m^2}{C^2}$ 5.4

RMA-2 detects whether n or C is being input by testing the magnitude of the input value. If
the value is greater than 1.0 then the Chezy formulation is used, if the value is less than 1.0
then the Manning formulation is used. Values for n and C may vary over the system but are
constant over each element. They are also fixed over time. There are however no inherent
limitations that restrict the formulation that may be used. Modifications into forms such as the
Darcy-Weisbach equation can be incorporated into the model equations

5.6 Model Convergence Parameters and Tests

Because the model uses an iterative procedure to solve the non-linear equations it is
necessary to define when an iteration is complete. RMA-2 uses two methods to terminate
iteration.

First, for both the initial steady state and subsequent dynamic iterations a maximum number
of allowable iterations are defined. The number required for this purpose varies with the type
of problem.   For example homogeneous solutions converge much faster than stratified
problems. Also steady state problems converge more slowly than individual time steps of a
dynamic solution. This is due to the superior initial estimate that can be used in dynamic
iterations.

Second a convergence tolerance may be specified for each active dependent variable used in the simulation. If the maximum absolute value of the change during an iteration is less than the specified value for any constituent or set of constituents, that constituent is considered convergence and solution automatically jumps to the next iteration that involves an unconverged constituent. If none are found the solution is considered converged.

As a guide to performance of the model, the convergence parameters, both maximum values and average values are saved in the output file for every iteration of every solution step.

Note that if the change in depth exceeds 100.00 in any solution step the system is declared divergent and execution terminated immediately.

5.7   Model Output

Model output is directed towards four different files.
1. The ASCII results file contains the most comprehensive output from RMA-2.
2. The binary results file contains the nodal values of velocity, depth and water surface elevation for all time steps of the simulations
3. The binary restart file contains a single set of results for the all dependent variables and their time derivatives for the last completed iteration of the simulation. This file is identical in format to the input restart file that may be used to continue the simulation with a later computer run.
4. A summary output file that permits hydrographs to be output for specified nodes.

The section that follows briefly describes the information available on the ASCII results file.

5.7.1   The Output Results File

The standard output file has the following major components
- Reprint of the major control data lines
- List of the generated element connections (optional)
- List of eddy coefficients and continuity lines
- List of the original and generated nodal coordinates and bottom elevations and nodal boundary and initial conditions (optional).
- Convergence information for each iteration, including average and maximum values of changes in active dependent variables during each iteration.
- Values of flow across designated continuity lines during each iteration.
- Values of nodal velocities, depths, optionally for each iteration, for each completed time step or at a specified frequency between time steps.

5.8   Interaction with Post-processors

The binary results file and the three-dimensional geometry file provide the major interaction with the post processors.

RMAPLT is the graphical post-processor, see King (1993), that may be used to:
(a)   Plot velocity vectors or specific flow vectors either at nodal points or on a regular grid that may be laid over the system.
(b)   To draw contours of depth or water surface elevation for the entire or selected areas of the system. The contours may be viewed on the screen as filled colour areas or as isolines.
(c)   To plot time histories at selected nodes for the various components.
(d)   To plot cross-section magnitudes of various components.

5.9   Notes on Application of RMA-2

5.9.1   Initial Conditions

Initial condition of exactly zero velocity at points can present difficulties if there is no location with a specified velocity that acts as a constraint, or no specified water surface elevation. This problem is caused because in a zero velocity condition there are no apparent energy losses and an iteration can shift the solution to a very high velocity regime.

5.9.2   Modellina of Point Inflows

Point inflows may be modelled using three methods:
1. Approximate the point inflow as a distributed element load and use a small element. This approach is suited to problems with small inflows where the detailed currents around the inflows are not required.
2. Create a nozzle structure, see Figure 5-3, for an example.
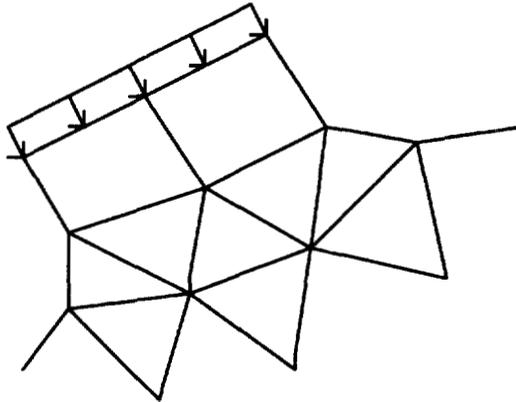
Figure 5-3   Sample Network near Inflow

RMA-2 does not permit inflows across straight line boundaries using continuity line input. That is Figure 5-4 below is illegal, if this input were attempted the flow directions at nodes n1 and n3 would be set as parallel *to* the line of the boundary (to eliminate leakage). This would generate very large components.
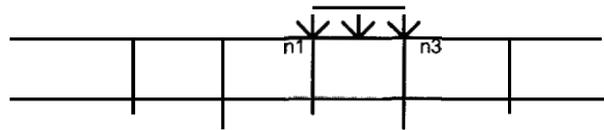
Figure 5-4   Illegally Distributed Inflow

3. Input the flows by using specified values at node points. Figure 5-5 shows examples of nodal inflows applied at midside and corner nodes. Note the difference in the distribution.

Figure 5-5   Legal Distributions of Nodal Flow

5.9.2   Modellinn of Flow across boundaries

Version 6.3 offers a new alternative for defining continuity line inflows. Data lines type QF allow the user to define flow across a boundary that does not form a nozzle. **A** typical application will be for distributed side flow into a river or estuary.
Figure **5.6** illustrates the way the flow is distributed along the line. Note that flow is set to zero at the end of each line. The flow in the direction normal to the boundary is not constrained. The data structure is identical to that used for type QC flow input. Total flow and a direction are defined. The user may optionally input dynamic flows using the hydrograph file.
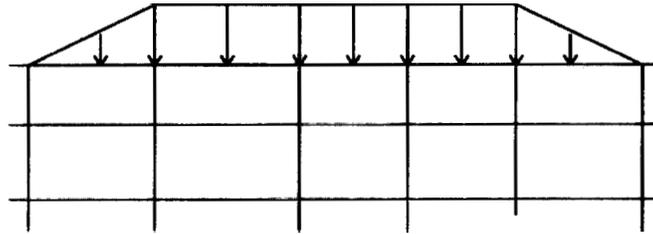
Figure 5.6 Distribution of flow for a type QF data set

6.   REFERENCES

King, I.P., (1970) *An Automatic Reordering Scheme for Simultaneous Equations Derived from Network Systems,* International Journal for Numerical Methods in Engineering, Vol. 2, pp. 523-533.

King, I.P., Norton, W.R. and Orlob, G.T. (1973), *A Finite Element Solution for Two-Dimensional Density Stratified Flow,* report prepared by Water Resources Engineers, Walnut Creek California, for the U.S. Department of the Interior, Office of Water Resources Research.

King,I.P. and Rachiele,R.R (1990), RMA-1, *A Program for Generation of Finite Element Networks, User Instructions,* report prepared by .Resource Management Associates, Lafayette California.

King,I.P. (1991), *Evaulation of Modelling Parameters for Simulation of Estuarial Systems,* proceedings of the 2nd ASCE Conference on Estuarine and Coastal Modeling.

King,I.P. (1992), RMAGEN, *A Program for Generation of Finite Element Networks, User Instructions,* report prepared by .Resource Management Associates, Lafayette California.

APPENDIX A

FINITE ELEMENT INTEGRALS **FOR** TWO−DIMENSIONAL DEPTH AVERAGED ELEMENTS

The finite element method is applied to the following two dimensional depth averaged equations, repeated here for completeness from section 3

Momentum Eauation X-Direction

$$\rho \left(h\frac{\partial u}{\partial t} + hu\frac{\partial u}{\partial x} + hv\frac{\partial u}{\partial y} + gh\left(\frac{\partial a}{\partial x} + \frac{\partial h}{\partial x}\right) + \frac{g}{C^2}u|V| + uq_s - \Omega vh \right) - h\frac{\partial}{\partial x}\left(\varepsilon_{xx}\frac{\partial u}{\partial x}\right)$$

$$- h\frac{\partial}{\partial y}\left(\varepsilon_{xy}\frac{\partial u}{\partial y}\right) - W_x = 0 \qquad\qquad\qquad A.1$$

Momentum Eauation Y-Direction

$$p\left(h\frac{\partial v}{\partial t} + hu\frac{\partial v}{\partial x} + hv\frac{\partial v}{\partial y} + gh\left(\frac{\partial a}{\partial y} + \frac{\partial h}{\partial y}\right) + \frac{g}{C^2}v|V| + vq_s + \Omega uh \right) - h\frac{\partial}{\partial x}\left(\varepsilon_{yx}\frac{\partial v}{\partial x}\right)$$

$$- h\frac{a}{\partial y}\left(\varepsilon_{yy}\frac{\partial v}{\partial y}\right) - W_y = 0 \qquad\qquad\qquad A.2$$

Continuitv Eauation

$$\left(h\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) + u\frac{\partial h}{\partial x} + v\frac{\partial h}{\partial y} + \frac{\partial h}{\partial t} = 0 \qquad\qquad\qquad A.3$$

where

x, y	=	horizontal Cartesian coordinates
t	=	time
u, v	=	the horizontal velocity components in the **x** and y directions respectively
h	=	depth
a	=	bottom elevation
$\varepsilon_{xx}, \varepsilon_{xy}, \varepsilon_{yx}$ and ε_{yy}	=	the turbulent eddy coefficients.
C	=	Chezy bottom friction coefficient
V	=	Total water velocity
q_s	=	Tributary flow into the system
Rvh and Ruh	=	The coriolis forcing in the **x** and y directions respectively.
W_x and W_y =		forces due to wind stresses in the **x** and y directions respectively.

Partial integration is applied for the viscous and depth terms. Terms are integrated over the horizontal plan area and along the line boundaries of the elements.

The following element residual vectors result.

Momentum Eauation X-Direction

$$f_x = \int_{A_h} N^T [\rho\,(h\frac{\partial u}{\partial t} + hu\frac{\partial u}{\partial x} + hv\frac{\partial u}{\partial y} + gh\frac{\partial a}{\partial x} + \frac{g}{c^2}u|V| + uq_s - Rvh - W_x)$$

$$+ \varepsilon_{xx}\frac{\partial u}{\partial x}\frac{\partial h}{\partial x} + \varepsilon_{xy}\frac{\partial u}{\partial y}\frac{\partial h}{\partial y}] + N_x^T [\varepsilon_{xx} h\frac{\partial u}{\partial x} - \rho g\frac{h^2}{2}] + N_y^T \varepsilon_{xy} h\frac{\partial u}{\partial y}\, dA \quad A.4$$

Momentum Equation Y-Direction

$$f_y = \int_{A_h} (N^T [\rho(h\frac{\partial v}{\partial t} + hu\frac{\partial v}{\partial x} + hv\frac{\partial v}{\partial y} + gh\frac{\partial a}{\partial y} + \frac{g}{c^2}v|V| + vq_s + Quh - W_y)$$

$$- \varepsilon_{yx}\frac{\partial v}{\partial x}\frac{\partial h}{\partial x} - \varepsilon_{yy}\frac{\partial v}{\partial y}\frac{\partial h}{\partial y}] + N_x^T \varepsilon_{yx}\frac{\partial u}{\partial x} h + N_y^T [\varepsilon_{yy}\frac{\partial u}{\partial y} h - \rho g\frac{h^2}{2}]\, dA \quad A.5$$

Continuity Equation

$$f_c = \int_{A_h} M^T [h(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}) + u\frac{\partial h}{\partial x} + v\frac{\partial h}{\partial y} + \frac{\partial h}{\partial t} - q_s]\, dA \qquad\qquad A.6$$

where A_h represents integration over the horizontal plan area

After differentiation the following integral derivatives form contributions to the finite element equations

Momentum Equation X-Direction

$$\frac{\partial f_x}{\partial u} = \int_{A_h} N^T [\rho(h\frac{\alpha}{\Delta t} + h\frac{\partial u}{\partial x} + q_s + \frac{g}{c^2|V|}\{2u^2 + v^2\})N + (\rho hu + \varepsilon_{xx}\frac{\partial h}{\partial x})N_x$$

$$+ (\rho hv + \varepsilon_{xy}\frac{\partial h}{\partial y})N_y] + N_x^T h\varepsilon_{xx}N_x + N_y^T h\varepsilon_{xy}N_y\, dA \qquad\qquad A.7$$

$$\frac{\partial f_x}{\partial v} = \int_{A_h} N^T [\rho(h\frac{\partial u}{\partial y} + \frac{g}{c^2|V|}uv - Qh)\, N]\, dA \qquad\qquad A.8$$

$$\frac{\partial f_x}{\partial h} = \int_{A_h} N^T [\rho(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} - \Omega v + g\frac{\partial a}{\partial x})N - \varepsilon_{xx}\frac{\partial u}{\partial x}N_x - \varepsilon_{xy}\frac{\partial u}{\partial y}N_y]$$

$$+ N_x^T [(\varepsilon_{xx}\frac{\partial u}{\partial x} - \rho gh)M] + N_y^T \varepsilon_{xy}\frac{\partial u}{\partial y}M\, dA \qquad\qquad A.9$$

Momentum Equation Y-Direction

The terms for the derivatives of f_y are symmetrical with those **off,** and will not be presented.

Continuity Equation

$$\frac{\partial \mathbf{f_c}}{\partial u} = \int_{A_h} \mathbf{M}^T \frac{\partial h}{\partial x} \mathbf{N} + \mathbf{M}^T h \mathbf{N_x} dA \qquad \qquad \text{A.10}$$

$$\frac{\partial \mathbf{f_c}}{\partial v} = \int_{A_h} \mathbf{M}^T \frac{\partial h}{\partial y} \mathbf{N} + \mathbf{M}^T h \mathbf{N_y} dA \qquad \qquad \text{A.11}$$

$$\frac{\partial \mathbf{f_c}}{\partial h} = \int_{A_h} \mathbf{M}^T [(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\alpha}{\Delta t}) \mathbf{M} + u \mathbf{M_x} + v \mathbf{M_y}] dA \qquad \text{A.12}$$

SMAGORINSKY TURBULENCE CLOSURE

An option of the model permits a version of the Smagorinsky closure for horizontal eddy viscosity to be applied in the model. That is when IEDSW = 2 on the CO line of the input data file.

As applied in RMA-2 the depth integrated turbulent eddy viscosity terms are given the following forms

For the x-momentum equation:

$$\frac{\partial}{\partial x}\left(2\,A_m\,h\,\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(A_m\,h\,\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right) \qquad\qquad \text{B.1}$$

For the y-momentum equation:

$$\frac{\partial}{\partial y}\left(2\,A_m\,h\,\frac{\partial v}{\partial y}\right) + \frac{\partial}{\partial x}\left(A_m\,h\,\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right) \qquad\qquad \text{B.2}$$

where:

$$A_m = \alpha\,(\text{area})\,\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + 0.5\,\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2\right]^{0.5} \qquad \text{B.3}$$

a = a coefficient usually in the range 0.01 - 0.5. In RMA-2 this value *is* an input parameter

area = the area of the current element.

In the Newton Raphson implementation of the finite element method formal derivatives of the equations are taken with respect to the dependent variables u, v, and h. This method is applied for all the contributions to the terms above including A.

Because it is possible for A_m to take on a zero value when the velocities are zero the model has been coded to automatically use data on the type "ED" line as if IEDSW = 1 for computation of values for the first two steady state or dynamic iterations if no initial conditions have been input from a file.

As an example terms will be derived for the momentum equation in the x-direction

$$f_x = \int N^T\left[\rho\left(h\frac{\partial u}{\partial t} + hu\frac{\partial u}{\partial x} + hv\frac{\partial u}{\partial y}\right) + gh\frac{\partial a}{\partial x} + \frac{g}{C^2}u|V| + uq_s - \Omega vh - W_x\right] + N_x^T\,2\,\rho\,A_m\,h\frac{\partial u}{\partial x} + A_h$$

$$N_y^T\,\rho\,A_m\,h\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\,dA \qquad\qquad \text{B.4}$$

The differentials are **as** follows:

$$\frac{\partial \mathbf{f_x}}{\partial u} = \int_{A_h} \mathbf{N^T}[\rho(h\frac{\alpha}{\Delta t} + h\frac{\partial u}{\partial x} + q_s + \frac{g}{C^2|V|}\{2u^2 + v^2\})\mathbf{N} + (phu)\mathbf{N_x}$$

$$+ \rho hv\,\mathbf{N_y}] + 2\rho h\,\mathbf{N_x^T}[A_m\,\mathbf{N_x} + \frac{\partial u}{\partial x}(c1\,\mathbf{N_x} + c2\,\mathbf{N_y})]$$

$$+ \rho h\mathbf{N_y^T}[A_m\mathbf{N_y} + (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})\,(c1\,\mathbf{N_x} + c2\,\mathbf{Ny})]\,dA \qquad\qquad \text{B.5}$$

$$\frac{\partial \mathbf{f_x}}{\partial v} = \int_{A_h} \mathbf{N^T}[\rho(h\frac{\partial u}{\partial y} + \frac{g}{C^2|V|}uv - Rh)\,\mathbf{N}] + \mathbf{N_x^T}\,2ph\frac{\partial u}{\partial x}(c3\,\mathbf{N_x}$$

$$+ c4\,\mathbf{N_y}) + \rho h\mathbf{N_y^T}[A_m\mathbf{N_x} + (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\,(c3\,\mathbf{N_x} + c4\,\mathbf{N_y})]\,dA \qquad\qquad \text{B.6}$$

$$\frac{\partial \mathbf{f_x}}{\partial h} = \int_{A_h} \mathbf{N^T}[\rho(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} - \Omega v + g\frac{\partial a}{\partial x})\mathbf{M}$$

$$+ \mathbf{N_x^T}[2\rho A_m\frac{\partial u}{\partial x} - \rho gh)\mathbf{M}] + \mathbf{N_y^T}\,\rho A_m(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})\,\mathbf{M}\,dA \qquad\qquad \textbf{B.7}$$

where

$$\frac{\partial A_m}{\partial u} = (c1\,\mathbf{N_x} + c2\,\mathbf{N_y}) \qquad\qquad \text{B.8}$$

$$\frac{\partial A_m}{\partial v} = (c3\,\mathbf{N} + c4\,\mathbf{N_y}) \qquad\qquad \text{B.9}$$

and

$$c1 = a\,(area)\frac{\partial u}{\partial x}[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + 0.5\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2]^{-0.5} \qquad\qquad \text{B.10}$$

$$c2 = a\,(area)\,0.5\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})\,[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + 0.5\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2]^{-0.5} \qquad\qquad \text{B.11}$$

$$c3 = a\,(area)\,0.5\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})\,[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + \textbf{0.5}\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2]^{-0.5} \qquad\qquad \text{B.12}$$

$$c4 = a\,(area)\frac{\partial v}{\partial y}[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + 0.5\,(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2]^{-0.5} \qquad\qquad \text{B.13}$$

APPENDIX C

MARSH ELEMENTS / FLOODING AND DRYING

The marsh element or equivalent porosity formulation has been added to RMA-2 to improve performance when simulating areas the flood and dry during the tidal cycle or flood event.
Historically the first approach used for simulation of these types of system was to automatically drop from the system any element where any **one** corner water depth dropped below a nominal minimum value (an input parameter). This technically works. The problems found were:

- Inconsistent performance when irregular boundaries resulted.
- Element dropped out too soon just because one corner was notionally dry.
- Poor convergence at some times when elements cycled in and out the system.
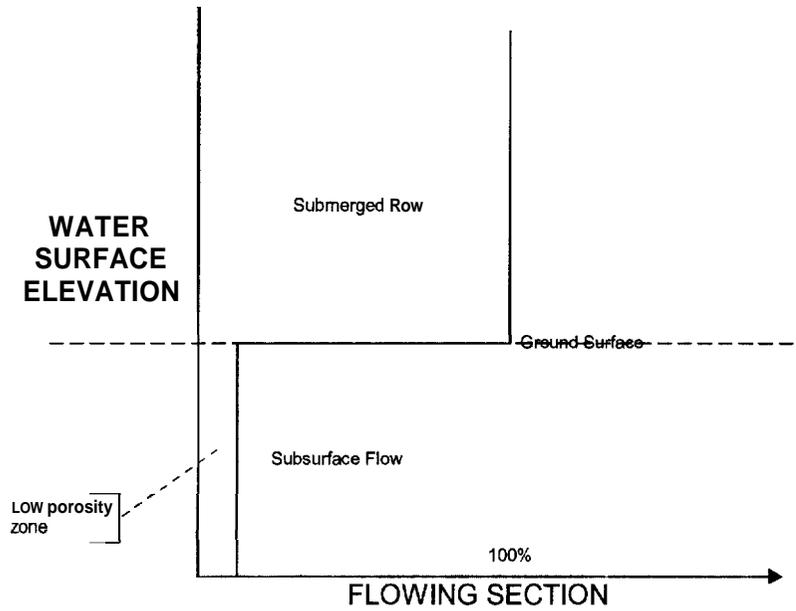- When an element was dropped or added the total water stored in this element was removed or added.

For these reasons an improved method was sought. Preferable a method where elements dropped out steadily (in a sense faded away) and would only be removed when **all** nodes were below the minimum.

The equivalent porosity/marsh method is the result. Conceptually the system was considered as an integration of both surface waters and subsurface groundwater. When the water surface elevation is below the ground surface flow is presumed to occur in the low porosity groundwater zone. However the governing equations are still the shallow water equations. The figure illustrates this concept. **As** a practical consideration the transition in porosity at the ground surface could no occur with a the infinite gradient shown. Instead an approximate system was conceptualised where porosity changed over a finite range as shown. This range has some physical basis. For example in sandy beach areas and overbank flood plains there are frequently uneven sections, in marshes there are small channels that are below the level of discretisation.
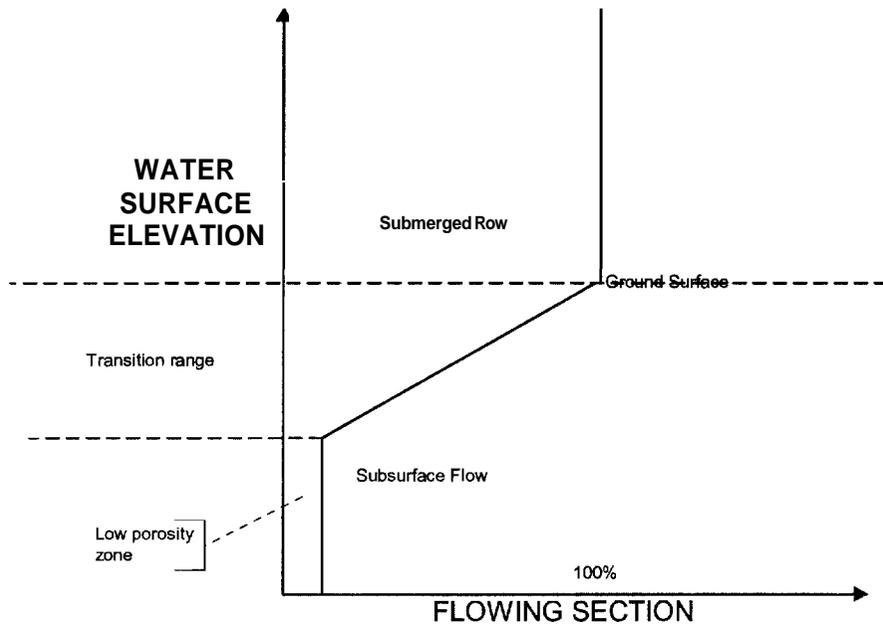
In order to assure mathematical consistency and not create water at any time the actual level of the transition must be slightly adjusted and the bottom elevation of the groundwater section had to be kept finite. The elevation of the groundwater section is thus best set to be just above the anticipated low water level.

The data requirements for the equivalent porosity formulation thus include:

- The range in feet or metres over which the element transitions from a fully flowing system to the fractional effective flowing section defined by the minimum porosity.
- The value of this minimum porosity.
- The depth below the transition over which the equivalent porosity is allowed to exist.

WATER SURFACE ELEVATION

Submerged Row

Ground Surface

Subsurface Flow

LOW porosity zone

100%

FLOWING SECTION

Idealized Representation

WATER SURFACE ELEVATION

Submerged Row

Ground Surface

Transition range

Subsurface Flow

Low porosity zone

100%

FLOWING SECTION

Approximate Representation

<u>Modellina Conseauences</u>
The model incorporates this process by formal integration into the differential equations. Without going into full details, the result is that the model operates with a transformed equivalent depth. When the water surface is above the transition level this equivalent depth is identical to the conventional depth and the water surface elevation is equal to the bottom elevation plus the depth. Below the start of the transition this relationship no longer holds. The depth decreases at a smaller rate than the water surface, or in other words the effective bottom moves down as the depths falls The figure below illustrates this concept. Note that the curve below is a mathematical result and there are no consequences from the perspective of overall flow conservation.
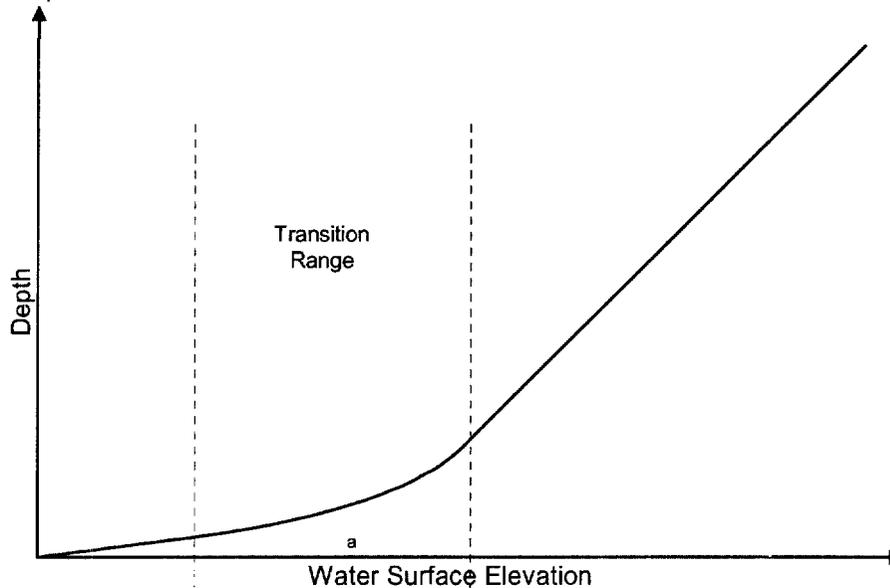
WATER SURFACE VS DEPTH FOR THE MARSH APPROXIMATION

From a numerical perspective the greatest difficulty and cause of instability for this method is the sharp break in gradient as the depth decreases through the transition. This may be demonstrated by observing that the Newton Raphson correction to depth when a point in the system is fully submerged or low in the porous zone will not mathematically expect the transition and over-correction can result. The model has been modified to automatically switch to under-relaxation for these cases. Model convergence can be significantly slowed by these conditions.

RMA-2 has an option, input controls that allow elements that have water depths entirely in the porous zone, to be dropped from the system. This option should be used with caution because the resulting irregular boundaries can cause local instabilities.
<u>Data Input</u>
The data requirements for the model are nodal distributions of:
• Depth below the transition to groundwater flow.
• The transition range.
• The porosity of the groundwater flow.

     In practical input terms the model allows the data to be input either as the default values, a revised global terms, or as distributed individual nodal values.

Actual values to use for the model are sensitive to the conditions and scale of the prototype system.