Environnement Canada Environment Canada

RIVICE Model – User's Manual

Environment Canada

RIVICE Steering Committee

January 2013

DISCLAIMER

The RIVICE software has been developed methodically and tested as thoroughly as practical. However, it is not possible to protect against all possible malfunctions of the code, especially those that may be caused by use of the software in a way that was not intended, or for river scenarios that differ from the tested cases. The users must apply this software at their own risk.

ACKNOWLEDGEMENTS

The final manual revisions have been completed through financial support supplied by Environment Canada. The work was completed by Maurice Sydor, M.Sc. P.Eng., S.E.S. Systems Evaluation Service Inc., formally head of the numerical modelling group which undertook complex model design and development and R.W. Carson, P.Eng. P.E., M.Sc.C.E, Senior Consultant KGS Group.

Original sponsors included members from the Canadian Coast Guard, Canadian Electrical Association, Indian and Northern Affairs, Manitoba Hydro, New York Power Authority, Ontario Hydro, Quebec Hydro, Saskatchewan Power, Swedish Power Authority, US Power Authority and Environment Canada. The above agencies provided their guidance through the establishment of the RIVICE Steering Committee which in turn directed funding and maintained control of the project progress.

Due to the re-organization within the Department, any manual or model errors, omissions and update requests may be addressed to Maurice Sydor at maurice.sydor@videotron,ca. The ice routines, ice theory and equations used in the model or any related questions may be addressed to Rick Carson at RCarson@kgsgroup.com.

TABLE OF CONTENTS

PAGE

1. Introduction 1
1.1 About This Manual 1
1.2 History of RIVICE 2
1.3 Objectives of RIVICE 2
1.4 installation instructions 3
1.5 COMPILATION PROCEDURES 4
1.6 Data Formats for ASCII Input Files 7
2. Software Structure And Theory 8
2.1 Software Overview 8
2.2 Rivice Description and Basic Theory 16
3. Software Strengths and Weaknesses 31
3.1 Strengths re: ice Simulation 31
3.2 Limitations re: ICE Simulation 31
4. Hardware Software Installation 33
4.1 Hardware Requirements 33
4.2 Installation Requirements 33
4.3 Compilation Procedures 34
4.4 Data Formats 37
5. RIVICE Modelling Procedure 39
5.1 General 39
5.2 Components of a RIVICE Model 39
5.3 Reaches & Nodes 39
5.4 Initial Conditions 40
5.5 Boundary Conditions for Hydraulic Calculations 41
5.6 Boundary Conditions for Ice 42
5.7 Lateral Inflows 42
5.8 Selection of Time Steps 42
5.9 Selection of Mesh Spacing 43
6. Pre-Processing 45
6.1 X-Section (KPA Version) 45
6.2 COORD1 46
7. Input Requirements 49
7.1 RIVICE Input File: TAPE 5 49
7.2 Notes on Selection and Input of Ice Parameters 102

8. RIVICE Output Files ... 107
8.1 Structures / Files ... 107
9. A Recent Case Study ... 111
9.1 introduction ... 111
9.2 study site ... 111
9.3 lower red river ice ... 113
9.4 breakup of ice along the lower red river ... 113
9.5 Justification for computer modelling using RIVICE ... 118
9.6 Data for model setup ... 119
9.7 results and discussion ... 122
9.8 Conclusions ... 130
9.9 References ... 131
10. Building and De-Bugging a RIVICE Model ... 133
10.1 Development Phase ... 133
10.2 Planning Hints/Tips ... 133
10.3 De-Bugging Hints - General Hydraulic Conditions ... 134
10.4 Debugging Hints – Ice Conditions ... 137
11. Example Simulations of Ice Conditions ... 139
11.1 Simple Hypothetical Channel ... 139
11.2 Thames River Ice Jam ... 141
12. Modifications by User ... 143
12.1 Guidelines ... 143
13. Potential Future Improvements ... 149
13.1 Graphical Assistance for User ... 149
13.2 Other Desirable Improvements ... 149
14. References ... 150
Appendix 1 Glossary of Terms ... 154
Appendix 2 Theoretical Background and Computational Methods for Hydraulics / Hydrodynamics ... 156
A2.a Theoretical Considerations ... 156
A2.b Theoretical Considerations ... 160
Appendix 3 Ice Processes - Theoretical Background and Computational Methods ... 170
A3.a Computational Methods Used to Represent Ice Processes ... 170
A3.b Hydraulic Resistance of River Ice Covers ... 174
Appendix 4 User Specified Parameters With Default Values ... 180
A4.a The Following Is a List of All the Default Values Currently Used in RIVICE ... 180

1. INTRODUCTION

1.1 ABOUT THIS MANUAL

This Manual contains instructions for the application of the River Ice (RIVICE) Computer Program and associated software on a FORTRAN compatible operating system such as Microsoft XP. This document combines and updates several earlier manuals for the hydraulic simulation program for One-Dimensional Hydrodynamics (ONED) prepared for Environment Canada and later modified for application to various large river or delta systems. It also provides examples and guidance for program application to several typical situations. This Manual does contain a temperature parameter from the earlier water quality work but does not have a sediment modelling capabilities, which are contained as routines within other versions of the ONE-D Program.

Further independent application was carried out by Manitoba Infrastructure and Transportation on two project sites. The first was for the assessment of and proposed mitigation by future dredging of the lower Red River near the mouth of Lake Winnipeg[1] described in detail in Appendix 5. The second project included the assessment and construction of protective dikes at the mouth of the Dauphin River which was to experience flows nearing 2.5 times those previously recorded. These studies were carried out with the associated results published in 2010-11[2]. The studies providing confirmation that the numerical tool performed well and resulted in establishing critical ice generated high water levels for final dike design and construction as well as providing levels for increased road grade following the river to the community dike section at the mouth of the Dauphin River.

It is assumed that users of this RIVICE Manual have had experience and training in dynamic open channel hydraulics, a basic understanding of river ice engineering, and some basic familiarity with numerical modelling and FORTRAN-based computing systems. Although not essential, some exposure to the FORTRAN programming language and procedures for compiling source code would greatly facilitate application of this software.

Volume 1 covers the detailed steps necessary to make each of the programs run. The procedures for installation and compilation are described in the earlier hydraulic manual[3], and the data requirements for each input file or command line prompt are outlined. Some guidance regarding application techniques are also included.

Volume 2 contains a series of test cases of the hydraulic routines[4]. The test routines include example input files, results of tests and comparisons of test routine output with data from independent hydraulic sources.

A CD containing all the source code files, accessory programs, and the test routines accompany this manual. Listings of the files included on the CD appear in Appendix 3.

---

[1] Ice jam modelling of the Red River in Winnipeg, Karl-Erich Lindenschmidt, Maurice Sydor, Richard W. Carson, 16th CRIPE Workshop on the Hydraulics of Ice Covered Rivers, Winnipeg, September 2011, pp. 274-290

[2] Modelling ice cover formation of a lake-river system with exceptionally high flows (lake St. Martin and Dauphin River, Manitoba), Karl-Erich Lindenschmidt, University of Saskatchewan, Maurice Sydor, Environment Canada, Richard W. Carson, KGS Group, Cold Region Science and Technology 82 (2012) 36-48

[3] ONE-D Hydrodynamic Program User's Manual, Volume 1, Environment Canada, Water Issues Branch; B.C. Environment, Water Management Division, March 1995

[4] ONE-D Hydrodynamic Program User's Manual, Volume 2, Environment Canada, Water Issues Branch; B.C. Environment, Water Management Division, March 1995

Suggestions for program improvements to meet current modelling needs that the RIVICE program cannot handle are welcomed by the program developers at Environment Canada. The contact person would be Head, Numerical Modelling and Analyses Section, Sustainable Water management Division.

The authors of this manual, KGS and Systems Evaluation Service Inc., wish to acknowledge the valuable assistance extended by Environment Canada's Numerical Modelling and Analyses Section, Dr. Spyros Beltaos of Environment Canada and by the Manitoba Infrastructure and Transportation.

1.2 HISTORY OF RIVICE

In late 1989, a consortium of consulting engineering firms prepared an unsolicited proposal for the development of a non-proprietary hydro-dynamic numerical model of river ice processes. A group of client companies and agencies formed a Steering Committee to administer the process of program development. A "design report" was submitted in 1992 that outlined the basic intent of the model, and after discussions with the Steering Committee, preparation of programming instructions that would define the implementation of the new software commenced. A subsequent report was submitted that included flowcharts and detailed line-by-line programming instructions. The development of the program code was undertaken by the programming group at Environment Canada. Individual subroutines were developed first, and were tested with sample input. However, the amalgamation of the entire program proved to be difficult and by 1995, a successfully working model had not yet been developed. At that point, the project went into abeyance, although some funds remained in the pooled financial support that had been raised for the project by the client organizations.

In 2003, KGS Group was approached by the Head of the Steering Committee at Environment Canada to complete the software. The choice was based upon the fact that a very knowledgeable member currently with the KGS Group's lead engineers had participated in the original development work, and had a working knowledge of the designed program.

The development by KGS Group was initially slow, primarily due to the desire and need to conserve budget by following a methodical strategy of development. However, by late 2008, the concept was essentially proven to be workable and several test cases had been successfully completed.

The version of RIVICE that emerged from this process was similar in functionality to what had been envisaged by the original developers, but differed in one major aspect. That was the abandonment of a separate control model called the "driver" for the program, and resulted in the adoption of the driver logic that was already available and was embodied in the original "ONED" hydrodynamic model software.

1.3 OBJECTIVES OF RIVICE

The initial RIVICE concept was a significant undertaking and not all of the objectives that had been originally envisioned for RIVICE in 1989 could be achieved. This was primarily due to the limitations of budget and the occurrence of difficulties that had not been anticipated in the 1990's or in the planning of the resumption of program development. On the other hand, some additional capabilities were developed that had not been contemplated in the original plans. These new developments are considered to be substantial improvements in the program capability. In addition, the program was restructured to permit, as much as practical, the ability of future investigators to insert their own subroutines in order to undertake specific tasks in the modelling, rather than to use the methodology that has been provided as an initial default.

The basic objectives of RIVICE are to:

- Permit the simulation of river ice cover development, primarily in swift rivers that do not permit the orderly formation of a smooth, stable, thermally developed ice cover like that on in a more stationary situation such as a lake;
- Apply the most current understanding of the ice processes involved, and allow the user the flexibility of choosing optional means of representing the ice development;
- Permit representation of the hydrodynamic effects of changing flows and the influence of the ice cover evolution on the changes in river flow;
- Provide a framework for the insertion of additional user developed subroutines that could be adopted in lieu of the logic that has been developed for the initial RIVICE program. Specific examples would be:
  - heat loss from open water;
  - rate of ice generation, or rate of ice cover melting when the incoming water temperature exceeds zero degrees Celsius;
  - criteria for the stability of ice approaching the leading edge of an ice cover;
  - provide for the mechanism that governs whether an ice entrained in the flow beneath an established ice cover will deposit on the ice under surface, or continue to be transported downstream with the flow;
  - border ice formation and breakup;
  - estimation of roughness characteristics of the under surface of the ice cover;
  - estimation of transport characteristics of entrained ice within the flow under an ice cover;
  - calculations of the ability of a fragmented ice cover to resist the hydraulic loads imposed upon it, and thickening to become stable;
  - and allow for a variable ice-force shedding parameter to account for the impact of islands, bridge piers, channel meander and near-shore heat sources that can significantly change this parameter.

1.4 INSTALLATION INSTRUCTIONS

Basic steps in using the model require the user to initially copy the program files to the computer of choice and then using the steps provided in the earlier manuals or developing other means to create the data inputs needed to run the model.

This section applies to installation of the ONE-D software that was compiled using the LAHEY FORTRAN Compiler, which is recommended for this purpose. Other FORTRAN compilers can be used but must have some have varying degrees of error detection and others follow coding rules that not necessarily provided in all commercial FORTAN compilers. For the installation of the ONE-D software that will be compiled using either the IBM Professional FORTRAN Compiler, or the Microsoft FORTRAN Compiler, please refer to their respective manuals provided with these compilers.

For a typical LAHEY installation of the FORTRAN and BASIC compilers, please refer to the respective manuals supplied for these programs. To install the ONE-D program and support software, proceed as follows:

1. Copy all the contents of the CD marked "PROGRAM SOURCE CODE" to a directory on the hard disk. It is assumed that this directory will be named C:\1D.
2. The data on the CD marked "TEST ROUTINES" should be copied to a separate directory, such as C:\1D\LIBTEST.

3. Assuming the executable programs created by the compiler will be stored in the C:\1D directory, make a PATH to this directory in the AUTOEXEC.BAT file located in the root directory.

An alternative arrangement which would separate the executable and source files would be to keep all the executable files, and the two LAHEY files RUN386.EXE and F77L.EER, in the C:\1D directory after compilation, and to move the source code files to a directory such as C:\1D\SOURCE\.

1.5 COMPILATION PROCEDURES

The following instructions pertain to compiling RIVICE and associated FORTRAN software using the LAHEY F77L EM/32 FORTRAN Compiler. For compilation using either the IBM Professional FORTRAN Compiler, or the Microsoft FORTRAN Compiler, please refer to the manuals provided with these compilers. For older versions of these compilers, it may be useful to also refer to the February 1988 document entitled, "One-Dimensional Hydrodynamic Model Computer Manual," by Environment Canada. The LAHEY compiler was one of the first to create executable programs that access extended memory, allowing them to exceed 640 Kb in size. This is a typical requirement for large models.

Some of the earlier programs in the ONE-D family, including BUILD, BROWSE, EXPORT and the most recent version of CD1PLOT will only work if compiled using the LAHEY FORTRAN compiler, as the LAHEY software has screen graphics capability that the others may not. Users are also warned that in the instances where one program creates a binary file that will be used as input into another program, both programs must be compiled with the same compiler, since the format structure for reading and writing binary files can vary from one compiler to another (word size or bits), and use of different compilers in such cases will likely lead to program execution failures or errors. For simplicity it is recommended that the LAHEY or initially chosen compiler be used throughout.

Users are cautioned that the current version of the RIVICE program has been extensively tested on LAHEY Fortran compiler mainly. To verify program performance on other compilers, the standard test data sets should be executed and the output checked against the standard test results.

To handle extremely large datasets prior to compiling the main RIVICE program, the embedded hydraulic ONE-D source code may require modification to increase the values of 23 variables used to define array sizes in the executable program. The procedure to define these variables is described in the subsection below entitled "Dimensioning Arrays in the ONE-D Source Code."

To compile any one of the FORTRAN source code modules, identified by the extension .FOR in the filename, proceed as follows:

1. Working from the C:\1D directory, with the source code file, <file>.FOR, present in the same directory, and with a path to the LAHEY compiler directory (set by the LAHEY install program as C:\F77L3\BIN), type the following at the DOS prompt:

```
F77L3 <file>.FOR↵
```

where <file> is the name of one of the following source code files to be compiled:

COORD1 (CD1X_d.FOR); **COORD2** (CD2PGM_b.FOR) and **RIVICE (**Rivice_Aug6_11d.FOR)

After the compiler responds with a listing of all subroutines compiled and any warnings, the system reverts to the DOS prompt. This will create several files with the extensions .OBJ, .MAP, .LST and .SLD in the working directory.

When compiling any of the source code modules, make sure that the BOUNDS checking compiler configuration option is set (i.e. /B is specified on the command line or in the "F77L3.FIG" file). This is the checking command to determine and flag any arrays that have been exceeded and that have to be reset. This is specifically important to find arrays that have been exceeded and will result in erroneous answers generated in the simulation process.

2. Upon completion of Step 1 for the programs CD1X, CD2PGM, and RIVICE, type the following:

```
386LINK <file>↵
```

For the programs CD1PLOT ensure the LAHEY graphics file GRAPH3.LIB resides in the directory named C:\F77L3\LIB (created by the LAHEY installation program) , and type the following:

```
386LINK <file> -LIB \F77L3\LIB\GRAPH3↵
```

Upon completion of this step the executable file named <file>.EXE will be created.

DIMENSIONING ARRAYS IN THE RIVICE / ONE-D SOURCE CODE

The RIVICE program on the CD provided with this Manual is in the form of a source code master file. The array dimensions must be redefined should the specific array become exceeded during model application or development.

These variables are listed here so as to provide the user with the understanding of internal interrelationships of variables which must be changed according to model size. The ability to redefine these variables allows the user flexibility to match a particular modelling size to the memory available on any available computer.

The explanation of these variables and their associated dimensions are provided below for a better understanding of the specific 23 variables and an example is provided in Table 1-1 below:

Table 1-1

Description of Variables Determining Array Sizes

Variable Number	Description of Variables
1	Maximum total numbers of hydraulic mesh points (interpolated channel cross sections) in a network.
2	Maximum total number of hydraulic table entries (number of rows in the table) for interpolated channel cross section data. This is the sum for all reaches of the number of table entries per cross section times the number of mesh points per reach.
3	Maximum total number of water quality mesh points (cross sections) in a network. (Set to 1 if only hydraulics option is used.)
4	Maximum total number of reaches in a network.
5	Maximum total number of lateral inflows in a network.
6	Maximum total number of individual discharge values for all lateral inflows (including QFCS, which each count as two lateral inflows).
7	Maximum number of individual discharge or water level values for hydraulic boundary conditions.
8	Maximum number of table entries for water quality boundary conditions (set to 1 if water quality option not used).
9	Maximum number of injection points (set to 1 if water quality option is not used).
10	Maximum number of table entries for injection points (set to 1 if water quality option is not used).
11	Maximum number of water quality parameters. The program currently has 11 parameters, such as salinity, temperature, BOD, DO, etc. (Set to 1 if only hydraulics option is

used.)

12	Maximum number of hydraulic mesh points per reach.
13	Maximum number of water quality mesh points per reach. (This must be set to 2 if only hydraulics option is used.)
14	Maximum number of nodes (at least one greater than the maximum number of reaches).
15	Maximum number of time graphs (hydro or water quality) to be listed in TAPE6.TXT.
16	Maximum number of profiles to be listed in TAPE6.TXT.
17	Maximum number of table entries for meteorological conditions (set to 1 if only hydraulics option is used).
18	Maximum number of elements in banded node matrix = (2 x number of reaches + number of nodes) 2.
19	Maximum number of quasi-dynamic flow control structures (QFCS).
20	Maximum pump station QFC number.
21	Maximum number of pumps per station.
22	Maximum number of head-discharge pairs used to define the pump curve.
23	Reserved for future use. Respond by entering a “1”.

An example set of values used for these variables, taken from the large Serpentine-Nicomeckl model, is listed below:

Table 1-2
Example Set of Array Dimensions

Variable	Value	Variable	Value	Variable	Value
1	4400	9	1	17	1
2	88000	10	1	18	600625
3	1	11	1	19	250
4	250	12	50	20	250
5	450	13	2	21	10
6	3024	14	275	22	12
7	8000	15	100	23	1
8	1	16	500		

The size of the executable ONE-D program resulting from the use of these array dimensions and the LAHEY compiler was 7.88 Mb.

Table 1-3
Dimensions of Key Variables for Ice Parameters

Variable Name or Type	Description	Dimension in RIVICE
NSEG, NSEGT	Number of ice segments with leading and trailing edges	5
NBRGSW	Maximum number of ice bridges that can be initiated by the user	5
USICEVOL	Maximum number of specified ice inflow volumes	50,000
VOLUMEICE	Number of ice volumes computed (one per time step)	50,000
DISTTRIGGER	Number of ice front locations for which output is requested	20
TIMETRIGGER	Number of specific time steps for which output of ice information is requested	20
Various parameters and ice information for cross sections	Controlled by the maximum number of cross sections	9001

1.6 DATA FORMATS FOR ASCII INPUT FILES

All of the ASCII input files require that the data is provided in a fixed format. If the input data does not conform to these formats, read input errors will most likely result. The specific format for each variable is defined in the file structure descriptions listed in this manual. The formats are coded using three standard FORTRAN format codes, which are summarized below:

Type	Format	Example of Format	Example of Number
Integers	Iw	I6	_ _ _ 366
Floating-point numbers	Fw.d	F10.2	_ _ _ _863.40
Character strings	Aw	A4	RUN1

In all the cases above, "w" represents the width (expressed as a number of spaces), of the field in which the data is to be placed. I-format numbers are right justified within the field. F-format numbers are positioned about the decimal point, which must fall anywhere within the field. The variable "d" represents the number of digits expected to the right of the decimal point. When input data is read in F-format and the number specified for d is not consistent with the actual position of the decimal point in the data, the position of the decimal point governs. A-format specification results in data being read as a character string with each string having w characters in width.

A number placed before any of the formats described above represents a multiplier. For example, 3I4 means three consecutive fields of I4 format.

2. SOFTWARE STRUCTURE AND THEORY

2.1 SOFTWARE OVERVIEW

The RIVICE software consists of primarily two pre-processing programs, a simulation program that consists of a main program and a series of individual subroutines that perform specific functions in the analysis, and a group of post-processing programs.

The key programs and their purposes are:

PRE-PROCESSING

The pre-processing programs were initially developed to generate hydraulic parameter elevation tables that are used in the dynamic hydraulic computations. These tables are used to reduce the computation time in the finite-difference solution scheme. By simple interpolation, the intermediate values are obtained quickly which in turn reduces computer computation time. The HEC programs derive their hydraulic values by processing each set of cross-section data points. A comparison was made using this method and the tables and it was found that the computation computer time increased by two-orders of magnitude.

The program called COORD1 (CD1X_d.FOR) and COORD2 (CD2PGM_b.FOR) are described as follows:

- **COORD1** prepares X-Y cross section data for plot verification and for further processing to generate model Data group B in the **TAPE5.TXT** file generated by the **COORD2** program**.**
- **COORD2** prepares the three table elevation data of hydraulic parameters required by ONE-D component of RIVICE derived from field or map extracted cross sections. This 3-elevation dataset of generated data is used to compute the hydraulic information and is constantly changed to reflect rapid variation in stage as well as the changes due to ice.
- **COORD1** is run a second time with the parameter **INTPL** set in the input dataset to generate all the interpolated sections used by the ice routines. The previous cross section source is input to the **COORD1** program and generates the **INTPSX.TXT** file to be used by RIVICE ice routines.

SIMULATION

- **Main – "driver"** logic that calls subroutines and manages the simulation process
- **RIVICE / ONED Subroutines** are not all used due to the fact that many have been meant for hydraulic computation options not currently available or programmed to be used with the **RIVICE** ice computation code.
    - **AMATRX**
        - Computes cross sectional area, hydraulic radius, friction slope at a defined water level
    - **ARPERK**
        - Computes the area, wetted perimeter and top width of a cross section
    - **BITRI**

- A solver routine used to compute the bi-tri diagonal matrix computation

- **BOOK**
  - Book keeping of all reaches and nodes entering or leaving a model node
- **BOUND**
  - Routine to setup coefficient matrix of all boundary nodes
- **BREACH**
  - A control structure to simulate a breach in a dike (not used in RIVICE)
- **CIRC**
  - Routine to compute circular pipe or culvert hydraulic tables (not used in RIVICE)
- **QF31A**
  - An equation used to estimate flow type in a culvert (not used in RIVICE)
- **CNTD2**
  - Routine to compute bi-direction flow through a culvert (not used in RIVICE)
- **CONSRV**
  - A parameter in the conservation of mass of conservative substances equation (not used in RIVICE)
- **CQT8**
  - A coefficient in the analyses of type 5 pipe flow hydraulics (not used in RIVICE)
- **CT7**
  - A coefficient in the analyses of type 4 pipe pressure flow (not used in RIVICE)
- **DECAY**
  - A routine to compute decay of non-conservative water quality substance (not used in RIVICE)
- **DEVICE**
  - A routine to setup input/output files according to internal integer numbers
- **DODSR**
  - A routine to compute re-aeration for water quality computations (not used in RIVICE)
- **DYKE**
  - A weir flow equation for flow over dikes (not used in RIVICE)
- **D2F28**
  - A function to compute a coefficient for rectangular culverts (not used in RIVICE)
- **FACTOR**
  - A routine used to compute water quality factors
- **FLUX**
  - An interpolation routine to compute meteorological conditions to the current model time
- **FOX**
  - A function used to initialize the dissolved oxygen level
  -
- **FOXF**
  - A function used to compute the dissolved oxygen level

- **GRIND**
  - A routine to re-compute the hydraulic computation tables each time the ice changes or is initiated
- **HGRAPH**
  - A routine to output time series hydrographs at the locations requested by the user in the **TAPE5** data input
- **HLATQ**
  - A routine to compute the variable lateral inflows provided as either actual flows or as controlled by pumping stations or physical structures such as a weir or an aboiteau
- **HYDPARM**
  - A routine used to change the physical cross section tables with the computed ice conditions either as surface or anchor ice
- **INITHY**
  - A routine which establishes the initial conditions for the hydraulic computations
- **INITWQ**
  - A routine which establishes the initial conditions for the water quality computations
- **INPUT1**
  - A routine that reads in the **TAPE5** reach input data related to hydraulic and water quality parameters
- **INPUT2**
  - A routine that reads in the **TAPE5** variable boundary input data related to hydraulic and water quality parameters
- **INPUT3**
  - A routine that reads the boundary data in the Water Survey of Canada typical format
- **IRREG**
  - A routine used to interpolate the irregularly spaced surveyed or generated cross sections to the fixed space sections used in the finite-difference computation scheme
- **KFF31B**
  - A function used to determine the hydraulic state in a culvert
- **KRECT**
  - A function to compute the conveyance factor in a rectangular culvert
- **NUTIN**
  - A routine to read and compute the nutrient values used in water quality simulations
- **NUTSR2**
  - A control routine to call specific water quality input data routines based upon the request water quality simulation requested
- **LATJEC**
  - A routine to read the lateral inflow water quality data
- **MLATQ**
  - A routine to read the lateral inflow water quality data and assign the space required for the simulation computations

- **OTHER**
  - A routine used to compute dissolved oxygen based upon BOD/DO parameters
- **OUTL**
  - A routine to output water quality computed data
- **OUTRTN**
  - A routine that generated the hydrographs and profiles requested in **TAPE5** input dataset
- **PARAM**
  - A subroutine to compute the matrix equations for the boundary nodes
- **PGRAPH**
  - A subroutine to output water quality profiles
- **REACH**
  - A routine to compute water quality matrix equations at internal nodes
- **RETRAP**
  - A routine to compute table values for a rectangular trapezoid cross section reach
- **SEARCH**
  - A function to find an internal node number from a user defined **TAPE5** node number
- **SOLVE**
  - A routine to solve the bi-tri-diagonal matrix
- **SOLVER**
  - A modified GELB SPP routine that executes the matrix solution
- **SOURCN**
  - A routine that computes the nutrient source load and decay coefficients
- **STORE**
  - A routine to store the banded matrix for both the hydraulic and water quality models
- **TABPAR**
  - A routine that interpolates fixed interval data using a three-degree parabolic method
- **TEMIN**
  - A routine to input temperature and update all associated reaches
- **TRISOL**
  - A routine that computes the matrix for one-dimension mass transport
- **WIER**
  - A specific project routine that computes the Rivière de Roches et Revillon Coupe weirs in the Peace-Athabasca Delta project
- **WSC**
  - A routine to read specific elevation or discharge data in Water Survey of Canada format
  -
- **WQIN**
  - A routine to read **TAPE5** water quality parameters
- **WQOPT**

        - A routine to output water quality data by checking the options chosen in **TAPE5** input
    - **WQSEQ**
        - A routine that tests for proper input of the water quality parameters

- **Ice Subroutines**:
    - **AXSECI**
        - Computes cross sectional area, hydraulic radius, friction slope at a defined water level
    - **BORDICE**
        - Computes border ice growth using method selected by the user
    - **BORDICEBREAK**
        - Computes breakage of border ice if triggered by rise in water level that exceeds a limit specified by the user
    - **CALCRIVERBANKS**
        - Calculates information of riverbank geometry for use in the simulation of border ice advancement
    - **DEFINEROUGH**
        - Estimates Manning's roughness of ice undersurface, based on ice thickness, and uses method selected by the user
    - **ELAREA**
        - Computes water level given cross sectional area
    - **FRICICE**
        - Manages the Manning's n-values and conveyance aspects of the hydraulic calculations
    - **ICECE**
        - Manages the calling of individual subroutines for ice cover evolution (ICECEA, ICECEB. ICECED
    - **ICECEA**
        - Computes leading edge stability for all ice segments, and determines if incoming ice will accumulate at the leading edge or be entrained into the flow that passes under the ice cover
    - **ICECEB**
        - Computes transport of ice under established ice cover
    - **ICECED**
        - Computes hydraulic forces on the ice cover and the thickening of the ice cover to resist these forces
    - **ICECI**
        - Ice cover initiation, based on user defined information on lodgement locations and timing
    - **ICEGENER**
        - Estimates the rate of frazil/slush ice generation at the open water surfaces
    - **ICEMOVEMENT**
        - Tracks and moves floating and suspended ice in the open water zone
    - **ICEMRG**
        - Checks for advancement of one ice segment up to the trailing edge of the next cross section; it merges such ice segments into a single segment

- **LEAD**
  - Function that computes leading edge stability
- **MELT**
  - Melts ice cover when incoming water is above zero degrees Celsius
- **NEWVEL**
  - Computes velocity at a cross section
- **OUTPUTICE**
  - Controls and provides output from ice calculations
- **RIVINI**
  - Ice related input
- **RIVINH**
  - Other input needed for ice subroutines
- **SLGTH**
  - Computes characteristic length of a given cross section

- **Post Processing**
  - **FILENAME**
    - Individual files for hydrographs and profiles are generated by this routine The routine provides an option for hydrographs and profiles to be produced as separate files for each location requested in **TAPE5** input. The format of the file name is as follows: "H_01_000600.txt" or "P_01_000050.txt"

      First character "H" or "P" are for hydrograph or profile; first two digit number is the reach number; second number is the location or "distance" along the reach for hydrographs or "time-step" for profile along the specific reach. These files can be further manipulated to produce individualized graphical output such as in EXCEL or other similar plot capable packages.
  - **TAPE6**
    - Text output of ice computations and ice forces plus volumetric computations
  - **OUTPUTICE**
    - Prepares intermediate and final output in a format requested by the user; can include general output only (water levels, ice thicknesses, etc.) or can also include detailed ice output, including force balances, Froude numbers, etc.

RIVICE has used the "parent" program, ONED, to maximum advantage. The driver for managing the use of the various ice subroutines has been incorporated in the original main program/driver of the ONED model. RIVICE uses the same conceptual setup of nodes and reaches as ONED, although it is not possible in RIVICE to address parallel arrangement of reaches, but rather only reaches in series.

Refresher on Program "ONED"

The ONE-D Program simulates transient flow conditions in rivers and tidal estuaries for divided flow or multiple channel situations where conventional steady-state routing models cannot provide reliable simulations. The program uses an implicit finite difference scheme to integrate the Saint Venant equations over a wide range of transient flows and conditions. As the name implies, ONE-D is designed to simulate dendretic or lopped networks where each channel segment is based on an assumption of one-dimensional flow; however, complex two-dimensional channel networks can be readily simulated under open water conditions. ONED has the ability to simulate the following features:

- irregular cross section geometry
- time-dependent lateral inflows or outflows
- multiple reaches connected to a node *
- two types of off-channel storage, non-conveying shoal areas and embayments *
- weirs, including road and dyke overflows *
- bridges * but may be characterized as a reach in RIVICE
- culverts * but may be characterized as a reach in RIVICE
- flood boxes (or aboiteaux) * but flood boxes may be characterized as a reach in RIVICE
- sea dams *
- pump stations with multiple pumps * but may be characterized as a lateral inflow/outflow in RIVICE
- dyke breaches *
- floodplain cells bounded by embankments *

Note that although the original ONED has the capabilities shown with asterisks, ***the program RIVICE does not.***

The theoretical basis for the finite difference solution procedure is described in Appendix 2 of the current manual.

EVOLUTION OF ONED MODEL DEVELOPMENT

The origins of ONE-D were in a research project at the Massachusetts Institute of Technology (MIT) by Gunaratnam and Perkins (1970), for which the solution scheme was developed. A multi-reach network model was later developed by Wood, Harley and Perkins (1972). MIT's Open Channel Network Model was modified and applied to several Canadian river systems by the Water Modelling Section of Environment Canada. These applications included the following locations with open water conditions:

- Ile de Montreal Dyking Project
- Confluence of Salmon and North Arm Rivers subjected to Bay of Fundy tides, Truro, N. S.
- Holland Marsh Model Studies, Lake Simcoe Region
- Red River Flood Forecasting, Emerson to Winnipeg Floodway
- Peace-Athabasca Delta Environmental Studies
- Lower Fraser River and Pitt River Hydraulic and Sediment Studies
- Serpentine-Nicomekl Floodplain Mapping Study
- Surrey Lowlands Flood Control Project

The program has been applied successfully in all cases. For the Lower Fraser River application, ONE-D simulated open-water hydrographs to within 0.2 ft. of the measured stages over a daily tidal cycle while reproducing instantaneous discharges within 4 to 7% of the measured flows. Verification was provided by discharge measurements by Water Survey of Canada using conventional and moving-boat methods in the Fraser and Pitt Rivers.

Special requirements for the simulation of the Serpentine and Nicomekl River system in open water conditions resulted in a number of significant program enhancements. Features such as sea dams and pump-stations were successfully simulated following the incorporation of software changes and new modelling techniques. This Manual reflects the stage in the development of the open water component of ONE-D following the Serpentine-Nicomekl Floodplain Mapping Project.

The original software was written in FORTRAN at a time when the only computers available were mainframe machines. This explains the origin of some of the terminology found in this Manual, such as "DATA" referring to a line of data in an ASCII file, or "TAPE" indicating a data file. Advancements in microcomputer technology and software created the opportunity for the programs to be efficiently executed on DOS-based microcomputers. This Manual targets applications on such machines.

BASIC ELEMENTS OF A ONE-D OR RIVICE MODEL

In a RIVICE or ONE-D model, each channel is divided into segments of assumed similar hydraulic characteristics called reaches. Each reach has a node at each end. Reaches can have any length, and are connected by nodes. For purposes of calculation by the finite difference method, each reach is further subdivided into segments of equal length called mesh spaces. The length of the mesh spaces can vary from one reach to another. The boundaries between adjacent mesh spaces are called mesh points, and it is for these point locations that the program computes water levels and discharges over a period of time. The physical properties of the channel at mesh points are interpolated from adjacent surveyed or derived cross sections.

The program achieves a dynamic solution by solving the equations governing fluid motion for the entire system at one instant in time, then solving them again for one increment of time, or time step, later. The process is repeated until the desired period of simulation is completed. Time steps are set by the user, and have a direct effect on the length of computation time required to complete a run. Time step length also has an effect on the stability and the accuracy of the numerical results generated by the program. Further comments on this aspect are located in Section _.

FAMILY OF PROGRAMS AND FILES

The RIVICE Program is the central piece of software in a large family of mostly batch-mode executable routines, some of which facilitate data preparation and input to RIVICE, and others that offer tabular or graphical post-processing of the output from a model run. An overview of the entire family of programs and how they are related to one another is illustrated in Figure 1-1, located at the back of this volume.

Two pre-processing programs, which facilitate the preparation of much of the data required for input to the main program, are included with the ONE-D package:

COORD1	prepares data for COORD2. One version allows the user to scan input cross section plots on the screen.
COORD2	prepares the tables of hydraulic parameters required by ONE-D from cross section data. This often forms the largest part of the ONE-D input data.

The computed results from a simulation with RIVICE can be presented in a variety of forms. For large models, and for runs with many time steps, the volume of data produced by a single run can be enormous, and the task of assessing all of the output can be greatly facilitated by using the graphical post-processors included in the RIVICE package. Traditional tabular summaries are provided directly in the TAPE6.TXT output file from the RIVICE run, and additional tables can be generated by the FILENAME subroutine generated from the **TAPE5** input process. These are useful when information at specific locations is required, or when a special summary is needed, such as a listing of the instantaneous peak water levels encountered at all locations during a flood simulation. However, the graphical presentation of model output as water level or discharge hydrographs, or water level profiles, provide the user with the ability to recognize trends and identify anomalies very quickly, even in very large output data files.

TYPICAL STRATEGY OF APPLICATION OF EITHER ONED OR RIVICE

A hydraulic modelling project typically involves three phases:

- Data provided through maps or previous HECII Datasets;
- Verification by plots generated by the **COORD1** program process;
- Data prepared for **RIVICE** in a finite-difference scheme of sections at regularly spaced intervals;
- Model development through a single channel configuration of single or multiple reaches;
- Calibration and verification through changing or modifying cross section spacing, time-step variability for more accurate ice evolution simulations and varying various roughness and ice parameters to better simulate calibration data;
- Final simulations of various scenarios or configurations.

During the model development phase, the input file to the RIVICE program, named TAPE5.TXT, is assembled. This is the stage for which the pre-processors were designed. TAPE5.TXT contains all the information necessary for a simulation to proceed, including cross section information, lateral inflow data, boundary conditions, and output parameters. Initial conditions can be specified in either TAPE5.TXT or, after at least one run has been completed, these can be read from output files of an earlier run for the same hydraulic network.

Once TAPE5.TXT is successfully assembled and debugged, the model can be calibrated and verified. This involves comparison of simulated results with measured data, and making repeated model runs with systematic adjustments in hydraulic parameters or other data that are not exactly known until the simulated results fit the measured data. This phase involves changing TAPE5.TXT, running the RIVICE program, viewing the results, making changes to TAPE5.TXT, then repeating the cycle.

When the model is satisfactorily calibrated, final simulations can be executed. The accuracy and validity of results from a final simulation are dependent upon the extent to which the model can be calibrated. There are two requirements for a good calibration; adequate coverage by observed data of good quality, and modellers with sufficient site knowledge and experience with open channel hydraulics that would allow them to exercise proper judgement throughout the modelling procedure.

2.2 RIVICE DESCRIPTION AND BASIC THEORY

The fundamental premise of the RIVICE software is that the calculations of ice generation and evolution can be separated from the hydraulic processes (water surface profiles, changes in flow and water level etc.) if they are done frequently. This is a so-called "loosely-coupled" relationship between the ice and the hydraulics. It does not require complex simultaneous solution of ice and hydraulic equations. However, the user must make a careful selection of the length of time step that suits the situation that he/she is trying to simulate.

The major processes of ice cover development that are represented by the software are described below, with the key algorithms that are available. Further details on programming strategy and structure are described in Appendix 3.

Two 'philosophies" that still persist in the program, and were based on original strategies that had been chosen in the 1990's are:

- Two sets of parameters were originally planned to be used, and identified by the variable "IEVOL". If IEVOL=1, the ice parameters were intended to represent ice cover evolution as a result of cold winter conditions and generation of frazil ice. If IEVOL=2, the ice parameters were intended to represent spring formation of ice jams resulting from accumulation of broken ice cover and floes. At this time, only one set of parameters is allowed (IEVOL=1). The user is able to adapt these parameters to whatever type of simulation (winter or spring) that he desires.
- The original strategy of RIVICE logic included the trial of "temporary" ice parameters such

as ice thickness ("THICKT" as the temporary trial version of "THICK", for example). The temporary parameters were intended to be modified in the logic if the first trial indicated that excessive changes (the phrase "big changes" was used by the original development team) were occurring. The fear was that "big changes" could trigger numerical instabilities from which recovery of the calculation process would not be possible. The notion was that "big changes" could be avoided temporarily if the time step and/or mesh spacing could be modified as necessary "on the fly". This strategy was not found to be necessary (so far) in the development of RIVICE. However, the concept of temporary variables (denoted by a letter "T" at the end of the variable) has been retained, in case new revelations as the program is tried under many heretofore untested conditions may warrant use of the original strategy.

A variety of options are available that could be deployed by users. Table 2.1 summarizes those options and lists the locations in the input data descriptions and the appropriate sections on relevant model theory.

Table 2.1 – Options Available to the User

Subroutine	Ice Phenomenon	Options Available	Input Data Identification / Description	Description of Theory
ICEGENER	Ice generation	Generation: 1 – Single heat transfer coefficient 2 – Detailed heat balance	Data I-y / I-z	Section 2.2.1
		Inflow of Ice Volumes as Boundary Condition (Supplements Generation)	Data I-ab	Section 2.2.1
ICECEA	Leading Edge Stability /Thickness	1 – Pariset / Hausser	Data I-h / I-i	Section 2.2.5
		2 - Ashton	Data I-h / I-i	Section 2.2.5
		3 – Fixed leading edge thickness (user)	Data I-h / I-i	Section 2.2.5
ICECEB	Deposition of Ice	1- Maximum velocity	Data I-b / I-c	Section 2.2.6
		2 – Meyer Peter Bed Load Analogy	Data I-b / I-d	Section 2.2.6
		3 – Densimetric Froude Number	Data I-b	Sectuion 2.2.6
ICECEB	Erosion of Ice	1 – Critical velocity	Data I-e / I-f	Section 2.2.6
		2 – Tractive force	Data I-e / I-g	Section 2.2.6
ICECED	Ice cover resistance to hydraulic forces	1 - Cohesionless	Data I-n	Section 2.2.7
		2 – With cohesion	Data I-n	Section 2.2.7
BORDICE	Development of border ice	1 - Modified Newbury	Data J-d, J-e, J-f, J-g	Section 2.2.2
		2 - Matousek	Data J-d, J-e, J-f, J-h	Section 2.2.2
		3 – User-defined	Data J-d, J-e, J-f, J-i	Section 2.2.2
DEFINEROUGH	Manning n-value of Ice Under Surface	1 – Beltaos Method	Data J-b / J-c	Section 2.2.8
		2 – KGS Method	Data J-b / J-c	Section 2.2.8
		3 – User-defined	Data J-b / J-c	Section 2.2.8
MELT	Melt of ice cover	1 – User defined heat transfer coefficient	Data J-j / J-k	Section 2.2.9

		2 – Approved RIVICE Method	Data J-j	Section 2.2.9
OUTPUTICE	Intermediate output generation	1 – General output	Data I-r to I-x	Section 2.2.10
		2 – Detailed ice force balances	Data I-r to I-x	Section 2.2.10
		3- Output at specific time steps	Data I-r to I-x	Section 2.2.10
		4 – Output at specific ice front advancement locations	Data I-r to I-x	Section 2.2.10

2.2.1 Ice Generation

Two subroutines manage the ice generation. "ICEGENER" computes the ice generated in each time step and at each cross section. "ICEMOVEMENT" computes the travel of the ice that has been generated in this and previous time steps at each cross section. There are two possible modes of calculating the ice generation. One is by using the water temperature algorithms that have been built into the Water Quality routines of ONED. This detailed method uses the detailed meteorological information that must be supplied by the user, including:

- Air temperature,
- Wind speed,
- Relative humidity
- The meteorological heat-loss methodology has not been extensively tested and should be considered as being in the developmental phase. Further work is needed to verify and confirm the results that are being produced before accepting to use the method that has been adopted for existing ONE-D open water conditions.

The program estimates the heat loss from the water surface. If the water temperature is driven below zero degrees Celsius, the amount of ice that can be potentially generated is directly proportional to the difference between the water temperature and zero degrees C.

Details on this rigorous method of computing heat loss from the open water surface is described more thoroughly in Appendix 6.

The second alternative method of estimating heat loss and ice generation, provided in **RIVICE**, is with a simplified algorithm as follows:

Heat loss = C * $(T_w – T_A)$ where:

Heat loss - heat transferred from water surface to the atmosphere, in Watts per square metre

T_w - Water temperature (degrees Celsius)

T_A - Air temperature (degrees Celsius)

C - Empirical coefficient (Watts per square metre per degree Celsius). A value of between 15 and 25 has been found to a good estimate of the appropriate coefficient for rivers in Canada.

The computation of the heat loss, for either method, compensates for the fact that ice pans on the surface of the open water will provide insulation and will significantly reduce the rate of heat loss that would normally occur from open water. Similarly, the open water can also be insulated by border ice coverage. RIVICE neglects the surface area covered by the floating ice pans and border ice, as the heat loss through that ice is at least one order of magnitude less than from the open water. Detailed calculation of the loss through the ice pans/border ice is considered unjustified, given the crude overall accuracy of the computation of ice generation.

Once the heat loss from the water has been computed has been computed, it is converted into a potential generation of ice using the heat of fusion of water to ice (69 000 Joules per kg, or 144 BTU per pound).

When each increment in ice generation has been computed as described above, ICEMOVEMENT is then deployed to compute how far each parcel of ice will travel during the time step. This is based on the computed flow velocity from the hydraulic subroutines. It is assumed that the slush ice travels at the same velocity of the water. If the ice parcel travels only part way to the next cross section or partially beyond it, a system of allocating and distributing an appropriate portion of the ice in a downstream direction is deployed. For example if a parcel of ice travels beyond a given cross section by, say 60% of the way to the next cross section, then 60% of the ice is allocated to the upstream cross section and 40% to the downstream cross section. This is demonstrated schematically in Appendix 3.2. While this is not a fully rigorous way of representing the movement of the ice, it is well within the needed accuracy for the purposes of this software.

2.2.2 Border Ice Formation

Border ice advances from the sides of the channel in most northern rivers, unless the flow is exceptionally swift. The means to predict the extent of border ice formation are not precise, and can be considered as approximate at best. Three possible means to define the border ice advancement are provided.

1. ***User Defined*** - The user defines the maximum border ice width at each cross section and the time when the border ice growth reaches the maximum width. The border ice width is calculated using the following equations,

$$W_b = mtW$$

$$m = \frac{a_1}{b_1}$$

Where:

W_b = Total border ice width at a cross section (m);

m = Ratio between the simulation time ratio and the maximum border ice width ratio;

t = Simulation time ratio between the current Simulation time and the total simulation period;

a_1 = Simulation time ratio defined by the user when the border ice width is equal to the maximum border ice width. Also, if $t > a_1$, $W_b = b_1$;

b_1 = Border ice Width ratio defined by the user at a cross-section (the maximum border ice width/top channel width).

W = Top width for flows at a cross section (m).

2. ***Newbury Empirical Method*** – This method has been used with moderate success in Northern Manitoba:

$$W_b = mD_d$$

$$m = \frac{a_1}{v^{b_1}}$$

Where :

W_b = Total border ice width at a cross section (m);

m = Coefficient to relate degree-days of freezing to border ice growth (m/^{o}C-Day);

D_d = Degree-days of freezing (^{o}C-days);

a_1, b_1 = Coefficients supplied by the user and based on observations of the river under study. Representative values for northern Manitoba conditions are a_1=0.054 and b_1=1.5;

v = Average velocity of flow at the cross section (m/s).

3. ***Matousek Method*** – According to Matousek, a static ice cover will form if the following condition is satisfied.

$T_A < 0$ and $T_w = 0$ and $V < V_{sb}$

Where :

T_w = Water temperature (^{o}C);

T_a = Air temperature (^{o}C)

V = Local depth-averaged flow velocity in the open water adjacent to edge of the existing border ice (m/s)

V_{sb} = Maximum vertically averaged velocity of flow into which border ice advances laterally (m/s);

$$v_{sb} = \frac{\phi_e}{1130(-1.1 - T_w)} - \frac{b_2 U_2}{1130}$$

b_2 = Coefficient =-0.9 + 5.8 log (open water width in metres)

U_2 = Wind velocity at an elevation of 2 m above the water surface (m/s)

ϕ_e = Net heat flux per unit area due to heat exchange between water surface and the atmosphere (watts/m^2).

If $0 > T_a > -12^oC$,

$$\phi_e = -81 + 12 T_a + 3.2(0.8 T_a + 0.1) U_2 + 0.1(318 + 4.6 T_a)$$

If $T_a < -12^oC$,

$$\phi_e = -96 + 11 T_a + 3.2(0.7 T_a - 0.9) U_2 + 0.1(326 + 4.6 T_a)$$

2.2.3 Border Ice Breakup

Changes in water level typically cause severance of the border ice from the riverbanks. This is simulated by the program and is triggered by changes in water level (up or down) from the water level that exists when the border ice first starts to form. If the water level rises or falls by an amount that exceeds the user-specified limits, then full severance of the border ice is represented, from both sides of the river. The severed border ice volume joins with whatever slush ice is in transit at that point in time at that location, to form the ice load that continues downriver in subsequent time steps. Eventually that ice load may reach and accumulate against an existing ice front.

2.2.4 Ice Cover initiation

There are no known reliable methods to predict where slush ice or ice floes will arrest/lodge and form a stationary ice cover. RIVICE has been developed with only two possible means of initiating an ice cover:

- Advancement of border ice that covers the full width of the river and has not broken up due to rising (or falling) water levels
- Formation of a bridge or lodgement that has been pre-selected in time and location by the user. The user is allowed to select up to 5 ice lodgement locations, and each one can be at separate time steps or on the same time step.

The simulation of initiation of the ice cover is managed in Subroutine "ICECI".

Initiation of an ice cover is assumed to occur with a thickness that is selected by the user (default value of 0.2 m), and covers the full length of the river reach at the identified cross section. That is, it covers the river from the midpoint between the identified cross section and the next downstream section (or mesh point in the context of the 1-D system), to the midpoint between the identified cross section and the next upstream cross section (or mesh point).

Once the bridge forms through either one of the processes described above (or both), the program begins to represent the accumulation of ice at the leading edge of the stationary ice segments. This is done by the Subroutine "ICECE", which in turn calls Subroutine "ICECEA", which manages the representation of the juxtaposition phenomenon (see Section 2.2.5).

In each time step, the program checks whether any ice segments that have been formed in previous time steps have grown to the point that they are starting to merge with another ice segment. If that occurs, Subroutine "ICEMRG" manages the process of merging two ice segments into one large one.

2.2.5 Ice Cover Evolution – Leading Edge Stability

Subroutine "ICECEA" manages the simulation of ice cover advancement and submergence of ice at the leading edge if incoming velocities exceed an allowable threshold. The phenomenon that is represented is the accumulation of the incoming ice at the leading edge of the ice cover, or submergence of the incoming ice if the flow is swift and the ice can be swept under the leading edge.

Three possible means of estimating the "stability" of the ice approaching the leading edge of the ice cover are included in the program:

1. ***<u>Juxtaposition</u>*** of the ice according to the following relationship (Pariset, 1966; Michel, 1971)

$$\frac{V}{\sqrt{gH}} = \sqrt{2\frac{(\rho - \rho_i)}{\rho}(1-e)\frac{t}{H}}\left(1-\frac{t}{H}\right)$$

Where :

V = Mean velocity of flow in open water upstream of leading edge (m/s)

H = Mean hydraulic depth in open water upstream of leading edge (m)

ρ = Density of water (kg/m^3)

ρ_i = Density of ice (kg/m^3)

e = Porosity of ice pans/floes at the leading edge (i.e. ratio of volume of voids filled with water to the total volume of the ice pan and is user specified)

t = Thickness of ice accumulation (m) which will form with the combination of V and H; this relationship only holds for **t/H** ratios less than 0.33, which is the limiting condition for juxtaposition.

2. **Stability of ice blocks** can be represented by the algorithm developed by G. Ashton (1986):
3. It has been observed that in conditions where ice approaches the leading edge in a **steady stream of pans** or as a continuous blanket of slush ice, that the classic leading edge stability equations as defined by (1) or (2) does not apply. The tendency of drawdown of individual ice fragments is significantly reduced by the protection provided by the continuous, congested stream of surface ice. The user can represent this case, which results in a leading edge thickness that is selected and quantified by the user, with a default of 0.15 m. Further thickening downstream of this edge would occur if shoving were shown to be necessary, as described in Section 2.2.7. No submergence of the approaching ice under the leading edge is permitted under this option, regardless of the velocity at the leading edge. All ice accumulates as an advancing thickness as defined by the user.

Advancement of the ice cover by juxtaposition occurs in finite steps depending on the rate of incoming ice. The partial advancement from one cross section to the next can occur and the length of the leading edge depends on the incoming ice volume, in addition to what might have existed in the previous time step. Key variable names that represent the ice in this leading position are:

- NFRT1T – upstream-most cross section number that is fully covered by ice over its entire length
- XFRZ1T – length of ice that extends towards the next cross section upstream of NFRT1T
- TLE1T - thickness of the ice cover over length XFRZ1T

These variables are shown schematically in Appendix 3.1.

2.2.6 Deposition / Erosion of Ice Cover

Subroutine "ICECEB" manages the representation of these processes. The basic phenomena that are represented are:

- Depositions of ice on the under-side of the stationary ice cover, if the velocity is below a computed, or specified, threshold, and there is ice-in-transit under the ice cover at that location.
- Erosion of the ice cover if the velocity exceeds a computed or specified limit. The ice will be thinned until the velocity reaches the specified limit. If an ice cover reaches a thickness that is less than 0.15 m, the erosion process is truncated at that point, and the user is warned of the erosion limit being unachievable.

DEPOSITION UNDER THE ICE COVER

The options to represent the deposition process are:

1. A simple **user-defined velocity**, when exceeded, will not allow further ice deposition. Ice that is in transit will continue to travel downstream if this threshold is exceeded.
2. A more complex limit based on an analogy with the **Meyer-Peter equation** that is used to estimate the transport of bed-load sediment in a river. The basic premise is that ice deposition on the ice under surface is analogous to deposition of bed-load sediment on the channel bed. The algorithm that is embedded in the program is:

$$3281\frac{V^2}{c^2} = 12.3\,d_i + 0.84\,q_u^{\frac{2}{3}}$$

Where:

V = mean velocity under ice cover (m/s)

C = Chezy roughness coefficient for water passage ($m^{½}/s$)

d_i = characteristic dimension of ice fragments (m)

q_u = ice discharge per unit width under the cover weighed under water with apparent density 0.08

The main difficulty of this method is the determination of the appropriate dimension "di" for the problem being analyzed. The transport rate computed by the Meyer-Peter method will be no more accurate for ice transport than it is for sediment transport. However, it does acknowledge the concept that ice will have more tendencies to deposit at higher velocities if the rate of the incoming ice volumes is high.

3. Deposition controlled by the magnitude of the densimetric Froude number under the ice cover. This is defined by the equation:

$$F_r = \frac{V}{\sqrt{gH\frac{(\rho - \rho_i)}{\rho}}}$$

Where:

F_r = maximum Froude number at which ice can deposit (user-selected)

V = mean velocity (m/s)

g = acceleration due to gravity (m/s/s)

H = hydraulic mean depth under the ice (m)

$\rho - \rho_i$ = density of water and ice respectively (kg/m^3)

EROSION OF THE UNDER SURFACE OF THE ICE COVER

If the velocity and shear stresses are large enough, erosion of the established ice cover can occur. The precise mechanism and means to predict this process are not well known, and two options are available in RIVICE to represent this phenomenon numerically:

1. **Maximum velocity**, above which erosion will commence. This is specified by the user in the input data and becomes a maximum velocity that can be maintained. Erosion of the ice cover is simulated so that this velocity can be maintained. If a minimum threshold (ice thickness of 15 cm) is threatened by the erosion process, the user will be warned, but no ice cover breakup is attempted.

2. **Tractive force,** above which erosion of the ice cover occurs. Tractive force under the ice cover is estimated with the following algorithm:

$$F_d = \alpha RS$$

Where:

F_d = tractive force (Pa)

α = specific weight of water (N/m^3)

R = hydraulic radius under ice cover (m)

S = friction slope (m/m)

In both options, any erosion that occurs releases the eroded ice volume into the volume in transit that is being moved downstream. It can as a result deposit eventually at a downstream location where the criteria for deposition is satisfied.

2.2.7 Ice Cover "Shoving" Due to Hydraulic Loading

An ice cover on flowing water is subjected to hydraulic forces which can cause deformation and thickening. The classic means of analyzing this has been with the "bell-curve" developed by Pariset, Hausser and Gagnon (1966), However, two disadvantages arise from direct use of the bell-curve :

- It can only represent the ice cover thickness and stability at a distance of several river widths from the leading edge.
- It represents the stability of a constant width channel, with constant velocity, etc., which rarely occurs.

A refinement to this concept which avoids the difficulties cited above has been used in RIVICE and is suited to computation by a computer program. It involves the incremental summation of computed forces on the ice cover in a step-mode beginning from the leading edge and advancing from cross-section to cross-section in the downstream direction. It computes :

(i) Forces exerted by the flowing water on the ice cover:

- Hydrodynamic thrust on the leading edge (Michel, 1971)

$$F_t = \left(1 - \frac{d}{H}\right)^2 V_u^2 BH \frac{\gamma}{2g}$$

Where :

F_t= Hydrodynamic thrust of the flow (N)

H = Depth of water upstream of leading edge (m)

d = Depth of flow under the leading edge (m)

V_u = Velocity under the leading edge (this is a mean value across the width of the channel, in m/s)

B = Width of ice cover (m)

γ = Specific weight of water (9800 N/m^3)

g = Acceleration due to gravity (m/s^2)

- Hydraulic drag of the flow on the ice under-surface (Michel, 1971) :

$$F_d = \left(\frac{\gamma\, d_t\, s\, n_i^{1.5}}{2\, n_c^{1.5}}\right) A_{iw}$$

Where:

F_d = Hydraulic frictional drag force (N)

γ = Specific weight of water (9800 N/m^3)

S = Slope of hydraulic grade line

n_i = Manning's roughness coefficient of the ice under-surface

n_c = Manning's roughness coefficient of the composite cross-section

d_t = Depth of flow under the ice cover (m)

A_{iw}= Under-surface area of ice exposed to flow (m^2)

- The component of weight of the ice cover and the water contained in its voids, acting along the hydraulic gradient:

$$F_w = \gamma_i V_o S$$

Where:

F_w = Gravitational force acting along the channel (N)

γ_i = Specific weight of ice (9020 N/m^3)

V_o = Volume of ice cover between the banks, in m^3, including voids infilled with water and voids above the phreatic line.

S = Slope of hydraulic grade line

(ii) Force shed to the river banks includes cohesion of the ice cover to the banks and friction of the ice cover against the river banks. The cohesion expression (Pariset, 1966) is given as:

$$F_c = 2c\,t\,L$$

Where :

F_c = Force of cohesion of ice to two river banks (N)

c = Cohesion per unit area of ice/bank interface (Pa)

t = Average thickness of ice cover between cross-sections (m)

L = Distance between cross sections (m)

The hydraulic forces exerted on the ice cover in the stream-wise direction create stresses in the ice, which are spread laterally towards the riverbanks. The lateral stress results in a reaction of static friction at the bank, which acts as a stabilizing influence on the cover.

From Pariset (1966) :

$$F_f = 2f\ t\ L\ K_1 \tan\phi$$

Where :

F_f = Friction force on the ice along the river bank (N)

f = Compressive stress in the ice cover along the channel (Pa)

K_1 = A coefficient equal to the ratio of lateral stress to longitudinal stress in the ice cover (a ratio less than or equal to 1.0)

tan φ = Tangent of angle of friction of ice/bank interface

L = Distance between cross-sections (m)

t = Average ice thickness between sections (m)

As the calculation proceeds downstream, the stress in the ice cover is determined from:

$$f_i = \frac{(F_t + F_d + F_w - F_c - F_f)}{t\,W}$$

Where :

f_i= Stress in the ice cover (Pa)

F_t, etc.= As defined above

t= Ice thickness (m)

W= Ice width (m)

The accumulation of stress in the ice cover is computed in increments of 10% of length between the mesh points. Load is shed to the banks in each increment according to the average stream-wise stress in the increment. The tenth increment (i.e. the downstream mesh point) thereby properly reflects the accumulation of hydraulic loading, which is reduced by the bank resistance.

If the stress exceeds the maximum resistance of the ice cover, shoving or telescoping of the ice must occur to attain the minimum required thickness. The resistance is determined from Pariset (1966):

$$F_{ir} = \gamma_i \left(1 - \frac{\gamma_i}{\gamma}\right) \frac{t^2 W}{2} K_2 \bullet e$$

Where :

F_{ir} = Internal resistance of ice cover (N)

K_2 = A coefficient greater than or equal to 1.0, a Rankine passive coefficient in soil mechanics

t = Ice thickness (m)

W= Ice width (m)

γ_i= Specific weight of ice (9020 N/m^3)

γ= Specific weight of water (9800 N/m^3)

e = Porosity of the surface ice cover (it is a function of the volume of voids to the total volume of the surface ice cover and is user specified)

The values of K_1, tan φ, and K_2 are key components of this procedure. The value of each is not known precisely, but it has been shown that the combination :

$$\mu = K_1 K_2 \tan\theta$$

The value of μ is normally between 1.0 and 1.60 (Acres, 1986; Pariset, 1966; Beltaos, 1983; Beltaos, 1988). The value of the individual factors K_1, K_2, tan φ is left to the discretion of the user, with default values of .18, 8.7 and .9, respectively.

It should be noted that some investigations show "μ" as including a term (1-e) where "e" is the ice cover porosity. However, the original derivation (Pariset, 1961 and 1966) did not include this term, and it is not included in RIVICE.

The simulation of a shove is done by :

- Thickening of the ice cover at an unstable location (i.e. stress in ice cover exceeds its internal resistance) to achieve a stable thickness; this may be restricted in any given time step by the maximum rate of movement of the ice as described below.
- Reduction in ice volume at the leading edge to be equivalent to the volume required to thicken at the unstable location (a downstream "recession" of the leading edge results).

The volume of ice which is supplied to thicken the cover at an unstable location is limited by the maximum rate of movement of the ice cover, estimated to have a maximum speed equal to the average flow velocity :

$$V_M = V_s t_s W_s \Delta t$$

Where : V_M= maximum volume which can shove to an unstable location during a given time step, (m^3)

V_s= mean flow velocity at the unstable cross section (m/s)

t_s= ice thickness at unstable cross section before shoving occurs (m)

W_s= width of ice cover at unstable location (m)

_t= time step (seconds)

This has been introduced because there must be an upper limit to the volume of ice that can move in a shove during a time step. It is believed that the local velocity of water flow is an indicator of this. The sensitivity of the simulation of shoves should be evaluated in future testing of RIVICE.

In addition, there are practical restraints that must be observed to avoid instabilities in the hydraulic solution. Abrupt changes in ice thickness that would exceed a rate of change of 0.5 m over a length of 20 m are avoided.

2.2.8 Hydraulic Roughness of Ice Cover and Riverbed

An ice cover over a river channel creates an additional fixed boundary and subjects the channel flow to additional frictional resistance. Where an ice cover must be simulated in a numerical model, the channel characteristics must be modified to properly reflect the effects of the ice cover. As in the case of open water flow, frictional head losses are normally computed using the Manning equation. However, the conveyance coefficient of an ice-covered channel is calculated using an adjusted form, as follows:

$$k_i = \frac{1}{n_c} A_i R_i^{2/3}$$

Where,

n_c= the composite Manning's roughness (this replaces the roughness coefficient for the channel bed that is used for open water calculations) ($m^{-1/3}s$);

k_i= the conveyance coefficient of an ice-covered channel (m^3/s);

A_i= the flow area beneath the ice cover (m^2);

R_i= the hydraulic radius under the ice cover

$$\left(= \frac{A^i}{(B_i + P_b)} \right) \text{(m)};$$

P_b= the wetted perimeter of the channel bottom and side slopes that lie beneath the ice cover (m);

B_i= the width of the underside of the ice cover from river bank to river bank (m).

The composite roughness n_c can be estimated using the Belokon-Saboneev formula:

$$n_c = \left(\frac{n_b^{3/2} + n_i^{3/2}}{2} \right)^{2/3}$$

Where,

n_b = the channel bed Manning's roughness;

n_i = the ice cover Manning's roughness.

Two optional means of defining the relationship between n-value and ice thickness are provided in RIVICE, with a third being a user-defined value:

1 KGS METHOD

Nezhikhovskiy (1964) provided the estimated coefficients of Manning's roughness of the under surface of ice for different types of ice cover that were measured in Russia. Figure 1 shows the curve that estimates the Manning's roughness of the under surface of the ice cover. The basic equation is shown in Figure 1, based on a Manning n-value of 0.105 for an ice thickness t = 8m. However, Nezhikovskiy indicated that this relationship is a function of the type of ice cover. For example, the Manning n-value could be substantially more for an ice cover formed by solid ice floes that form an ice jam, and could be less for frazil ice deposits.

The method that is adopted for RIVICE, based on the methodology that has been used by KGS Group for many years, is to allow the user to define the magnitude of the n-value at an ice thickness t = 8m, and allow the program to prorate that value for other thicknesses, in accordance with the shape of the curve shown in Figure 1. Figure 2 shows a family of curves that demonstrate the principle.

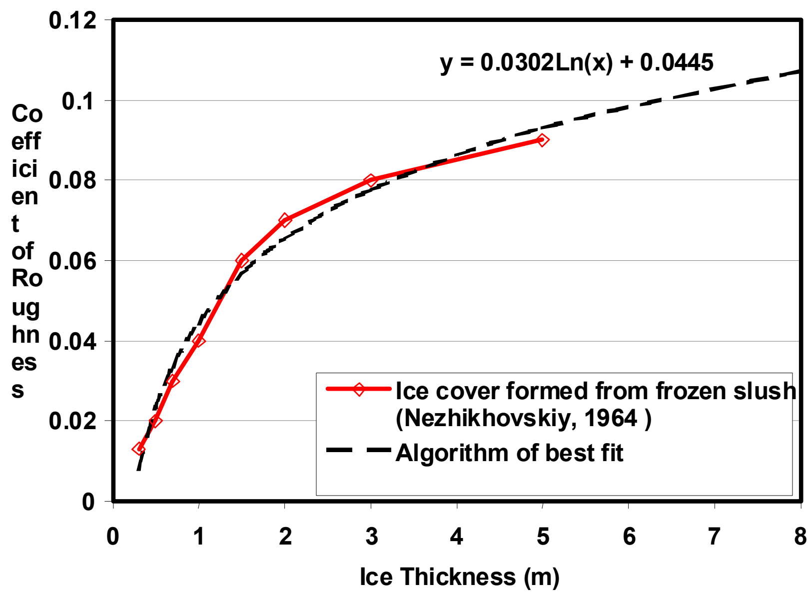


Figure 1: Manning Coefficient of Roughness vs. Ice Thickness

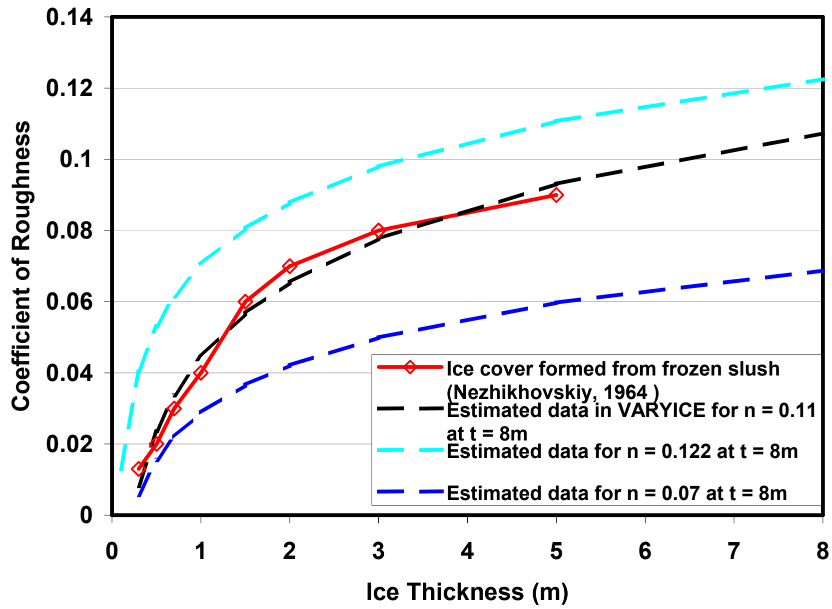


Figure 2: Manning Coefficient of Roughness vs. Ice Thickness for Various User-defined Conditions

This methodology allows the user to define the general shape of the relationship of n-value vs. ice thickness, depending on field observations of water levels and flows. It allows an effective means of model calibration that is based on real field data.

2 METHOD DEVELOPED BY DR. S. BELTAOS

The basic relationship developed by Dr. Beltaos is:

$$n_o = 0.10\sqrt{c}\, t_s^{\ 1/2} h^{-1/3}$$

Where:

n_o = Composite Manning n-value for ice and riverbed

c = coefficient, with default range of 0.4 to 0.6

t = ice thickness (m)

h = depth (m)

with the constraints: $0.03 < n_o < 0.10$ (or other, user-specified, limits).

In RIVICE, it is required to also calculate the values of the ice- and bed- coefficients n_i and n_b. This can be accomplished by noting that

$$n_i / n_o = (R_i / R_o)^{2/3} = (f_i / f_o)^{2/3} \qquad n_b / n_o = (R_b / R_o)^{2/3} = (f_b / f_o)^{2/3}$$

Details of this method are provided in Appendix 3.

3 USER-DEFINED MANNING N-VALUE

This is a straightforward user-supplied value of the ice cover n-value for each cross section.

2.2.9 Melt of Ice Cover

If the user has chosen the detailed method of temperature calculation, the program can allow melting of the ice cover if the incoming water temperature at the leading edge of an ice cover exceeds zero degrees Celsius. The ice cover is allowed to decrease in thickness to a minimum value of 0.15 m, at which point a warning is issued to the user. Full elimination of the ice cover due to melting is not attempted, as this is a process that is beyond the numerical capability of this software.

The rate of melting of the ice cover can be estimated with one of two possible methods:

1. With a **<u>user-defined heat transfer coefficient</u>**. The heat transferred to the ice, and thereby focussed on melting the ice is computed as follows by the program:

   $$H_{wi} = C$$

   Where: C is a user supplied value in BTU/m^2/day
2. The algorithm that was originally proposed by the TALAS engineers that devised the original methodology for the heat balance analysis for RIVICE. It is:

   $$H_{wi} = 44757 * VELSECTION^{0.9} * TLXWT / (AEX/TW1)^{0.1}$$

   Where:

   44757 is a computational conversion factor

   VELSECTION is the section velocity

   TLXWT is water temperature at this section for this time step in °C

   AEX is the actual pre-erosion flow area

   TW1 is the section top width

3. SOFTWARE STRENGTHS AND WEAKNESSES

3.1 STRENGTHS RE: ICE SIMULATION

1. The RIVICE model has been developed mainly to assist the analysis of thick ice covers that are characteristic of relatively steep rivers where formation of ice dams and ice jams are dominant.
2. The program includes analyses for estimating the rate of generation of frazil/slush ice in the open water, and to represent the advancement of border ice across the channel.
3. The program analyses the stability of the incoming ice pans at the leading edge of the established ice cover to determine whether they will be swept under the ice cover or accumulate at the leading edge by the process of "juxtaposition". The program also analyses the hydraulic forces that are generated by the flow on the evolving ice cover and estimates the amount of ice cover thickening that must occur for the ice cover to remain in place.
4. The program analyses the movement of submersed ice and estimates the locations where the velocity reduces sufficiently to allow the ice to deposit on the underside of the stationary ice cover. Representation of "hanging ice dams" with this numerical technique is feasible.
5. The program is structured in a manner that allocates the simulation of each basic ice phenomenon to separate subroutines. The information at large is available to each subroutine through common blocks of variables. This facilitates debugging, and allows for insertion of user-developed subroutines, where desired.
6. Spring break-up of ice covers is not directly represented. However, the program can be used to simulate a spring ice jam lodgement and accumulation at a pre-selected location.

3.2 LIMITATIONS RE: ICE SIMULATION

1. The model is not well suited to simulating the formation of ice at velocities less than 0.3 m/s where the ice formation processes are more thermally driven than mechanical, and ice thickening is typically due to frost penetration as the winter proceeds.
2. The program is oriented mostly towards the formation stage of ice covers and ice jams. The technology in predicting the break-up and movement of previously established ice covers is poor. The only capability that has been included in RIVICE for breakup is a user defined rate of ice inflow that can accumulate a pre-selected lodgement or bridge location, thereby enabling the simulation of an ice jam.
3. The software as was originally conceived and developed does not readily permit the user to start with a pre-established ice cover (which was to become part of the TAPE17.TXT file already available in the ONE-D hydraulics), from which additional evolution could extend. This may be a desirable future capability to consider (see Section 13).
4. In the simulation of the ice cover evolution; there is a capability to represent a cohesive component of the ice cover resistance. However, the cohesion does not vary in time as frost penetration develops. Similarly, there are no means to allow the Manning n-value of

the ice to change over time due to the smoothing effects that are known to occur under river ice covers.

5. There is no representation of suspended ice in the water column, and the slow rise of ice under flotation forces. All ice generated is assumed to be carried at the surface, or adjacent to the ice under surface when transport under the ice cover occurs.
6. There is a subroutine to estimate the lateral advance of border ice, and breakup if the water level rises sufficiently to cause it to sever from the banks. However, the accuracy of this representation is not good. The technology of prediction of border ice advancement is the limiting factor. Furthermore, the optional method of Matousek for estimating the lateral advancement of border ice ideally requires a two-dimensional calculation of velocity, and the pseudo-2 D approach used by RIVICE is only a rough approximation at best.
7. Thinning of an established ice cover is possible if the incoming water temperature exceeds zero degrees. However, thinning below 0.15 m is not permitted and breakup of the ice cover is not represented.

4. HARDWARE SOFTWARE INSTALLATION

4.1 HARDWARE REQUIREMENTS

The minimum recommended hardware and software requirements are largely dictated by the size of the model to be created. Currently available laptop or desktop have installed memory that far exceeds the requirements for the RIVICE model. The current challenge is that FORTRAN processors must be used that are compatible with the operating systems installed on the computer. In some cases a version compiled in XP does not perform on a WINDOWS 7 operating system. This is primarily due to the computer architecture and associated removal of ealier operating system routines.

If custom hard copy graphics of the ONE-D model output through a spreadsheet are desired, then a spreadsheet program, such as Quattro or Excel, would be required. If the CAD routine for creating arrays of plotted hydrographs is intended to be used, then an installation of AutoCAD Release 12, or later, must be available.

4.2 INSTALLATION REQUIREMENTS

This section applies to installation of the RIVICE software that will be compiled using the LAHEY FORTRAN Compiler, which is recommended for this purpose. For installations of the RIVICE software that will be compiled using other compilers (such as Microsoft and Compaq), the user is referred to the manuals provided with those compilers.

For installation of other compilers themselves, the user is also referred to the manuals supplied with these programs. To install the RIVICE program and support software, proceed as follows:

1. Copy all the contents of the CD marked "PROGRAM SOURCE CODE" to a directory on the hard disk. It is assumed that this directory will be named C:\1D.
2. The data on the CD marked "TEST ROUTINES" should be copied to a separate directory, such as C:\1D\LIBTEST.
3. Assuming the executable programs created by the compiler will be stored in the C:\1D directory, make a PATH to this directory in the AUTOEXEC.BAT file located in the root directory.

An alternative arrangement which would separate the executable and source files would be to keep all the executable files, and the two LAHEY files RUN386.EXE and F77L.EER, in the C:\1D directory after compilation, and to move the source code files to a directory such as C:\1D\SOURCE\.

1. For the certain routine, a large number of files must be open simultaneously. Therefore, change the line:

```
FILES = xx
```

in the CONFIG.SYS file such that the number of files, xx, is equal to greater than 12 plus the total number of reaches expected in the model. If no such line exists in the CONFIG.SYS file, then it should be added.

Very early DOS operating systems only allowed a maximum of 256 files to be open at one time; therefore for models with more than 244 reaches will not run. This limit was being addressed by the developers of ONE-D at the time of writing.

4.3 COMPILATION PROCEDURES

The following instructions pertain to compiling RIVICE and associated software using the LAHEY F77L EM/32 FORTRAN Compiler. For compilation using other compilers, please refer to the manuals provided with these compilers. For older versions of these compilers, it may be useful to also refer to the February 1988 document entitled, "One-Dimensional Hydrodynamic Model Computer Manual," by Environment Canada. The LAHEY compiler was one of the first to create executable programs that access extended memory, allowing them to exceed 640 Kb in size. This is a requirement for large models.

Some of the programs in the RIVICE family, including BUILD, BROWSE, EXPORT and the most recent version of CD1PLOT will only work if compiled using the LAHEY FORTRAN compiler, as the LAHEY software has screen graphics capability that the others do not. Users are also warned that in the instances where one program creates a binary file that will be used as input into another program, both programs must be compiled with the same compiler, since the format for reading and writing binary files can vary from one compiler to another, and use of different compilers in such cases will likely lead to program execution failures or errors. For simplicity it is recommended that the LAHEY compiler be used throughout.

Users are cautioned that the current version of the RIVICE program has been tested on a LAHEY Fortran compiler and on a Compaq Fortran compiler only. To verify program performance on other compilers, the standard test data sets (available from Environment Canada) should be executed and the output checked against the standard test results.

Prior to compiling the main RIVICE program, the source code file requires modification to include the values of 23 variables used to define array sizes in the executable program. The procedure to define these variables is described in the subsection below entitled "Dimensioning Arrays in the RIVICE Source Code."

To compile any one of the FORTRAN source code modules, identified by the extension .FOR in the filename, proceed as follows using the LAHEY Compiler (MS will update as appropriate):

1. Working from the C:\1D directory, with the source code file, <file>.FOR, present in the same directory, and with a path to the LAHEY compiler directory (set by the LAHEY install program as C:\F77L3\BIN), type the following at the DOS prompt:

```
F77L3 <file>.FOR↲
```

where <file> is the name of one of the following source code files to be compiled:

CD1PGM	ONEDPGM
CD1PLOT	CD2PGM

After the compiler responds with a listing of all subroutines compiled and any warnings, the system reverts to the DOS prompt. This will create several files with the extensions .OBJ, .MAP, .LST and .SLD in the working directory.

When compiling any of the source code modules, make sure that the Bounds checking compiler configuration option is set (i.e. /B is specified on the command line or in the "F77L3.FIG" file).

2. Upon completion of Step 1 for the programs CD1PGM, CD2PGM, and ONEDPGM, type the following:

```
386LINK <file>↵
```

For the program CD1PLOT ensure the LAHEY graphics file GRAPH3.LIB resides in the directory named C:\F77L3\LIB (created by the LAHEY installation program) , and type the following:

```
386LINK <file> -LIB \F77L3\LIB\GRAPH3↵
```

Upon completion of this step the executable file named <file>.EXE will be created.

The following instructions pertain to compiling RIVICE and associated software using the LAHEY F77L EM/32 FORTRAN Compiler. For compilation using other compilers, please refer to the manuals provided with these compilers. For older versions of these compilers, it may be useful to also refer to the February 1988 document entitled, "One-Dimensional Hydrodynamic Model Computer Manual," by Environment Canada. The LAHEY compiler was one of the first to create executable programs that access extended memory, allowing them to exceed 640 Kb in size. This is a requirement for large models.

Some of the programs in the RIVICE family, including the most recent version of CD1PLOT will only work if compiled using the LAHEY FORTRAN compiler, as the LAHEY software has screen graphics capability that the others do not. Users are also warned that in the instances where one program creates a binary file that will be used as input into another program, both programs must be compiled with the same compiler, since the format for reading and writing binary files can vary from one compiler to another, and use of different compilers in such cases will likely lead to program execution failures or errors. For simplicity it is recommended that the LAHEY compiler be used throughout.

Users are cautioned that the current version of the RIVICE program has been tested on a LAHEY Fortran compilers and a Compaq Fortran compiler only. To verify program performance on other compilers, the standard test data sets (available from Environment Canada) should be executed and the output checked against the standard test results.

Prior to compiling the main RIVICE program, the source code file requires modification to include the values of 23 variables used to define array sizes in the executable program. The procedure to define these variables is described in the subsection below entitled "Dimensioning Arrays in the RIVICE Source Code."

To compile any one of the FORTRAN source code modules, identified by the extension .FOR in the filename, proceed as follows using the LAHEY Compiler.

3. Working from the C:\1D directory, with the source code file, <file>.FOR, present in the same directory, and with a path to the LAHEY compiler directory (set by the LAHEY install program as C:\F77L3\BIN), type the following at the DOS prompt:

```
F77L3 <file>.FOR↵
```

where <file> is the name of one of the following source code files to be compiled:

CD1PGM	ONEDPGM
CD1PLOT	CD2PGM

After the compiler responds with a listing of all subroutines compiled and any warnings, the system reverts to the DOS prompt. This will create several files with the extensions .OBJ, .MAP, .LST and .SLD in the working directory.

When compiling any of the source code modules, make sure that the Bounds checking compiler configuration option is set (i.e. /B is specified on the command line or in the "F77L3.FIG" file).

4. Upon completion of Step 1 for the programs CD1PGM, CD2PGM, and ONEDPGM, type the

following:

```
386LINK <file>↲
```

For the program CD1PLOT ensure the LAHEY graphics file GRAPH3.LIB resides in the directory named C:\F77L3\LIB (created by the LAHEY installation program) , and type the following:

```
386LINK <file> -LIB \F77L3\LIB\GRAPH3↲
```

Upon completion of this step the executable file named <file>.EXE will be created.

The RIVICE program on the CD provided with this Manual is in the form of a source code master file, with assigned variables used in array dimensions. These variables are preset because the size of the executable program meets all the current model requirements but may need to be changed depending on such things as the increased number of reaches, nodes, and lateral in-flows, etc.

The ability to redefine these variables allows the user flexibility to match a particular modelling need to the memory available on any particular computer.

The process of replacing the variables in the *.FOR file with user-defined dimensions has been done through changes that need to be carried out in the COMMON blocks already embedded or external to the RIVICE program.

To replace these dimensions, replacement values are placed in the example attached file called through the main program using the following **FORTRAN INCLUDE** command:

INCLUDE 'ICEVAR.CMN'

This ICEVAR CMN is the common block external to the program and is as follows:

```
INTEGER DEPOPT
COMMON /ICE1/ BORDVOL(9001),COHESN,CN(9001),DTT,DEPOPT(2)
COMMON /ICE2/ DICE(9001),KRCH(9001),LEOPT(2),COHBRK,NSEG
COMMON /ICE3/ NFRT1T(20),NICES(9001),NICEST(9001),NISEG,NSEGT
COMMON /ICE4/ NTRL1T(20),QT(9001),THICKT(9001),TLE1T(20)
COMMON /ICE5/ NTRL1(20),NFRT1(20),XFRZ1(20),TLE1(20)
COMMON /ICE6/ VELT(9001),XFRZ1T(20),ZTT(9001),VDEP(2),VERODE(2)
COMMON /ICE7/ VFACTR(2),VOLSUB(30),VTRNT(9001),VOLIN,ZT(9001)
COMMON /ICE8/ VOLOUT(20),TIMED,IEVOL,POROSC(2),VTRN(9001)
COMMON /ICE9/ ZZK1TAN,ZZK2,itrace,vel(9001),NSNTOT
INTEGER RLOCBRG
COMMON /ICE10/ NBRGSW,RLOCBRG(5),KBRG(5),DAYSBR(5),BRIDTH(5)
INTEGER EROPT
COMMON /ICE11/ FTRLIM(2),EROPT(2),FRMAX(2),DIAICE(2)
REAL(4)
TEMPAIRT,HLC,HLOSS,VELOC,ICEVOLPREV,ICEVOLCUR,VOLNFRT1,USICEVOL
REAL(4) VOLNFRT0
REAL(4) TTINT,AIRTEMP,ICEG,USICESEG
INTEGER NTT,DEBUGICEGEN
COMMON /ICEGEN1/ DEBUGICEGEN, TEMPAIRT(50),USICEVOL(50)
COMMON /ICEGEN2/ TTINT,NTT,AIRTEMP,HLC,HLOSS(9001)
COMMON /ICEGEN3/ ICEG(9001),VELOC(9001),USICESEG
COMMON /ICEMOVE1/ICEVOLPREV(9001),ICEVOLCUR(9001),VOLNFRT1(20)
COMMON /ICEMOVE2/VOLNFRT0
```

4.4 DATA FORMATS

The data formats are currently in fixed format form and are provided in Section 7 along with the variable description and the size and location of the variable being entered. It is prudent not to use the "tab" key to enter data as this will be interpreted differently by differing compilers. Most keep the format "fixed" and expect a specific number of blanks between data points.

One most common error is to use special characters that are not normally recognized by the compiler and leads to a significant loss in time trying to find the error. Using a common text file editor, like NOTEPAD, provides a more secure means of locating and eliminating special characters. At times hidden characters can also be a problem which requires more than one text editor to find these problems.

5. RIVICE MODELLING PROCEDURE

5.1 GENERAL

The steps required to generate a model is composed of several simple steps that require user vigilance in assuring that the data is consistent and is as close to the physical representation of the problem as possible. The initial step is to divide the model into physical reaches that would be considered similar in hydraulic character. Once the river is divided into these similar parts, a process to examine this data using the **COORD1** system to examine each and every section for input errors or erroneous data by plotting each and every section. Once this has been accomplished, the user must prepare the data input to generate the hydraulic tables as well as cross sections that will be used by the model.

The following steps must include a calibration and verification phase as well as a sensitivity test to assure that the model is as "robust" as possible to carry out the simulations. A common problem is to create a complete model before each of the individual components are checked, run separately and verified with some physical flow or level data.

The following describes in more detail the actual procedure and the steps that are needed to produce a successful model application.

5.2 COMPONENTS OF A RIVICE MODEL

An overview of the basic components of a RIVICE model was provided in Section 1.2 of the Introduction. A more thorough discussion of the building blocks for the model is provided in this section. All fundamental rules for program application in this section are printed in **bold type.** Practical advice related to building, running and debugging a model is presented in Section 10.

5.3 REACHES & NODES

The most fundamental element of a RIVICE model is a reach. A reach can represent a river channel, canal, ditch, pond, lake, bridge or culvert waterway, or a floodplain storage cell. Usually, a reach represents a segment of a waterway that has similar hydraulic characteristics along its length. **The basic requirements of a reach are that it must be capable of conveying discharge under subcritical conditions, and that it always must contain at least some water.** If, anywhere along a reach, the calculated water level reaches the lowest bed elevation at any time step, the program will abort the simulation. The results of the simulation up until the time of failure are stored in the TAPE10.TXT file and, for the majority of such cases, in the TAPE6.TXT file also.

The endpoints of reaches are called nodes, and each reach always has two of them. When at least two reaches share a node, it is called a junction node. When this occurs, the two reaches will be hydraulically connected in the simulation, meaning that the water level at the node shall be the same for both reaches, and the quantity of discharge arriving at that node shall equal to the discharge leaving it.

Nodes which are not junction nodes are called terminal nodes. These occur at the model boundaries, or at control structures. Each terminal node must have either the water level or the discharge at the node defined by a boundary condition or have a stage-discharge curve or stage-routing boundary condition defined there.

By convention, the node number appearing first (ie. on the left hand side) in the reach-node connectivity table in the A Data Group of TAPE5.TXT is defined as the upstream end of the reach. Discharges entering the reach through this node will have a positive sign, and flow leaving the reach through the upstream node will have a negative sign. Discharges and velocities in the reach moving from the upstream node toward the downstream node will have a positive sign.

For calculation purposes by the finite difference method, each reach is subdivided into equal length segments called mesh spaces. The length of the mesh spaces can vary from one reach to another. The boundaries between adjacent mesh spaces are called mesh points, and it is for these point locations that the program computes water levels and discharges over a period of time.

A minimum of five mesh points must be specified for each reach. In general, more mesh points in a reach allow for higher resolution of results, but demand more computation time. If flow conditions are expected to change significantly along a reach, the mesh spacing should be set close

The physical and hydraulic characteristics of reaches are described to the program by hydraulic tables which are provided by the user and are based on channel cross sections. The program interpolates the hydraulic tables for locations between these cross section locations to create hydraulic tables for every mesh point in the system enough to capture sufficient detail.

5.4 INITIAL CONDITIONS

To proceed with the simulation of a hydraulic network, the model must be given all the estimated or known water levels and discharges throughout the system at the beginning of a simulation. It is important to provide initial conditions that represent a hydraulically stable situation, or a program crash may occur soon after the execution begins. In RIVICE, the initial conditions can be specified by the user in the TAPE5.TXT file or obtained from the output of an earlier simulation for the same network. For the first run of a network, the only source of data on initial conditions is the input data file, TAPE5.TXT. Initialization performed in this manner is referred to as a "cold start".

For the user-specified initial data case, this data is provided for each cross section, and the model interpolates the values for each mesh point prior to the solution of the first time step. For the simulations that start with data at a particular time step of an earlier run, a transfer of data for each mesh point is made. There are two alternative sources for this data, the TAPE10.TXT and TAPE17.TXT files from the earlier run. TAPE17.TXT stores the water level and discharge data calculated for the entire network for the last time step, and TAPE10.TXT stores the same data for every time step of the entire simulation period.

When it is intended to stop a simulation, restart it using the last set of computed values of the first run, the TAPE17.TXT file produced by the first run is renamed to TAPE16.TXT, and the appropriate option is specified in Data A-b of TAPE5.TXT. This will result in a "warm start" for the second part of the simulation, which will use the TAPE16.TXT data for initial conditions. This option is not available for a run that aborts prematurely, because the TAPE17.TXT file will be empty.

An alternative "warm start" method uses the TAPE10.TXT file. The advantage of this is that the user can specify any time step available on the TAPE10.TXT file as the starting point of the second run, and a TAPE10.TXT file is produced even when a run crashes. To select this option the user must specify the appropriate values on Data A-b and A-e. Occasionally, this technique suffers the disadvantage that a very large TAPE10.TXT file cannot be erased as long as there is a need to start the subsequent runs.

To qualify for a warm start, the basic network of reaches, nodes and the number of mesh points in every reach must remain unaltered between the earlier run and the warm start run. Changes may be made to QFC structures, inflow hydrographs, Manning's n values, lengths of reaches, time step duration, etc., but the one-to-one correspondence of mesh points from the earlier run to the later run must be preserved.

One type of data which is lost during a warm start is the discharge information through any QFCS at the first time step of the second run. Often, this does not have a significant effect on the results of a simulation and may be safely ignored. However, the user should be aware of this and apply judgement accordingly.

5.5 BOUNDARY CONDITIONS FOR HYDRAULIC CALCULATIONS

A boundary condition is a water level or discharge which occurs at an edge or limit of the system to be solved by the finite difference scheme. There are two types of boundary conditions: external and internal. External boundaries are those which connect to the world outside the model space. Internal boundary conditions occur at the two ends of control structures which are both tied to different locations in the network. These control structures cannot be simulated by the finite difference method, therefore are isolated by boundaries.

An example of an external boundary condition for a model of an estuary would be the tide level at the downstream end of an estuary. Examples of internal boundary conditions would be the upstream and downstream ends of a waterfall in a river. Boundary conditions may be constant, or they may vary with time, such as a time series representing the sea surface levels over several tide cycles. In a hydrodynamic model, the boundary condition must be provided continuously for the entire time period of the simulation.

As stated above, each terminal node must have its discharge or water level defined through a boundary condition. Conversely, **a boundary condition for the model can only be applied at a terminal node. Also, only one boundary condition may be applied to any terminal node.** The model will only yield a hydraulic solution if that solution will satisfy all the boundary conditions.

ONE-D offers the following alternative choices for boundary conditions:

- user-specified water levels (upstream or downstream end)
- user-specified discharges (upstream or downstream end)
- stage-routing boundary condition (downstream end only)
- stage-discharge rating curve (downstream end only)
- upstream or downstream end of a control structure

The first two types of boundary conditions listed above are self-explanatory. The rating curve boundary condition allows a user-specified stage-discharge relationship to be applied at the downstream end of a watercourse. This can also be used to define the boundary at the upstream end of a weir or other feature in the main channel which would cause the flow to approach critical depth. The program computes a new value of discharge every time step based on the water level that occurred there during the previous time step. In conditions where velocities are high and changes from one time step to another are large, the one time step lag may cause numerical instabilities.

A more stable alternative in such a case is offered by the stage-routing boundary condition, which computes a dynamic stage-discharge boundary condition based on the assumption that the river system beyond the downstream limit of the model is infinitely long and that no backwater effects exist downstream that would elevate the water surface above normal depth. In this case it is imperative that the user supply a realistic stage-discharge pair for the initial condition at this downstream node, since the stage-discharge relationship will be developed from that initial point. If the values for stage and discharge are unknown, the user should estimate an initial pair that is above, rather than below, normal depth at that cross section.

The RIVICE program allows a cyclic water level boundary condition of a given duration to be repeated automatically for a specified number of times in a simulation. Each repetition is referred to as a period. This is used to generate a simplified tidal cycle for an estuary; simplified because it would not contain the day-to-day changes that naturally occur in real tidal cycles.

An additional feature for this type of application is an option that allows the program to find a cyclic-stable solution, which is a stable hydraulic solution that is repeated from one cycle to the next. This is used to establish stable initial conditions that can then be used to "warm start" a run for a specified number of periods to test the effect of some change to the system.

Boundary conditions are defined in the F Data Group of TAPE5.TXT, although some overlap occurs with the lateral flow definition in the D Data Group.

5.6 BOUNDARY CONDITIONS FOR ICE

Boundary conditions for the ice processes are driven primarily by the conditions described for hydraulics in Section 5.5. However, two boundary conditions can be provided specifically for ice:

- Ice bridge or lodgements – These are specified to occur at specific times during the simulation and at specific locations (cross-sections).
- Incoming volumes of ice flowing into the upstream and of the reach.

These conditions are specified through Data Groups I-o / I-p and I-aa / I-ab, respectively.

5.7 LATERAL INFLOWS

Flows, which are added to or extracted from a reach at any location except the nodes, are referred to as lateral inflows. For example, these may represent minor tributary inflows where the tributary is not part of the model. In such a case the flows are specified by the user and are similar to external boundary conditions, except that the flows enter the model, not through a node, but through a location along the reach.

Lateral inflows may be applied at a point or evenly distributed along part of a reach. **Point lateral inflows must only be applied to locations along a reach that are exactly midway between adjacent mesh points.** It is up to the user to ensure that the correct distance is specified to locate the point inflow at the mid-mesh location, and to update this distance whenever the mesh spacing is altered in a reach.

Lateral inflows are defined in the D Data Group of the TAPE5.TXT file.

5.8 SELECTION OF TIME STEPS

The program achieves a dynamic solution by solving the equations governing fluid motion for the entire system at one instant in time, then solving them again for one increment of time, or time step, later. The process is repeated until the desired period of simulation is completed. Time steps are set by the user, and have a direct effect on the length of computation time required to complete a run. Time step length also has an effect on the stability and the accuracy of the numerical results generated by the program. Generally, shorter time steps yield a more stable and accurate simulation, but require longer execution times and more disk storage space for the solution. All subsequent post-processing of the output will require more time and disk storage space also.

The implicit finite difference solution scheme used by RIVICE allows relatively long time steps in comparison with time step limitations governing explicit schemes. As a rough guide for initial choice of time step, the user should estimate the highest velocity (v) and wave celerity (c) expected for a typical mesh space length (Δx) in a long reach of the model. An initial trial time step duration (t_i) can be estimated using the following formula:

$$t_i = 15 \frac{\Delta x}{(v + c)}$$

It has been found that some models are stable and yield reliable results using time steps of much longer duration than indicated by the above formula. The user should test the sensitivity of the model results to changes in time step duration before using the results for any practical purpose.

The orderly simulation of ice passage through the system would ideally require time steps that do no result in passage of ice beyond the length of one cross-section per time step. However, that is normally overly restrictive and the computer logic has been developed to allow ice transmission beyond one cross-section per time step. The user is cautioned that the further the model deviates from the ideal of ice movement within one cross-section in one time step, the greater will be the chance that instabilities and inaccuracies will occur.

5.9 SELECTION OF MESH SPACING

The most important aspect of choosing mesh spacing is that the model properly represents the changes in the channel. Generally speaking, the mesh spacing should be similar to the actual spacing of surveyed cross sections that are available as a basis of the channel simulation. It can, however, be more densely spaced so that the model channel represents gradual expansions and contractions in the channel geometry.

The ideal mesh spacing for the purposes of simulation of the ice cover evolution is believed to be not more than one river width. The simulation through bridges or large culverts can require much finer mesh spacing. It is conceivable that spacing as small as one to two meters are needed and this in turn will require a much smaller time-step. This finer time step can be below one minute to properly simulate rapid changes in these situations.

6. PRE-PROCESSING

6.1 X-SECTION (KPA VERSION)

PURPOSE

In an ONE-D model, a system of channels is described by a network of reaches interconnected at nodes. The hydraulic parameters of each reach are calculated from personalized software from a set of representative cross sections. Cross sections may be established from hydrographic charts, survey data or topographic maps from which the elevations and horizontal distance data are taken. In some instances, this information may already be available in the HEC-2 format.

The XSECTION program is a user program, developed by KPA Engineering, that converts channel cross section data from the common HEC-2 GR format to that required for the COORD1 program. Input data for the XSECTION program must conform to the fixed format specified (see Data Formats for ASCII Input Files) to prevent reading errors. These small programs do not have manuals but are simply adapted to situation where a significant number of input cross sections must be processed. In this case, the number of cross sections was in the one thousand value range.

This example by KPA Engineering takes and manipulates the HEC-2 cross sections that extend beyond the simulation range required for the study for input to the ONE-D model. Examples of this are cross sections through multiple channels, or cross sections extending beyond a dyke or levee. XSECTION allows the user to truncate any number of points from either end of the cross section while it is reformatting the data.

COMMAND SYNTAX

With the input files TAPE4.SEC and TAPE5.SEC present in the working directory and a path to the program XSECTION.EXE established, type the following at the DOS prompt:

XSECTION↵

INPUT FILES

Two ASCII input files are required to run the XSECTION program, TAPE4.SEC and TAPE5.SEC.

The input file TAPE4.SEC is made up of three types of Data, a title Data, a modified X1 Data and the GR Data. The title Data describes the watercourse to be modelled. The X1 Data is a header line for each cross section which pertains to the information to be found in the GR Data. This is not exactly the same format as required for HEC-2, and must be altered as described below. The GR Data contain the unaltered cross section data directly from a HEC-2 data file.

For subcritical flows, HEC-2 cross sections are organized starting at the downstream end and progressing upstream. It is advisable to reformat the data from upstream to downstream making sure the appropriate distances between cross sections are maintained. The alternative of leaving the data unchanged would lead to negative flows being computed from the ONE-D model. This negative discharge data is technically correct but may be difficult for some users to interpret.

The detailed data requirements for the TAPE4.SEC file are described as follows:

Title Data (one required for each TAPE4.SEC file)

6.2 COORD1

Purpose

The COORD1 program is used to prepare cross section data for further processing by the COORD2 program. One version of COORD1 named CD1PLOT.EXE will also create screen plots of all cross sections. The CD1PGM program carries out the same subroutines as CD1PLOT, but it does not provide screen plots of the cross sections.

Command Syntax

With the input file TESTCD1.TXT present in the working directory, at the DOS prompt, type:

```
CD1PLOT↵
```

for the LAHEY-compiled version of CD1PLOT.FOR, or:

```
CD1PGM↵
```

for the compiled version of CD1PGM.FOR.

Input File

One ASCII input file is required for input and it must be named TESTCD1.TXT. If the XSECTION program has been used to reformat HEC-2 cross section data, the output file TAPE10.SEC can be renamed to TESTCD1.TXT, and no additional changes are required.

When XSECTION is not used, the input file must be created manually. It is made up of six types of Data, 1-a to 1-f. The input requirements for each variable are listed below and additional explanation follows the listing.

6.3 COORD2

Purpose

The COORD2 program, run after COORD1, will produce a table of hydraulic parameters for each cross section created. The table includes water surface elevation, total and core top widths, total and core areas and wetted perimeter. These tables are used as input data corresponding to Data GROUP B in the TAPE5.TXT input file for the ONE-D program.

Command Syntax

With the input files TESTCD2.TXT, INPUT10.TXT and INPUT11.TXT in the working directory, at the DOS prompt type:

```
CD2PGM↵
```

Input Files

A total of one ASCII and two binary input files are required. The ASCII file must be created by the user and must be named TESTCD2.TXT. The two input binary files INPUT10.TXT and INPUT11.TXT, are output files created by the COORD1 program and require no changes. If the XSECTION program has been used to reformat HEC-2 GR Data, the output file TAPE11.SEC can become the input file TESTCD2.TXT after the user has defined the following variables:

```
DATUP, DATDN
```

```
DX(K)
FR1C3(J), FR1C4(J)
CL(J), CR(J)
H1(J), H2(J)
```

Any embayment areas would also require user input to TAPE11.SEC. The last step would be re-naming TAPE11.SEC to TESTCD2.TXT.

The input file TESTCD2.TXT can contain as many as nine types of Data, numbered from 2-A to 2-H. The file structure and input requirements for each variable are listed below:

NOTE: This page left intentionally blank

7. INPUT REQUIREMENTS

7.1 RIVICE INPUT FILE: TAPE 5

TAPE5.TXT is a text file that is the main input file for the RIVICE program. The data contained in this file represents the model structure of the network to be simulated, and usually describes all that the program requires to represent the network. There are only two cases in which input data is acquired from sources other than the TAPE5.TXT file. One of these would occur if and when the initial ice conditions are taken from a TAPE10.TXT or TAPE17.TXT file for a warm start, and the other when one or two sea dams are included in the model, requiring up to four additional input files named TAPE51.TXT through TAPE54.TXT. Currently ***only*** the **TAPE5.TXT** file can be used with the existing code.

The stages of model debugging, calibration, testing and simulation of various what-if scenarios each involves a cyclical procedure of adjusting data in the TAPE5.TXT file, running the model and reviewing the results. Therefore, the TAPE5.TXT file is typically the most frequently modified input file of all, and users should develop a system for organizing their TAPE5.TXT files. The most recent version of the RIVICE program allows user-specified filenames to replace the TAPE5.TXT filename, and this feature can be used to maintain organization of the individual simulations performed for a project.

The TAPE5.TXT file is organized into seven basic units, called Data groups. Every one of these groups requires at least some data. The nine data groups, their prime functions and the individual "Data," or lines of data, within each group are listed below in the same order that they belong in a TAPE5.TXT file:

Index of Data Types used in TAPE5.TXT

A Data Group	***provides basic job control options, time parameters and network topology (reach-node connectivity table)***
A-a	Title Data
A-b	Hydraulic solution options
A-c	Time parameters
A-f	Network parameters
A-g	Reach-node connectivity Data
A-h	Control structure identification Data
B Data Group	***describes physical and hydraulic parameters of each reach, including hydraulic tables for each cross section***
B-a	Reach title Data
B-b	Reach characterization Data
B-c	Reach parameters
B-d	Elevation table parameters
B-e	Cross section geometry parameters
B-f	Irregular cross section title Data

B-g	Geometry for cross section with constant top width
or	
B-g*	Hydraulic table data for cross section with variable top width

C Data Group	***Description of meteorological data required only if JOPT(2) has been set (Current RIVICE option tested is temperature)***
C-a	Water quality identification
C-b	Water quality parameter coefficients by parameter: network specification
C-c-s	Override data for salinity
C-c-T1	Override data for temperature and meteorological conditions
C-c-T2	Override data for salinity
C-c-BOD	Override data for BOD
C-C-Nutr1	Override data - nutrients
C-C-Nutr1	Nutrient coefficient override data
C-c-DO	Override data for DO
C-c-FCOL	Override data for fecal coliforms
C-c-DLIG	Override data for decaying lignins
C-d-1	Water quality reach data - mesh points
C-d-2 etc.	Mesh point location data
C-e-1	Reach override identification
C-e-2	Parameter identification data
C-f-1	Initial condition for reach by parameter
C-f-2	Initial condition table

D Data Group	***lists all lateral inflows, including QFC structures that have at least one end as a lateral inflow***
D-a	Lateral inflow identification Data
D-b	Number of lateral inflows
D-c	Input lateral inflow parameters
D-d	Input lateral inflow
D-f	WSC discharge data input
D-c*	QFCS computed lateral inflows
D-d*	QFC structure description
D-e*	QFC structure input - upstream side
D-f*	QFC structure input - upstream side

D-g*	QFC structure input - upstream side
D-e**	Pump station parameters
D-f**	Pump station emergency and remote switch settings
D-g**	Pump switch settings
D-h**	Pump parameters
D-I**	Pump intake and discharge pipe parameters
D-j**	Pump curve data

E Data Group	***Description of input data required only if JOPT(2) has been set (current RIVICE option tested is temperature)***
E-a	Injection data identification
E-b	Number of injection points
E-c-1	Injection parameters
E-c-2	Injection parameters: water quality names
E-d	Injection data

F Data Group	***lists all boundary conditions, including QFC structures that have both ends connected to terminal nodes***
F-a	Identify Data for hydraulic boundary conditions
F-b	Node parameters
F-c	Boundary node conditions
F-d	Boundary node condition
F-e	WSC elevation data input
F-f	WSC discharge data input
F-b*	QFC structure at node
F-c*	Last Data of rating curve for shift option

G Data Group	***lists user requests for specific output data as locations for hydrographs and time steps for profiles***
G-a	Identification Data
G-b	Number of hydrographs
G-c	Hydrograph parameters
G-d	Number of hydraulic profiles
G-e	Profile parameters

H Data Group	***Description of input data required only if JOPT(2) has been set (current RIVICE option tested is temperature)***
H-a	Identity data for water quality boundary conditions
H-b	Water quality node parameters
H-c	Constant or variable boundary conditions
H-d	Time constant data for ocean boundary conditions
H-e	Water quality constituent data for ocean boundary conditions
H-f	Water quality graphs and profile output – identity data
H-g	Number of quality graphs
H-h	Water quality graph parameters
H-i	Number of water quality profiles
H-j	Water quality profile parameters

I Data Group	*defines parameters for the simulation of ice*
I-a	Ice Description Option
I-b	Ice Deposit Velocity
I-c	Ice Deposit Velocity for DEPOPT=1
I-d	Ice particle diameter
I-e	Ice erosion option
I-f	Ice deposit Option
I-g	Tractive force for ice cover erosion
I-h	Option number for evaluation of leading edge stability
I-j	Ice transport speed factor
I-k	Ice cover porosity
I-l	Slush ice porosity
I-m	Slush ice pan thickness
I-n	Cohesion of ice cover to riverbanks
I-o	Number of user invoked ice bridges
I-p	Ice bridge information
I-q	Simulation stop instruction
I-r	Output controls – general
I-s	Output controls – time based
I-t	Output controls – increments in cross sections for general output
I-u	Output controls – for specific time steps
I-v	Output controls – based on increments in leading edge progression

I-x	Output controls – for specific points of leading edge progress
I-y	Ice generation method
I-z	Heat loss coefficient if ICEGENMETHOD=2 (see Section 4_)
I-aa	Controls for specification of ice volumes inflowing to the study reach
I-ab	Controls for specification of ice volumes inflowing to the study reach
I-ac	Description Data to display the end of the Input Type I
J Data Group	***Constraints and Miscellaneous Information for Ice***
J-a	Ice cover strength parameters
J-b	Method to estimate Manning n-value of ice under surface
J-c	Manning n-value information for each cross section
J-d	Border ice prediction method (Refer to Section 2.2.2)
J-e	Time to start Border Ice Generation
J-f	Border ice breakup triggers
J-g	Border ice parameters for Newbury method (if IBORD=2)
J-h	Border ice parameters for user-defined method (if IBORD=1)
J-i	Border ice parameters for Matousek Method (if IBORD=3)
J-j	Controls for Ice Cover Melting
J-k	Heat transfer coefficient (if MELTOPT=1)

The input data is not referenced by Data identifiers, and the RIVICE program interprets what each piece of data represents by counting lines. Users are cautioned that missing Data or extra Data, such as an extra blank line, in TAPE5.TXT will cause errors that may or may not result in premature termination of the run. Detailed data requirements and formats, and explanations of what each input variable represents, are presented on the following pages in the same order that the Data appear in the TAPE5.TXT file:

A DATA GROUP

Data A-a Title Data (one required)

Variable	*Column*	*Format*	*Description*
HEADER	01-80	20A4	Description of run, identifying output for later reference. For convenient labelling of plots by the BROWSE and EXPORT post-processors, the river name or main title should start in column 16.

Data A-b Hydraulic Solution Options (one required)

Variable	*Column*	*Format*	*Description*
IOPT(1)	01-10	I10	1, hydraulic solution computations executed 2, only water quality solution computations executed
IOPT(2)	11-20	I10	1, hydraulic steady state or cyclic-stable solution to be found by program (see NPER, MAXITR and EPS on Data A-e, and "Boundary Conditions" in Section 3.1.3) 2, hydraulic transient solution
IOPT(3)	21-30	I10	1, hydraulic solution written on TAPE10.TXT (required for any post-processing) 2, hydraulic solution not stored (seldom used for current hardware)
IOPT(4)	31-40	I10	1, river network (indicates boundary conditions to be provided for entire period of simulation) 2, estuary network using simplified tidal cycles (indicates tidal boundary condition to be repeated for each tide cycle)
IOPT(5)	41-50	I10	1, hydraulic initialization read from TAPE10.TXT for a warm start run 2, hydraulic initialization read from input Data for a cold start run 3, hydraulic initialization taken from TAPE16.TXT for a warm start run (see discussion under "Initial Conditions" in Section 3.1.2)
KOPT	51-60	I10	1, input data given in Imperial units only, output will be produced in both Imperial and Metric units 2, input data given in Metric units only, output will be produced in Metric only (default is 1)

Data A-c Water quality solution options

Variable	*Column*	*Format*	*Description*
JOPT(1)	01-10	I10	1, solution computations executed 2, solution computations deleted
JOPT(2)	11-20	I10	1, steady state solution 2, transient solution
JOPT(3)	21-30	I10	1, solution executed 2, solution storage deleted NOTE: The computer program assumes that if the hydraulic solution is deleted and the water quality solution is executed, then the hydraulics necessary for the water quality solution will be read from storage. Nevertheless, Data groups F and G must be removed as indicated in Table 1.

Data A-d-1 Water quality parameter options

Variable	*Column*	*Format*	*Description*
NPARM	01-10	I10	= total number of water quality parameters being modelled, whether calculated or read-in from a previous calculation.

Data A-d-2 Parameter options

Variable	*Column*	*Format*	*Description*
WQPAR(I)	01-10	A4,6X	= abbreviation of water quality parameter as given in Table A-1 (start in Data column 1).
INOP	11-20	I10	= 0 or blank, parameter is calculated = 1, parameter is read-in (temperature and BOD only) = 2, parameter is of constant concentration as specified by initial conditions (temperature and BOD only)
OUTOP	21-30	I10	= 0 or blank, no offline storage or output = 1, output stored on sequential output file
DOCALC	31-40	I10	= blank except for dissolved oxygen (DO) = 1, DO calculated as a function of BOD alone = 2, DO calculated as a function of nutrients alone = 3, DO calculated as a function of both BOD and nutrients

TABLE A-1

WATER QUALITY PARAMETER ABBREVIATIONS

(Nutrients as a group) (Data A-d-2)

Abbreviation	Parameter
S	Salinity
T	Temperature
BOD	Biochemical oxygen demand
NUTR	Nutrients: 1) organic nitrogen 2) inorganic nitrogen 3) organic phosphate 4) inorganic phosphate 5) phytoplankton 6) zooplankton
DO	Dissolved oxygen
FCOL	Fecal Coliforms
CLIG	Lignins - conservative
DLIG	Lignins - decaying

TABLE A-2

COMPLETE SYMBOLIC IDENTIFICATION OF WATER QUALITY PARAMETERS AND SEQUENCE OF IDENTIFICATION

Abbreviation	Parameter
S	Salinity
T	Temperature
BOD	Biochemical oxygen demand
ON	Organic nitrogen
N	Inorganic nitrogen
OP	Organic phosphate
P	Inorganic phosphate
CP	Phytoplankton
CZ	Zooplankton
DO	Dissolved oxygen
FCOL	Fecal coliforms
CLIG	Lignins - conservative
DLIG	Lignins - decaying

Data A-e Time Parameters (one required)

Variable	*Column*	*Format*	*Description*
NPER	01-10	I10	Number of periods the solution is to propagate. Specify 1 for the steady state or cyclic-stable solution option, or in the case of a river system. Values greater than 1 are used with the simplified tidal estuary (IOPT(4)=2)
NINC	11-20	I10	Number of time steps to be executed within each period for the hydraulic model. (See Data B-c and discussion under "Time Steps" in Section 3.1.8)
PERIOD	21-30	F10.0	Length of time period (NPER) in seconds. This is not the time step, which is equal to PERIOD/NINC. In estuaries using simplified tide cycles, it is convenient to use the tidal period. For river systems, this is the total time being modelled
RATIO	31-40	F10.0	Ratio of the water quality time increment to the hydraulic time increment (May be left blank if no water quality computation is to be done)
MAXITR	41-50	I10	Maximum number of periods allowed for the program to try to compute a cyclic-stable initial condition (Only for IOPT(2)=1).
EPS	51-60	F10.5	The maximum allowed change in the discharges at each mesh point from one tidal point to the next. This defines a steady state or cyclic-stable initial condition (Only for IOPT(2)=1)
LPER	61-70	I10	Number of lead-in periods of hydraulics to be read from TAPE10.TXT, before solution starts for estuary network using simplified tidal cycles. This parameter is only used for extend-ing the file containing hydraulic solution on TAPE10.TXT.
NRINC	71-80	I10	For a warm start run using TAPE10.TXT, this is the number of the time step of the earlier run at which the later run will start (see discussion in Section 3.1 under "Initial Conditions").

Data A-f Network Parameters (one required)

Variable	*Column*	*Format*	*Description*
NREACH	01-10	I10	Number of reaches in the network. If more than 244 reaches are used, the BUILD post-processor cannot be used.
NNODE	11-20	I10	Number of nodes in the network
NCTR	21-30	I10	Number of control structures (other than QFC structures, such as waterfalls) defined on Data A-h.

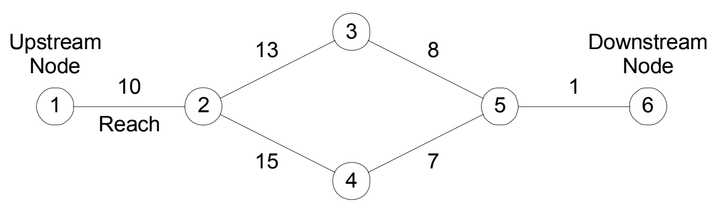
Data A-g Reach-node Connectivity Data (one required for each reach)

Variable	*Column*	*Format*	*Description*
K	01-10	I10	Sequence number of reach numbers (must start at 1 and increment by 1 for every reach)
IRCH(K)	11-20	I10	Numerical identification of the reach to which the information applies. This can be any integer as long as it is a unique reach number.
IREACH(K,1)	21-30	I10	The node number of the upstream end of the reach. This is the end from which the distances are measured.
IREACH(K,2)	31-40	I10	Node number of the downstream end of the reach

NOTE: The nodes in any network may be given any unique number.

Reach-node connections are defined as shown in the following example:

K	IRCH	IREACH(K,1)	IREACH(K,2)
1	10	1	2
2	13	2	3
3	15	2	4
4	8	3	5
5	7	4	5
6	1	5	6



Data A-h Control Structure Identification Data (optional, one required for each control structure)

Variable	*Column*	*Format*	*Description*
ICRS(I,1)	01-10	I10	Node number at upstream end of control structure
ICRS(I,2)	11-20	I10	Node number at downstream end of control structure

This Data must be repeated according to the number of control structures, NCTR, (excluding QFC structures) indicated on Data A-f.

B DATA GROUP

Data B-a Reach Title Data (one required for each reach)

Variable	*Column*	*Format*	*Description*
HHEAD(K,1)	01-80	20A4	Descriptive identification of reach. This must begin with the letter B in column 1 for the BUILD program to be able to find the necessary reach data. For convenient labelling of plots by the BROWSE and EXPORT post-processors, the main reach title should start in column 16.

Data B-b Reach Characterization Data (one required for each reach)

Variable	*Column*	*Format*	*Description*
JK	01-10	I10	Numerical identification of the reach (corresponding with IRCH(K) OF Data A-g) to which the information applies. The reaches must be in the same order as that specified in the reach-node connectivity table
IS(K)	11-20	I10	Specifies the shape of the channel cross section within the reach: 1, irregular 2, rectangular 3, trapezoidal 4, circular
IP(K)	21-30	I10	Indicates type of channel: 1, prismatic channel along the length of the reach 2, non-prismatic channel (varying width and/or depth)
ISL(K)	31-40	I10	Indicates source of bottom slope information: 1, bottom slope of reach is constant and specified by user (for option to use single cross section for entire reach) 2, bottom slope is variable and computed from bottom elevations at each section

Data B-b Reach Characterization Data (one required for each reach)

Variable	*Column*	*Format*	*Description*
IF(K)	41-50	I10	Indicates whether entrance and exit loss coefficients will be used: 1, Manning's coefficient specified by user, but no entrance and exit loss coefficients are used 2, Program to determine equivalent Manning's 'n' based on entrance and exit loss

Variable	*Column*	*Format*	*Description*
			coefficients for a bridge or culvert reach. (See COEFF(1,K) and COEFF(2,K) on Data B-c)
ICE(1,K)	51-55	I5	Percentage of ice cover change per time step, x 100
ICE(2,K)	56-60	I5	Percentage of initial ice cover, x 100
ICE(3,K)	61-65	I5	Time lag for ice cover change to begin, given by number of time steps from time zero
IDTABL(K)	66-70	I5	Specifies level of output to be included in TAPE6.TXT: 1, include all input and interpolated cross section tables 0, print of all input and no interpolated cross section tables -1, include no input cross section tables except first section of each reach and no interpolated cross section tables

Data B-c Reach Parameters (one required for each reach)

Variable	*Column*	*Format*	*Description*
SL(K)	01-10	F10.5	Bottom slope of the channel expressed as a fraction, $\Delta V/\Delta H$, if it is to be specified. Can be left blank if ISL(K)=2
SS(K)	11-20	F10.5	Side slope for a trapezoidal channel expressed as $\Delta V/\Delta H$, This can be left blank if $IS(K)\neq3$.
XMANN	21-30	F10.4	Manning's 'n' at lowest hydraulic table elevation, H1 (K) in Data B-d, for the reach (variable FRIC1 in COORD2 program) See discussion in Section 2.3, and refer to Figures 2-7 and 2-8 and to Table 2-1.
BMANN	31-40	F10.4	Manning's 'n' at channel bottom (variable FRIC2 in COORD2 program) (Default: BMANN is set equal to XMANN when not specified)
TL(K)	41-50	F10.3	Total length of the reach from upstream to downstream node
DX(K)	51-60	F10.3	Initial estimate of the computational mesh space length in reach K. There must be a minimum of five mesh points, counting the nodes, in a reach. Final value of DX(K) is computed in the program to make sure that the reach length is divided into exactly equal increments. The number of mesh spaces, TL(K)/DX(K), must be an even number. DX(K) should be compatible with the hydraulic time step (See discussion under "Time Steps" in Section 3.1.8).
COEFF(1,K)	61-70	F10.3	Entrance or contraction loss coefficient for a bridge, culvert or sea dam reach.

Variable	*Column*	*Format*	*Description*
COEFF(2,K)	71-80	F10.3	Exit or expansion loss coefficient for a bridge, culvert or sea dam reach.

Data B-d Elevation Table Parameters (one required for each reach)

Variable	*Column*	*Format*	*Description*
NS(K)	01-10	I10	Number of cross sections in the reach at which geometric information is provided. It must be at least 1 if ISL(K) = 1 (Data B-b). It must be at least 2 if ISL(K) = 2.
IIZ(K)	11-20	I10	Number of rows of data in elevation tables of geometric parameters. This number should represent coverage of the entire range of expected water levels using an elevation increment that would minimize interpolation errors for width, area and wetted perimeter.
H1(K)	21-30	F10.5	Minimum elevation for the hydraulic tables for reach K in the case where the tables are to be calculated for a rectangular, trapezoidal, circular, or constant top width irregular cross section. This is the minimum expected water surface elevation for the reach
H2(K)	31-40	F10.5	Maximum elevation for the hydraulic tables for reach K in the same cases. This is the maximum expected water surface elevation
ADJ1	41-50	F10.5	Upstream cross section table adjustment (can be positive or negative in value). This parameter effectively raises or lowers the hydraulic tables at the upstream end of a reach, but does not alter the minimum bed elevation. The interpolation tables are adjusted in proportion to the distance from the upstream end. ADJ1 and ADJ2 may be used with Data Group B-g* to raise or lower elevation tables to account for reference datum error, bed degradation, aggradation (i.e. HEAD(II) +ADJ) = new value of HEAD(II)).
ADJ2	51-60	F10.5	Downstream cross section table adjustment (can be positive or negative in value). See ADJ1 above.
ADJ3	61-70	F10.5	Upstream bottom elevation adjustment (can be positive or negative in value). This variable raises or lowers the minimum bed elevation at the upstream end of the reach. It does not affect any table values, such as area. The bottom elevations at interpolated sections are also adjusted in proportion to distance.
ADJ4	71-80	F10.5	Downstream bottom elevation adjustment (can be positive or negative in value). See ADJ3 above.

Variable	*Column*	*Format*	*Description*

An internal table of parameters as a function of elevation will be generated. IIZ is the number of entries in this table and should be a number sufficiently large to permit reasonable interpolation of values for the range of active surface water elevation computations. H1(K) and H2(K) represent the expected minimum and maximum water surface elevations

Data B-e Cross Section Geometry Parameters (one required for every input cross section)

Variable	*Column*	*Format*	*Description*
TLX(JJ)	01-10	F10.5	Distance from upstream end to cross section J in reach K. (J starts at 1 to NS(K))

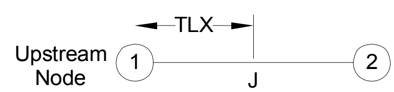


Fig B-e

Variable	*Column*	*Format*	*Description*
BW(JJ)	11-20	F10.5	Bottom width at TLX(JJ). Supplied for rectangular and trapezoidal cross section shapes. Fill in if IS(K)=2 or 3, otherwise leave blank
BEL(JJ)	21-30	F10.5	Bottom elevation at TLX(JJ). Required when bottom slope is not specified, and required at final cross section when slope is specified
R(JJ)	31-40	F10.5	Pipe radius at TLX(JJ). Leave blank if channel shape is not circular. (At present, the program allows only a constant radius pipe)
Z(JJ)	41-50	F10.2	Estimate of initial water surface elevation at TLX(JJ). This value is not adjusted by the factors ADJ1-ADJ4.
Q(JJ)	51-60	F10.1	Estimate of initial discharge at TLX(JJ)
FRIC(1,JJ)	61-70	F10.4	Manning's 'n' at maximum hydraulic table elevation H2(J) for the section (optional)
FRIC(2,JJ)	71-80	F10.4	Manning's 'n' at minimum hydraulic table elevation H1(J) for the section (optional)

Data B-e, B-f, B-g and B-g* constitute the cross section sub package, which must be supplied for each data cross section indicated by the parameter NS(K). For prismatic cross section of regular geometric shape (IP(K)=1) only Data B-e is

Variable	*Column*	*Format*	*Description*

necessary. Data B-f, and either B-g or B-g* are supplied only if the cross section shape is specified as irregular (IS(K)=1).

Data B-f Irregular Cross Section Title Data (one required for every input cross section)

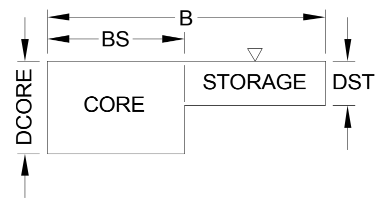
Variable	*Column*	*Format*	*Description*
ITW	01-10	I10	1, for constant top width 2, for variable width
HEADER(I)	11-80	17A4,A2	Identification, location or description of cross section

Data B-f is supplied only if the cross section shape is specified as irregular, (Data B-b, IS(K)=1)

Data B-g Geometry for Cross Section with Constant Top Width (one required for every input cross section)

Variable	*Column*	*Format*	*Description*
B	01-10	F10.5	Total topwidth in feet or metres
BS	11-20	F10.5	Core topwidth in feet or metres
DST	21-30	F10.5	Equivalent depth of storage area
DCORE	31-40	F10.5	Equivalent depth of core area

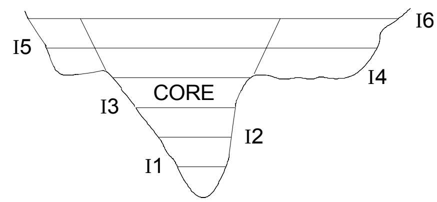
Data B-g is supplied only if IS(K)=1 (Data B-b) and ITW=1 (Data B-f). Refer to **figure below** for definition of section parameters.



Data B-g* Hydraulic Table Data for Cross Section with Variable Top Width(one required for each elevation in hydraulic table)

Variable	*Column*	*Format*	*Description*
HEAD(II)	01-10	F10.5	Water surface elevation entry I for cross section J in reach K, where I ranges from 1 to IIZ(K), (Data B-d)
TW(II)	11-20	F10.5	Total topwidth for entry I
CW(II)	21-30	F10.5	Core width for entry I

Variable	*Column*	*Format*	*Description*
AREA(II)	31-40	F10.5	Core area for entry I
WPERM(II)	41-50	F10.5	Wetted perimeter of core area for entry I
TAREA(II)	31-40	F10.5	Total cross-sectional area for entry I (core area plus storage area). Default: TAREA = AREA



Data B-g* is supplied only if IS(K)=1 (Data B-b) and ITW=2 (Data B-f). This Data is to be repeated, corresponding to the number of elevation entries indicated by IIZ(K), (Data B-d). These Data must be arranged in order of increasing depth.

DESCRIPTION OF DATA GROUP - C

As the water quality parameter possibilities consist of as many as 13, the form of definition has been designed to give the user flexibility in specifying coefficients, mesh point locations and initial conditions. The parameter coefficients can be specified at two levels. The first is for the entire network, the second is for an individual reach. Default values (Table C-1) may be used or the user can override default values at either level.

TABLE C-1

DEFAULT METEOROLOGICAL CONDITIONS

Symbolic Name	Default Value	Description
ATM(IMC)	0.0	Time from the beginning for entry IMC in hours
ATAMB(IMC)	60.0	Ambient temperature in degrees F
ARELG(IMC)	75.0	Relative Humidity (%)
AW2(IMC)	10.0	Wind velocity a 2 m (mph)
ARFS(IMC)	1800.0	Net solar flux (BTU/ft²/day) $_{sf} = _{s} - _{sr}$ where $_{s}$ = incident solar flux $_{sr}$ = reflected solar flux
ARFA(IMC)	2500.0	Net atmospheric flux (BTU/ft²/day) $_{as} = _{a} - _{ar}$ where $_{a}$ = incident atmospheric flux $_{ar}$ = reflected atmospheric flux
APRESS(IMC)	760.0	Atmospheric pressure in mm Hg

TABLE C-2

DEFAULT QUALITY CONDITIONS

Symbolic Name	Default Value	Description
		B.O.D.
KB20	0.7	Decay coefficient (day^{-1}) in the equation $KBOD = KB20 \times QT^{(T-20)}$
QT	1.047	Empirical coefficient in above equation
		D.O.
KD20	10.8	Re-aeration coefficient in the equation: $KDO = KD20(V^{0.60}/H^{1.40})QT^{(T-20)}$ where V = absolute velocity H = depth in units of day^{-1} (base e)
QT	1.016	Empirical coefficient in above equation
		Fecal Coliforms
KFCOL20	2.8	Decay coefficient at 20 C, where: $KFCOL = KFCOL20 \times QT^{(T-20)}$ in units of day^{-1} (base e)
QT	1.045	Empirical coefficient in above equation
		Decaying Lignins
KLIG20	0.115	Decay coefficient in equation: $KLIG = KLIG20 \times QT^{(T-20)}$ in units of day^{-1} (base e)
QT	1.045	Empirical coefficient in above equation

TABLE C-3

DEFAULT NUTRIENT COEFFICIENTS

Symbolic Name	Default Value	Description
EL	1.00	Light efficiency factor for zooplankton grazing on phytoplankton
FIP	0.500	Fraction of phosphate in inorganic form in zooplankton excretion
FIN	0.900	Fraction of nitrogen in inorganic form in zooplankton excretion
FP	0.015	Fraction of total phosphorus in biomass
FN	0.100	Fraction of total nitrogen in biomass
IS	0.040	Optimum light intensity for phytoplankton growth
KCV	0.080	Zooplankton food assimilation coefficient
KDP2	1.000	Maximal value of KDP at 20 C (day^{-1})
KDZT	0.020	Coefficient for zooplankton death through toxicity $(day*toxicity\ unit)^{1}$
KDZ2	0.040	Maximal value of KDZ at zero toxicity and 20 C (day^{-1})
KEL1	0.050	Natural water light extinction coefficient
KEL2	0.060	Phytoplankton self-shading factor $(mg/L)^{-1}$
KGP2	2.000	Maximal phytoplankton growth coefficient at 20 C (day^{-1})

KOP2	0.300	Organic phosphate decay coefficient at 20 C (day^{-1})
KON2	0.300	Organic nitrogen decay coefficient at 20 C (day^{-1})
KRP2	0.050	Phytoplankton respiration coefficient
KRZ2	0.015	Zooplankton respiration coefficient at
MMCP	1.000	Michaelis-Menten coefficient for
MMCZ	0.150	Michaelis-Menten coefficient for zooplankton removal by grazing (mg/L)
MMN	0.030	Michaelis-Menten coefficient for inorganic phosphorus limiting factor in phtyoplankton growth rate (mg/L)
MMP	0.030	Michaelis-Menten coefficient for inorganic phosphorus limiting factor in phytoplankton growth rate (mg/L)
QT	1.050	Constant for coefficient dependence on temperature
ZFCV	0.50000	Zooplankton food calorific value factor
TLM	infinity	Median tolerance limit for organism survival in a waste water, i.e. the concentration of waste water at which the organism will have a 50% survival after 96 hours
SP	0.0	Inorganic phosphate sink term to account for sedimentation (mg/L*day)

C DATA GROUP

This data group must be omitted if the water quality computations are deleted, JOPT(1) = 2.

Data C-a Water quality identification

Variable	*Column*	*Format*	*Description*
HEADER	01-80	20A4	Water quality description of the reaches

Data C-b Water quality parameter coefficients by parameter: network specification

Variable	*Column*	*Format*	*Description*
WQPAR(I)	01-10	A4,6X	Abbreviation of the parameter (S, T, BOD, NUTR, DO, FCOL, DLIG, of Table A-1)
KEY	11-20	I10	0 or blank, no network specification, default values are taken for the entire network (see Table C-1). These values can be overridden by reach. 1, the specification is by network subject to override by reach. NOTE: One data for each parameter being calculated. If network specification is selected follow by a parameter subgroup. Omit this data for Conservative Lignins

Data C-c-s Override data for salinity

Variable	*Column*	*Format*	*Description*
DISP	01-10	F10.0	Salinity region dispersion parameter, in ft^2/s, as

Variable	*Column*	*Format*	*Description*
			described in section 3.
REFS	11-20	F10.0	Salinity region reference salinity, S_o, in ppm (refer to section 3).
REFL	21-30	F10.0	Salinity region reference length, L, in ft (refer to section 3).

Data C-c-T1 Override data for temperature and meteorological conditions

Variable	*Column*	*Format*	*Description*
NMC	01-10	I10.0	Number of meteorological time entries. Specify 1 for constant conditions.

Data C-c-T2 Override data for salinity

Variable	*Column*	*Format*	*Description*
ATM(IMC)	01-10	F10.2	Time from the beginning for entry IMC (in hours)
ATAMB(IMC)	11-20	F10.2	Ambient temperature (in degrees Fahrenheit)
ARELH(IMC)	21-30	F10.2	Relative humidity (in percent)
AW2(IMC)	31-40	F10.2	Wind velocity at 2 m (in miles/hour)
ARFS(IMC)	41-50	F10.2	Net solar flux (in BTU/ft²/day) $= f_s - f_{sr}$ where f_s = incident solar flux f_{sr} = reflected solar flux
ARFA(IMC)	51-60	F10.2	Net atmospheric flux (in BTU/ft²/day) $= f_a - f_{ar}$ Where f_a = incident atmospheric flux f_{ar} = reflected atmospheric flux
APRESS(IMC)	61-70	F10.2	Atmospheric pressure (in mm Hg) NOTE: NMC data required, where IMC is the data

Data C-c- BOD Override data for BOD

Variable	*Column*	*Format*	*Description*
BOD	01-10	A3,7X	Symbolic name
KB20	11-20	F10.0	B.O.D. decay coefficient (in day^{-1} to the base e) in the equation: $KBOD = KB20 \times QT^{(T-20)}$
QT	21-30	F10.0	Empirical coefficient Default values are KB20 = 0.7 QT= 1.047

Data C-c- NUTR1 Override data - nutrients

Variable	*Column*	*Format*	*Description*
NUTOR	01-10	I10	Number of overridden coefficients to be

Variable	Column	Format	Description
			specified on the following lines (one per line)

Data C-c- NUTR2 Nutrient coefficient override data

Variable	Column	Format	Description
SYM	01-10	A4,6X	The symbolic name of the nutrient coefficient as specified in Table C-3
VALUE	11-20	F10.0	The new value of the coefficient NOTE: One data per coefficient to be overridden is required.

Data C-c- DO Override data for DO

Variable	Column	Format	Description
DO	01-10	A4,6X	Symbolic name
KD20	11-20	F10.0	Reaeration coefficient (in day^{-1} to the base e) in the expression: $KDO = KD20(V^{0.60}/H^{1.14})QT^{(T-20)}H(B/A)$ Where B = total top width A = total area Default value is 10.8 day^{-1}, base e, by Bansal, M.K. (1973)
QT	21-30	F10.0	Empirical coefficient in the above equation Default value is 1.016

Data C-c- FCOL Override data for fecal coliforms

Variable	Column	Format	Description
FCOL	01-10	A4,6X	Symbolic name
KFCOL20	11-20	F10.0	Decay coefficient at 20°C (in day^{-1} to the base e) in the expression: $KFCOL = KFCOL20 \times QT^{(T-20)}$ Default value is 2.8 day^{-1}, base e
QT	21-30	F10.0	Empirical coefficient in the above equation Default value is 1.045

Data C-c- DLIG Override data for decaying lignins

Variable	Column	Format	Description
DLIG	01-10	A4,6X	Symbolic name
KLIG20	11-20	F10.0	Coefficient KL20 (in day^{-1} to the base e) in the expression: $KLIG = KL20 \times QT^{(T-20)}$ Default value is 0.115 day^{-1}, base e (sulphite)
QT	21-30	F10.0	Empirical coefficient in the above equation

Variable	Column	Format	Description
			Default value is 1.045

Data C-d- 1 Water quality reach data - mesh points

Variable	Column	Format	Description
REACH	01-10	10X	(leave blank)
K	11-20	I10	Numerical identification of the reach. (These should be in the order specified in data A-g)
MESHPT(K)	21-30	I10	Number of mesh points for each reach K. For each reach the user must define the locations at which the finite difference calculation is to be made. The ability to have a non-uniform mesh point grid enables the user to have a closer spacing at locations of steep concentration gradients such as outfalls and confluences, as compared with locations of relatively small changes in concentration. NOTE: One data for each reach followed by other reach data is required.

Data C-d- 2 etc. Mesh point location data

Variable	Column	Format	Description
X(I)	01-10	F10.0	Location of mesh point from upstream node
X(I+1)	11-20	F10.0	Location of mesh point from upstream node
X(I+2)	21-30	F10.0	Location of mesh point from upstream node
X(I+3)	31-40	F10.0	Location of mesh point from upstream node
X(I+4)	41-50	F10.0	Location of mesh point from upstream node
X(I+5)	51-60	F10.0	Location of mesh point from upstream node
X(I+6)	61-70	F10.0	Location of mesh point from upstream node
			As many data as necessary of the above format should be prepared (with 7 items per data). The values should be in numerical order. The first value, X(1), must be 0. and the last value, X(MESHPT(K)), must be equal to TL(K), the total length of the reach as defined in Data B-c.

Data C-e- 1 Reach override identification

Variable	Column	Format	Description
OVERRIDES	01-10	10X	OVERRIDES
K	11-20	I10	Reach identification number of this reach
NMPAR	21-30	I10	Number of parameters whose network specified coefficients are being overridden. (Nutrients are considered one parameter in this case).

Data C-e- 2 Parameter identification data

Variable	*Column*	*Format*	*Description*
WQPAR(I)	01-10	A4	Abbreviation of the parameter (S, T, BOD, NUTR, DO, FCOL, DLIG) NOTE: This data must be followed by the redefinition of the coefficients using data of format C-c-S, C-c-T, C-c-BOD, C-c-NUTR, C-c-DO, C-c-FCOL, C-c-LIG.

Data C-f- 1 Initial condition for reach by parameter

Variable	*Column*	*Format*	*Description*
NAME	01-10	A4,6X	One to form letter identification of the parameter, in the following sequence: S – Salinity T – Temperature BOD - Biochemical Oxygen Demand ON - Organic Nitrogen N - Inorganic Nitrogen OP - Organic Phosphate P - Inorganic Phosphate CP – Phytoplankton CZ – Zooplankton DO - Dissolved Oxygen FCOL - Fecal Coliforms CLIG - Conservative Lignins DLIG - Decaying Lignins
NPTS	11-20	I10	Number of points defining the initial condition as given by the following data. If NPTS = 1, the value is applicable over the entire reach. NOTE: If NUTR is used, ON, N, OP, P, CP and CZ will be present as a group.

Data C-f- 2 Initial condition table

Variable	*Column*	*Format*	*Description*
X	01-10	F10.0	Distance from upstream node to location at which initial concentration is specified. (Can have any value for the case of NPTS = 1).
CON	11-20	F10.0	= initial concentration

D DATA GROUP

Data D-a Lateral Inflow Identification Data (one required)

Variable	*Column*	*Format*	*Description*
HEADER	01-80	20A4	Description of the lateral inflows and QFC structures

Data D-b Number of Lateral Inflows (one required)

Variable	*Column*	*Format*	*Description*
NLAT	01-10	I10	Total number of lateral inflows, including QFC structures, except those QFC structures that are fully described in the F Data Group. Add two for each QFCS because it describes inflow or outflow at both the upstream and down-stream sides.

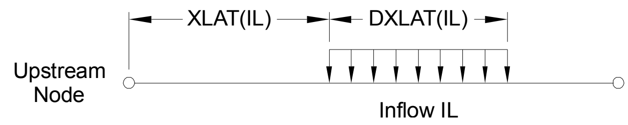
If there are no lateral inflows and QFC structures, then NLAT = 0, and this Data would represent the last Data of Data Group D. If lateral in-flows do exist in the model, then one of the following Data combinations will apply for each lateral inflow:

Lateral Inflow Type	D
User-defined lateral inflow in simple format	
User-defined lateral inflow in WSC format	
Upstream end of a QFC structure (except pumpstation)	
Upstream end of a pumpstation	
Downstream end of a QFC (including pumpstation)	

Data D-c Input Lateral Inflow Parameters (one required for each user-defined lateral inflow)

Variable	*Column*	*Format*	*Description*
IL	01-10	I10	Number of the lateral or QFC structure outflow or inflow to which this information applies
KLAT(IL)	11-20	I10	Number of the reach in which inflow IL is located
XLAT(IL)	21-30	F10.5	Distance from upstream end of reach from which inflow IL is being withdrawn from or input to. The distance must be at a node location or exactly at a mid-mesh point NOTE: DO NOT locate a lateral inflow within the length of one-half mesh spacing from either end of the reach. A lateral inflow located at either end of the reach will be ignored by the computational scheme, and those located within one-half mesh distance of the ends of the channel will be reduced by a value inversely proportional to the distance from mid-mesh point to the inflow location. NOTE: If the location of either end of a QFCS is at a node, then the QFCS number must be cross referenced in the hydraulic boundary conditions (Data F-b* and F-d)
DXLAT(IL)	31-40	F10.5	Width of inflow IL. (A point inflow can be

Variable	*Column*	*Format*	*Description*
			defined by a width of 0.0)



ILAT(IL)	41-50	I10	1, constant lateral inflow 2, variable lateral inflow
IT(IL)	51-55	I5	Number of table entries for inflow IL. One entry is necessary for constant lateral inflow, with more as needed to describe variable lateral inflow
NPAR	56-60	15	Number of water quality parameters used. Leave blank if water quality option is not used.
STIME	61-70	I10	Time in seconds, relative to model simulation starting time, of first data listed on first D-f Data. Leave blank if WSC format is not used.
TINC	71-80	I10	Time increment between data points in seconds for WSC format data. Leave blank if WSC format is not used.

Data D-c-1 Lateral inflow parameters: water quality parameter names

Variable	*Column*	*Format*	*Description*
SYM(L)	01-10	A4,6X	= the one to four letter identification of the water quality parameter in the given sequence and as described in Table A-2. The number of parameters should be equal to NPAR(IL). Only those parameters whose concentrations are non-zero need be specified. <u>NOTE:</u> Each data can accommodate up to 7 parameters, repeating the format specified above. Use more than one card if more than 7 parameters are required.

Data D-d Input Lateral Inflows (one required for every simple format discharge value)

Variable	*Column*	*Format*	*Description*
TIL(II)	01-10	F10.0	Time in seconds for table entry I, relative to the beginning of the period.
QLAT(II)	11-20	F10.0	Magnitude of the inflow in cfs/ft or cms/m for a distributed lateral inflow or cfs or cms for a

Variable	*Column*	*Format*	*Description*
			point inflow. One entry describes constant inflow and further entries describe a lateral inflow varying with time.
CLAT(IL,I,L)	21-30	F10.0	The specified concentration corresponding to the water quality parameter SYM(L) of Data D-c-2. Repeat for up to five concentration specifications (in columns 21 through 70) using this format. NOTE: Repeat this data for NPAR(IL) greater than 5, using the same format. NOTE: For dissolved oxygen (DO), the dissolved oxygen deficit (DOD) concentration should be entered in ppm, not the DO concentration.

Data D-f Water Survey of Canada Discharge Data Input (one required for each row of WSC data)

Variable	*Column*	*Format*	*Description*
ICHECK	01-01	I1	Verification number identifying the type of data being read, ICHECK = 1 or 5 means water discharge data in cfs or cms, respectively.
STA	02-16	15X	Station number identification and date of information as provided in WSC format (not used by the program but helpful for data management)
QLAT(I+i)	17-76	6F10.0	Individual time period discharge values at time TIL(I+i) = STIME + (i-1) x TINC (in seconds). NOTE: i starts at 1 to a maximum value of IT(IL).

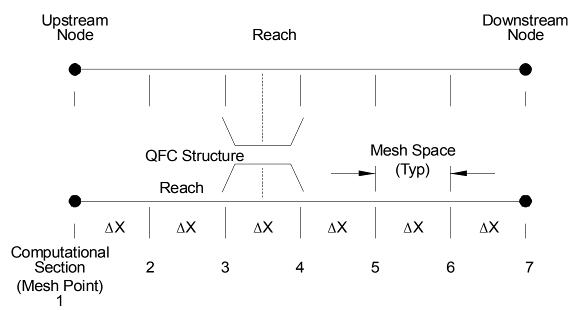
Data D-c* QFC Computed Lateral Inflows (one required for each end of a QFCS)

Variable	*Column*	*Format*	*Description*
IL	01-10	I10	Number of the lateral or QFC inflow or outflow to which this information applies
KLAT(IL)	11-20	I10	Number of the reach in which inflow IL is located
XLAT(IL)	21-30	F10.5	Distance from the upstream end of the reach where inflow IL is being withdrawn from or input to. NOTE: A QFC structure can be located at the extremities of a reach such as at distance zero or the end; but it will not be considered in the computational procedure as a lateral inflow. This option is used when a structure is needed to be described as a lateral at its downstream side, but the upstream side is located at a node. The node description is given in Data F-b* and parameter NOBC(KN) = 11.
DXLAT	31-40	F10.5	Width of inflow IL. (A point inflow can be

Variable	*Column*	*Format*	*Description*
			defined by a width of 0.0)
ILAT(IL)	41-50	I10	3, If QFC structure is a dyke, culvert or floodbox (aboiteau) 4, If QFC structure is a pumpstation
IT(IL)	51-55	I5	Number of table entries for inflow IL. For a QFCS set to 1.

Data D-d* QFC Structure Description (one required for each end of a QFCS)

Variable	*Column*	*Format*	*Description*
TIL(II)	01-10	F10.0	For a QFC structure, this is the sequential numerical identification. Start the first structure with 1. (One pumpstation would count as one QFC structure and two lateral inflows.)
QLAT(II)	11-20	F10.0	A QFC structure upstream or downstream definition 0, upstream side of the structure, and thus requires Data D-c* through D-g* 1, downstream side of the structure, and thus requires D-c* and D-d* only.



A QFC structure, considered as a lateral flow from one channel to another, must be located at a mid-mesh distance within each respective channel as shown above (or one-half computed Δx).

Data D-e* QFC Structure Input (Except Pump station)- Upstream Side (one required for each non-pump station QFCS)

Variable	*Column*	*Format*	*Description*
ISTYLE	01-06	I6	1, culvert with a dyke 2, floodbox (aboiteau) with dyke 3, dyke only 4, culvert without dyke 5, floodbox (aboiteau) without dyke Default = 1
ISHAPE	07-10	I4	11, concrete rectangular 21, concrete circular conduit

Variable	*Column*	*Format*	*Description*
			22, corrugated circular pipe
			Default = 11 when ISTYLE = 2 or 5, and 22 when ISTYLE = 1 or 4)
D	11-20	F10.3	Diameter for circular pipe or width for rectangular conduit (in feet or metres)
QCINT	21-30	F10.3	Initial discharge through conduit
CONDL	31-40	F10.3	Length of conduit (Default = 100 units, either feet or metres, depending which system of units was selected).
RDIVD	41-50	F10.5	Entrance parameter of conduit. (Bevel or Rounding / D)
			NOTE: Default is 0.06 when ISHAPE = 11 or 0.04 when ISHAPE = 22) If RDIVD < 0.2, long version (CNDT2) is invoked. If RDIVD > 0.2, short version (CNDT3) is invoked meaning that RDIVD changes to CALBR. Normally one assigns a value of 1.0 to CALBR. If, however, one finds that the computed discharge by CNDT3 is too high or too low, one can change the value of CALBR from 1.0 to say 0.9 or 1.1; in other words, it is a quick way to calibrate the computed discharges from CNDT3
	51-60		BLANK
RINVUP	61-70	F10.3	The elevation of the upstream end of the conduit's invert
CRNUP	71-80	F10.3	The elevation of the upstream end of the conduit's crown

Data D-f* QFC Structure Input - Upstream Side (one required for each non-pump station QFCS)

Variable	*Column*	*Format*	*Description*
ZDELTA	01-10	F10.3	The difference in elevation between the upstream invert and the downstream invert. It has a positive value if the upstream invert is higher than the downstream invert
RNMAN	11-20	F10.6	Manning's 'n' (not used in CNDT3, see RDIVD parameter in Data D-e*)
			Default RNMAN = 0.012 for concrete structures (ISHAPE=11 or 21) and RNMAN = 0.024 for corrugated culverts or floodboxes (ISHAPE = 22)
AINT	21-30	F10.3	Initial cross-sectional area of flow. Not used in CNDT2 or CNDT3.
			Default = D/2
DYKEH	31-40	F10.3	Elevation of the crest of the dyke (not used if ISTYLE = 4 or 5)
DYKEL	41-50	F10.3	Length of dyke over which flow may occur (not used if ISTYLE = 4 or 5)

Variable	*Column*	*Format*	*Description*
QDINT	51-60	F10.3	Initial discharge over dyke (not used if ISTYLE = 4 or 5)
DQ	61-70	F10.3	Maximum allowable change in discharge during one hour, in cfs/hr or cms/hr NOTE: This parameter was intended to allow the model to stabilize when a large structure is employed and has a capacity greater than the connecting channels on either the upstream or downstream sides of the QFC structure. Use with caution, if at all, as this can result in unnecessarily large flow restrictions.
BARREL	71-80	F10.0	Number of culvert barrels with the same diameter and invert elevation.

Data D-g* QFC Structure Input - Upstream Side (one required for each non-pump station QFCS)

Variable	*Column*	*Format*	*Description*
CONSTY 1,ISTRUC	01-10	F10.3	Constant for type I flow Default = 0.98
CONSTY 2,ISTRUC	11-20	F10.3	Constant for type II flow Default = 0.95
CONSTY 3,ISTRUC	21-30	F10.3	Constant for type III flow Default = 0.80
CONSTY 4,ISTRUC	31-40	F10.3	Constant for type IV flow Default = 0.56
CONSTY 5,ISTRUC	41-50	F10.3	Constant for type V flow Default = 0.55
CONSTY 6,ISTRUC	51-60	F10.3	Constant for type VI flow Default = 0.6

NOTE: When RDIVD is less than 0.2, the long version (CNDT2) is automatically invoked. In this case, the program calculates its own constants for each type of flow. For this reason, the user will leave this Data blank.

When RDIVD is greater than 0.2 the short version (CNDT3) is invoked. In this case the user must define the values for the constants or simply use the default values.

Data D-e** Pump station Parameters (one Data required for each pump station)

Variable	*Column*	*Format*	*Description*
NPUMP(I)	01-10	I10	Number of pumps at this station
ISTORM(I)	11-20	I10	Pump switch range at beginning of run, ISTORM(I)=0 (normal mode) and ISTORM(I)=1 (storm mode)
HTOL(I)	21-30	F10.3	Allowable difference between the trial head and the calculated head loss through each pump (Default; HTOL(I)=0.01). Used to end iterative routine.
KREMOT(I)	31-40	I10	Reach number where remote switches are located (required only if remote switches exist)
XREMOT(I)	41-50	F10.3	Distance along reach where remote level switches are located (required only if remote switches exist)

Data D-f** Pump station Emergency and Remote Switch Settings (one Data required for each pump station)

Variable	*Column*	*Format*	*Description*
PSELEV(1,I)	01-10	F10.3	Elevation of emergency high level "off" switch setting (higher of two settings)
PSELEV(2,I)	11-20	F10.3	Elevation of emergency high level "on" switch setting (lower of two settings)
PSELEV(3,I)	21-30	F10.3	Elevation of remote site "off" switch setting (lower of two settings), pump control reverts to normal mode when water falls below this level
PSELEV(4,I)	31-40	F10.3	Elevation of remote site "on" switch setting (higher of two settings), pumps control switches to storm mode when water rises above this level

Data D-g** Pump Switch Settings (one Data required for each pump)

Data D-g, D-h, D-i and D-j form a unit which must be repeated for each pump)

Variable	*Column*	*Format*	*Description*
PMELEV (1,J,K)	01-10	F10.3	Elevation of local intake "off" switch setting (lower of two settings)
PMELEV (2,J,K)	11-20	F10.3	Elevation of local intake "on" switch setting (higher of two settings)
PMELEV (3,J,K)	21-30	F10.3	Elevation of storm mode local intake "off" switch setting (lower of two settings)
PMELEV (4,J,K)	31-40	F10.3	Elevation of storm mode local intake "on" switch setting (higher of two settings)

Data D-h** Pump Parameters (one Data required for each pump)

Variable	*Column*	*Format*	*Description*
UNIT1	01-10	I10	Variable which allows selection of units for pump head when entering pump curve data. (Not implemented at this stage. Enter a zero. Data must be entered in feet.)
UNIT2	11-20	I10	Variable which allows selection of units for pump discharge when entering pump curve data. (Not implemented at this stage. Enter a zero. Data must be entered in USgpm)
NUM	21-30	I10	Number of table entries of pump head and discharge (pump curve data)
MULT	31-40		Multiplier to simulate a number of identical pumps, all using identical switch settings
ICASE(I,K)	41-50		Initial pump status indicator, ICASE(I,K)=2 (default) where pump is off due to low water level, ICASE(I,K)=1 indicates pump is off due to high water level and ICASE(I,K)=0 indicates pump is on
ELOUTP(I,K)	51-60	F10.3	Elevation of the outlet end of the discharge pipe, if horizontal this would be the centreline elevation

Data D-i** Pump Intake and Discharge Pipe Parameters (one Data required for each pump)

Variable	*Column*	*Format*	*Description*
DIINTP(I,K)	01-10	F10.3	Diameter of the intake pipe to the pump (Default; DIINTP(I,K)=10)
DIOUTP(I,K)	11-20	F10.3	Diameter of the outlet pipe from the pump (Default; DIOUTP(I,K)=10)
XLINTP(I,K)	21-30	F10.3	Length of the intake pipe to the pump upstream of the factory pump inlet
XLOUTP(I,K)	31-40	F10.3	Length of the discharge pipe to the pump downstream from the factory pump outlet
XEINTP(I,K)	41-50	F10.3	Equivalent length of pipe to account for all fittings, bends and other losses (excluding friction) in the intake pipe upstream of the factory pump inlet
XEOUTP(I,K)	51-60	F10.3	Equivalent length of pipe to account for all fittings, bends and other losses (excluding friction) in the discharge pipe downstream from the factory pump outlet
FINTP(I,K)	61-70	F10.3	Manning's 'n' value for the intake pipe to the pump
FOUTP(I,K)	71-80	F10.3	Manning's 'n' value for discharge pipe from the pump

Data D-j** Pump Curve Data (one Data required for each head-discharge pair)

Variable	*Column*	*Format*	*Description*
PDATA(I,J)	01-10	F10.3	Total pumping head (this version of program will only accept head in feet)
PDATA(2,J)	11-20	F10.3	Discharge (this version of program will only accept discharge in USgpm) **Note:** Discharge data must decrease with increasing head. The pump curve data may be ordered from highest to lowest head, or lowest to highest head, but the series of data pairs must be in order.

E DATA GROUP

This data group is omitted if water quality calculations are not to be executed, JOPT(1) = 2.

Data E-a Injection identification data

Variable	*Column*	*Format*	*Description*
Header	01-80	20A4	Descriptions of injections NOTE: After the identification data, E-a, there is a package of data which is repeated for all injection data.

Data E-b Number of injection points

Variable	*Column*	*Format*	*Description*
NJECT	01-10	I10	Total number of injection locations NOTE: If there are no injection points, NJECT = 0, the computer will skip to the next data group. Otherwise it will expect the injection point data from Data E-c and E-d.

Data E-c-1 Injection parameters

Variable	*Column*	*Format*	*Description*
IL	01-10	I10	Number of the injection point to which the information applies
KJECT(IL)	11-20	I10	Number of the reach in which injection IL is located
XJECT(IL)	21-30	F10.0	Distance from the upstream end of reach to the injection point
IJECT(IL)	31-40	I10	1, constant injection rate 2, variable injection rate
ITJ(IL)	41-50	I10	Number of table entries for injection IL. One entry for constant injection rate with more as needed, to describe variable injection rates.

NPAR(IL)	51-60	I10	Number of water quality parameters being injected NOTE: Data E-c and E-d constitute a package and must be repeated according to the number of injection points specified, NJECT.

Data E-c-2 Injection parameters: water quality names

Variable	*Column*	*Format*	*Description*
SYM(L)	01-10	A4,6X	The one to four letter identification of the water quality parameter in the given sequence and as described in Table A-2. ONLY THOSE PARAMETERS BEING INJECTED NEED BE SPECIFIED. NOTE: Each data can accommodate up to 7 parameters, repeating the format specified above. Use more than one data line if more than 7 parameters are required.

Data E-d Injection data

Variable	*Column*	*Format*	*Description*
TJIL(IL,I)	01-10	F10.0	Time (in seconds) for table entry I, relative to the beginning of the period.
PJECT(IL,I,L)	11-70	F10.0	Units: Phytoplankton - pounds of dry weight per day Zooplankton - pounds of dry weight per day Temperature - BTU per day Coliforms - number per hour All others - pounds per day NOTE: 1 BTU = 1° F/ lb_m water

F DATA GROUP

For additional information please refer to "Boundary Conditions" in Section 3.1.

Data F-a Identity Data for Hydraulic Boundary Conditions (one required)

Variable	*Column*	*Format*	*Description*
HEADER	01-80	20A4	Hydraulic description of the nodes

After the identification Data, F-a, a variety of possible data combinations might follow, depending on the type of boundary condition. The following table lists all the possibilities:

Boundary Condition Type	NOBC(KN)	Required Type and Sequence of Data
User-defined constant or sinusoidal water level	1	one F-b and one F-c
User-defined constant discharge	2	one F-b and one F-c
User-defined variable water level in simple format	1	one F-b and more than one F-c
User-defined variable water level in WSC format	1	one F-b and one or more F-e
User-defined variable discharge in simple format	2	one F-b and more than one F-c
User-defined variable discharge in WSC format	2	one F-b and one or more F-f
Stage-routing	3	one F-b and one F-c
Stage-discharge rating curve	4	one F-b and more than one Fc; for shift option, end with F-c*
Upstream end of non-QFC control structure	3 or 4	as for stage-routing or rating curve (listed above), whichever applies
Downstream end of non-QFC control structure	5	one F-b and one F-d
Upstream end of QFC, except pumpstation	9(a) or 11(b)	one F-b* and D-e* through D-g* if NOBC (KN) = 9 (see D-Data Group), or one F-b* only if NOBC(KN) = 11
Downstream end of QFC, except pumpstation	10	one F-b* only
Upstream end of pumpstation	12(a) or 14(b)	one F-b* and D-e** through D-j** if NOBC(KN) = 12 (see D-Data Group) or one F-b* only if NOBC(KN) = 14
Downstream end of pumpstation	13	one F-b* only

(a) use this if the structure is to be described in the F-Data Group (node to node QFC)
(b) use this if the structure is to be defined in the D-Data Group (node to lateral QFC or lateral to node QFC)

Data F-b Node Parameters (one required) (For QFC structure, see Data F-b*)

Variable	*Column*	*Format*	*Description*
KN	01-10	I10	Number of the node for the following information
NOBC(KN)	11-20	I10	Indicates the type of boundary condition to be applied at node KN.

0, junction or interior node
1, water surface elevation prescribed
2, discharge prescribed
3, stage-routing boundary condition
4, rating curve (z vs Q - table)
5, downstream end of control structure

- WEIRS - Peace-Athabasca model option

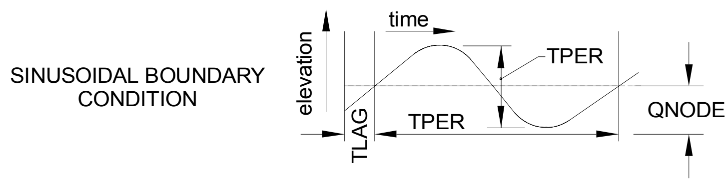
Variable	*Column*	*Format*	*Description*
			6, Rivière des Rochers weir 7, Revillon Coupé weir 8, Rivière des Rochers natural weir NOTE: for NOBC(KN) = 0, 3 and 5 no additional information on this Data is required.
IBC(KN)	21-30	I10	Indicates the time dependence of the boundary condition at node KN. Is used when NOBC(KN) = 1, 2 and 4. 1, constant with time 2, variable with time 3, sinusoidal with time (see Data F-c)
ITX(KN)	31-40	I10	Number of discharge values for the boundary condition specifications. For constant boundary conditions, and downstream side of control structure, only one value is required. Sinusoidal boundary conditions are handled as one value also,. More discharge values are required when IBC(KN)=2
INT(KN)	41-50	I10	Type of interpolation between consecutive discharge values that are variable with time (only used if IBC(KN) = 2) 1, linear interpolation of variable boundary condition data. 2, cosine interpolation of variable boundary condition data. 3, parabolic interpolation using a three degree parabola
STIME	51-60	I10	Time, in seconds, relative to model simulation starting time, of first data listed on first F-e or F-f Data. This Data is required only if data is in WSC format: add Data F-e for NOBC(KN) = 1 and Data F-f for NOBC(KN) = 2. STIME can be positive, negative or zero.
TINC	61-70	I10	Time increment between data points (in seconds) for WSC format data. Leave blank if WSC format is not used.

Data F-c Boundary Node Conditions

Variable	*Column*	*Format*	*Description*
TIME	01-10	F10.5	Elapsed time in seconds from the beginning of each individual period. This can be left blank if the boundary condition is constant or sinusoidal.
ZNODE	11-20	F10.5	Water surface elevation. If constant, the water surface elevation is assigned the value for J=1. If the time dependence is sinusoidal, the mean value about which the surface elevation oscillates is assigned the value for J=1. (J is the subscript of the time increment)
QNODE	21-30	F10.5	Discharge at node KN at time TI(KN,J). If the time dependence is constant, the discharge is assigned the value for J=1. If the time

Variable	*Column*	*Format*	*Description*
			dependence is sinusoidal, the mean discharge about which the discharge oscillates is assigned the value for J=1
TPER(KN)	31-40	F10.5	Period of oscillation for the sinusoidal boundary condition at node KN
PEAK(KN)	41-50	F10.5	Amplitude of oscillation for the sinusoidal boundary condition at node KN
TLAG(KN)	51-60	F10.5	Time lag for the sinusoidal boundary condition

Data F-c allows the user to specify a sinusoidal boundary condition as shown in the sketch below.



Data F-d Boundary node condition

Variable	*Column*	*Format*	*Description*
NCTUP(KN)	01-10	I10	The number of the node on the upstream side of the control structure. The discharge of the downstream node is taken from that of the upstream node. There are four cases: 1) the upstream node has discharge specified. In this case, the program takes this value directly. 2) the upstream node is a rating curve type boundary. In this case, the program sets the discharge of the downstream node equal to that determined by the program for the upstream node at the end of the previous time step. 3) the upstream node is a stage-routing type boundary. The discharge is handled as in Case 2. 4) the upstream node has the water level specified. The discharge is handled as in Case 2

Data F-e Water Survey of Canada elevation data input (one required for each set of 8 water levels)

Variable	*Column*	*Format*	*Description*
ICHECK	01-01	I1	Verification number identifying the type of data being read, ICHECK = 4 or 8 means water surface elevation data is given in feet or metres, respectively
STA	02-16	15X	Station number identification and date of information as provided in WSC format (not used by program but helpful for data management)

ZQ(I+i)	17-80	8F8.3	Water surface elevation values at time TZ(I+i) = STIME + (i-1) x TINC (in seconds) NOTE: i starts at 1 to a maximum value of ITX(KN)

Data F-f Water Survey of Canada discharge data input (one required for each set of 6 discharges)

Variable	*Column*	*Format*	*Description*
ICHECK	01-01	I1	Verification number identifying the type of data being read, ICHECK = 1 or 5 means water discharge data is given in cfs or cms, respectively
STA	02-16	15X	Station number identification and date of information as provided in WSC format (not used by program but helpful for data management)
ZQ(I+i)	17-76	6F10.0	Discharge values at time TZ(I+i) = STIME + (i-1) x TINC (in seconds) NOTE: i starts at 1 to a maximum value of ITX(KN).

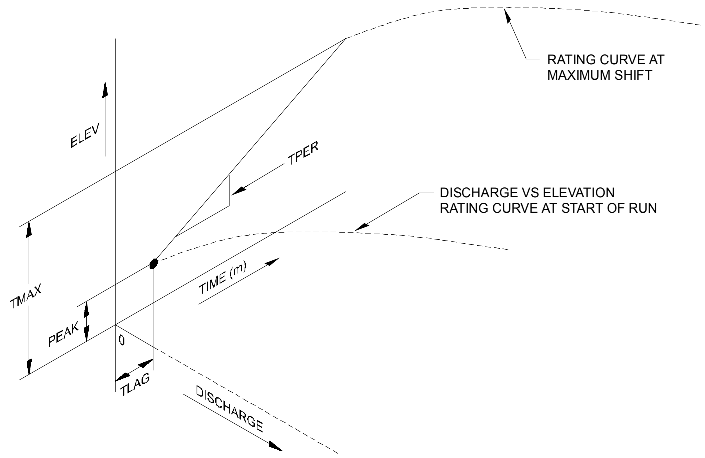
Data F-b* QFC Structure at a Node

Variable	*Column*	*Format*	*Description*
KN	01-10	I10	Number of the node for the following information
NOBC(KN)	11-20	I10	Indicated the type of condition to be applied at node KN 9, upstream end of the structure (add Data D-e* through D-g* to describe the QFC structure) 10, downstream end of the structure (no additional Data are required). The value of 10 may only be used when the lateral inflow is being withdrawn from an upstream node. 11, upstream end of the structure described in lateral inflow section (no additional Data are required) If the pumping option is selected 12, upstream end of a pumpstation to be described in the F Data Group (add Data D-e** through D-j** to describe the pumpstation) 13, downstream end of the pumpstation (no additional Data are required) 14, upstream end of the pumpstation described in lateral inflow section (no additional Data are required)

Variable	*Column*	*Format*	*Description*
IBC(KN)	21-30	I10	Node number of connecting downstream node; given only for NOBC(KN) = 9 or 11, 12 or 14. Upstream node for NOBC(KN) = 10 or 13. Leave blank if QFCS connects to a mid-mesh point at the other end.
ITX(KN)	31-40	I10	1 for QFCS (including pumpstations)
INT(KN)	41-50	I10	QFCS identification number. This is a consecutive sequence number following those specified in the lateral inflow section (variable T1L(II) on Data D-d*) NOTE: pumpstations are treated as any other QFCS and included in this numbering scheme.

Data F-c*   Last Data of Rating Curve for Shift Option (optional Data)

Variable	*Column*	*Format*	*Description*
TIME	01-10	F10.5	BLANK
ZNODE	11-20	F10.5	Water surface elevation. If constant, the water surface elevation is assigned the value for J=1. If the time dependence is sinusoidal, the mean value about which the surface elevation oscillates, is assigned the value for J=1. (J is the subscript of the time increment)
QNODE	21-30	F10.5	Discharge at node KN at time TI(KN,J). If the time dependence is constant, the discharge is assigned the value for J=1. If the time dependence is sinusoidal, the mean discharge, about which the discharge oscillates, is assigned the value for J=1
TPER(KN)	31-40	F10.5	Shift in elevation in feet or metres per time step
PEAK(KN)	41-50	F10.5	Shift in elevation prior to start of simulation run
TLAG(KN)	51-60	F10.5	Time shift lag in number of time steps to when first computational shift per time step must begin
TMAX(KN)	61-70	F10.5	Maximum desired shift in elevation of the stage-discharge table



Data F-c* allows the user to specify a shift to the stage-discharge boundary condition as shown in the sketch above.

G DATA GROUP

Data G-a Identification Data (one required)

Variable	*Column*	*Format*	*Description*
HEADER	01-80	20A4	Description of hydraulic output parameters

Data G-b Number of Hydrographs (one required)

Variable	*Column*	*Format*	*Description*
NYHD	01-10	I10	Number of hydrographs requested to be printed in TAPE61.TXT.

Data G-b and G-c constitute a package. However, if the user sets NHYD = 0, no hydrographs are to be printed and it is not necessary to include Data G-c.

Data G-c Hydrograph Parameters (one required for each requested hydrograph)

Variable	*Column*	*Format*	*Description*
KHYD(IH)	01-10	I10	Reach in which hydrograph IH is to be produced
XHYD(IH)	11-20	F10.5	Desired location in the reach for the hydrograph. The program will find the nearest computational mesh point to this location and print the hydrograph for that location. (No interpolation.)
IHPER	21-30	I10	Period for which the desired hydrograph is to be produced. In a cyclic-stable solution, all

			hydrographs must be produced in the same period. In a transient solution, this value should be the number of the current period. The retrieval system is awkward and comprehensive output is best obtained by storing the solution in TAPE10.TXT. Output can then be produced graphically using the post-processors.
LHYD(IH)	31-40	I10	1, reach in which hydrograph IH is to be produced as a separate file
MHYD(IH)	41-50	I10	1, reach in which hydrograph IH is to be produced as a separate file

Data G-d Number of Hydraulic Profiles (one required)

Variable	*Column*	*Format*	*Description*
NPRO	01-10	I10	Number of hydraulic profiles requested to be printed in TAPE61.TXT. Data G-d and G-e constitute a package. However, if the user sets NPRO = 0, no profiles are to be printed and it is not necessary to include Data G-e.

Data G-e Profile Parameters (one required for each requested profile)

Variable	*Column*	*Format*	*Description*
KPRO(IPHY)	01-10	I10	Reach in which profile IPHY is to be printed
INCHY(IPHY)	11-20	I10	Time increment at which profile IPHY is to be printed. Should be less than or equal to NINC. Refer to Data A-e for total number of increments, NINC. NOTE: The profiles must be specified consecutively with time, because the computed output data is stored by time step. For example, a profile request for any reach at time step 10 must follow any profile request for time step 9. Within each time step, the reaches must be requested in the same order that they appear in the reach-node connectivity table.
IPPER(IPHY)	21-30	I10	Period in which profile IPHY is to be printed, refer to Data G-c variable IHPER for additional advice.
JPRO(IPHY)	31-40	I10	1, reach in which profile IPHY is to be produced as a separate file
LPRO(IPHY)	41-50	I10	1, reach in which profile IPHY is to be produced as a separate file

H DATA GROUP

This data group is omitted if water quality calculations are not to be executed, JOPT(1) = 2.

Data H-a Identity data for water quality boundary conditions

Variable	*Column*	*Format*	*Description*
Header	01-80	20A4	Water quality boundary conditions

Data H-b Water quality node parameters

Variable	*Column*	*Format*	*Description*
KN	01-10	I10	Number of the node for the following information
NOBCM(KN)	11-20	I10	Indicates the type of boundary condition to be applied to node KN 0, junction or interior node 1, concentration specified 2, dispersive flux specified 3, total flux specified 4, ocean boundary condition 5, control structure node (up or downstream)
IBCM(KN)	21-30	I10	Indicates the time dependence of the boundary condition at node KN 1, constant with time 2, variable with time
ITXM(KN)	31-40	I10	Number of table entries per parameter modelled for the boundary condition specified. For constant boundary conditions, or the ocean boundary condition, only one data per parameter modelled is required. Variable boundary conditions will require additional table entries. NOTE: For interior nodes, or control structure nodes, no other data are required and the computer skips to the next Node Parameter Data. However, for boundary nodes there are three classes, one for constant or variable boundary conditions (Data H-c), a second for ocean boundaries (Data H-d and H-e) and a third for a control structure boundary (Data H-f). One of these sets of data must be used. NOTE: Options NOBCM(KN) = 3 and 4, total flux specified and ocean boundary conditions, are presently not functioning properly and should not be used.

Data H-c Constant or variable boundary conditions

Variable	*Column*	*Format*	*Description*
SYM(L)	01-10	A4,6X	The symbolic (one to four letters) name of the water quality parameter being specified. Use sequence of Table A-2.

TIM(KN,J)	11-20	F10.5	Prototype time referred to the beginning of the period for the table entry. This may be omitted if the boundary condition is constant, IBCM(KN) = 1.
CNODE	21-30	F10.5	Specified concentration of the water quality parameter
DFNODE	31-40	F10.5	Specified dispersive flux of the water quality parameter
TFNODE	41-50	F10.5	Specified total flux of the water quality parameter Units: Phytoplankton - pounds of dry weight per day Zooplankton - pounds of dry weight per day Temperature - BTU per day Coliforms - number per hour All others - pounds per day NOTE: In case of dispersive and total flux, quantity for parameter dissolved oxygen should be in terms of dissolved oxygen demand. NOTE: Data H-c must be repeated for each quality constituent specified on Data A-d, the Water Quality Parameter Options. For variable boundary conditions and for each quality constituent, a package of data corresponding to the time varying input must be supplied. If Data H-c is supplied then Data H-d, H-e and H-f are not to be supplied.

Data H-d Time constant data for ocean boundary conditions

Variable	*Column*	*Format*	*Description*
TCON(KN)	01-10	F10.5	Time constant for decay of the concentration difference CO(KN) - CS(KN) at the ocean boundary node, where CO(KN) is the concentration leaving the estuary on ebb flow and CS(KN) is specified on the next data. The boundary concentration is specified by: $CONC(KN)=CS(KN)+(CO(KN)-CS(KN))e^{(-TCON(KN) \times t)}$ where: t = (time - time that flood began) $TCON(KN) \times t < 88$ $CONC(KN) = CS(KN)$ when $TCON(KN) \times t \geq 88$ NOTE: If an ocean boundary, NOBCM(KN) = 4, is specified then 2 or more data are required: the time constant data, H-d, and quality constituent data H-e for each constituent modelled.

Data H-e Water quality constituent data for ocean boundary conditions

Variable	*Column*	*Format*	*Description*
SYM(L)	01-10	A4,6X	The symbolic (one to four letter) name of the water quality parameter being specified. Use sequence of Table A-2.

CS(KN,L)	11-20	E10.2	Concentration of the water quality parameter of the incoming ocean water on flood at the ocean boundary node. NOTE: If an ocean boundary is specified, NOBCM(KN) = 4, then this data must be repeated for each quality constituent specified on data A-2, the Water Quality Parameter Options Data.

Data H-f Water quality graphs and profile output - i**dentity data**

Variable	*Column*	*Format*	*Description*
Header	01-80	20A4	Water quality graphs and profile output parameters

Data H-g Number of quality graphs

Variable	*Column*	*Format*	*Description*
NPOL	01-10	I10	= number of quality graphs requested NOTE: Data I-b and I-c constitute a package; however, if the user does not wish to see the hydrographs it is not necessary to include data I-b and I-c.

Data H-h Water quality graph parameters

Variable	*Column*	*Format*	*Description*
KPOL(IC)	01-10	I10	Reach in which quality graph IC is to be produced
XPOL(IC)	11-20	F10.5	Desired location (in feet) in the reach for the quality graphs. The program will find the nearest computational mesh point to this location, and produce the quality graph there.
MCPER	21-30	I10	Period over which the desired quality graph is to be produced. The remarks made in Data Group G about hydrographs apply here also. At a given location over several periods, the data group must be split into individual quality graphs covering single periods.

Data H-i Number of water quality profiles

Variable	*Column*	*Format*	*Description*
NMPRO	01-10	I10	Number of concentration profiles requested NOTE: Data I-d and I-e constitute a package, however, if the user does not wish to see the quality profiles it is not necessary to include data I-d and I-e.

Data H-j Water quality profile parameters

Variable	Column	Format	Description
MPRO(IPWQ)	01-10	I10	Reach in which profile IPWQ is to be printed
INCWQ(IPWQ)	11-20	I10	Water quality time increment at which profile IPWQ is to be printed
MPPER(IPWQ)	21-30	I10	Water quality period in which profile IPWQ is to be printed NOTE: The profile must be specified consecutively with time. NOTE: Data I-e must be repeated for the number of quality profiles requested, NMPRO.

I DATA GROUP

Data I-a Ice evolution type

Variable	Column	Format	Description
Header(j), j=1,20	1, 80	20A4	Description of run

Data I-b Ice deposit option

Variable	Column	Format	Description
DEPOPT(IEVOL) (If DEPOPT is set at "99", then the default value is automatically adopted: DEPOPT=1)	1 to 5	I5	Ice deposition method: 1 – User defined limiting velocity for deposition 2 – Meyer Peter bed load analogy 3 - Densimetric Froude number
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable (does not appear in output)

Data I-c Ice deposit velocity for DEPOPT=1

Variable	Column	Format	Description
VDEP(IEVOL) (If VDEP is set at "99.", then the default value is automatically adopted: VDEP= 1.2m/s)	1 to 5	F5.0	Maximum velocity for ice deposition (m/s). This is used if DEPOPT=1. If DEPOPT>1, this variable is not used, *but must still be provided (a value of zero is allowable)*
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-d Ice particle diameter

Variable	*Column*	*Format*	*Description*
DIAICE(IEVOL) (If DIAICE is set at "99.", then the default value is automatically adopted: DIAICE=0.15m)	1 to 5	F5.0	If DEPOPT=2, this parameter is used to represent the average diameter of the floating ice pans (in m). *Note: if DEPOPT is not=2, this parameter must still be defined (although not used in the calcs).*
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-e Densimetric Froude Number

Variable	*Column*	*Format*	*Description*
FRMAX(IEVOL) (If FRMAX is set at "99", then the default value is automatically adopted: FRMAX=0.2)	1 to 5	F5.0	Densimetric Froude Number, required if DEPOPT=3.
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-f Ice erosion option

Variable	*Column*	*Format*	*Description*
EROPT(IEVOL) (If EROPT is set at "99", then the default value is automatically adopted: EROPT=1)	1 to 5	I5	Ice erosion method: 1 – User defined minimum velocity at which erosion of ice cover commences 2 – User defined minimum tractive force at which erosion commences
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-g Ice deposit option

Variable	*Column*	*Format*	*Description*
VERODE(IEVOL) (If VDERODE is set at "99.", then the default value is automatically adopted: VDEP=1.8m/s)	1 to 5	F5.0	If EROPT=1, this value will trigger erosion of the ice cover to maintain a velocity of "VERODE" (m/s). *This value must still be provided, even if EROPT is not 1*.
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-h Tractive force for ice cover erosion

Variable	*Column*	*Format*	*Description*
FTRLIM(IEVOL) (If FTRLIM is set at "99.", then the default value is automatically adopted: FTRLIM=0.1)	1 to 5	F5.0	If EROPT=2, this value will trigger ice cover erosion to maintain a maximum tractive force "FTRLIM" (units??). *This value must be provided even if EROPT is* not 2.
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-i Option number for evaluation of leading edge stability

Variable	*Column*	*Format*	*Description*
LEOPT(IEVOL) (If LEOPT is set at "99", then the default value is automatically adopted: LEOPT=1)	1 to 5	I5	Leading edge stability method (see explanations in Section 4_): 1 – Pariset- Hausser 2 – Ashton 3 - User defined leading edge thickness
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-j Ice front thickness for LEOPT=3

Variable	*Column*	*Format*	*Description*
FRONTTHICK (If FRONTTHICK is set at "99.", then the default value is automatically adopted: FRONTTHICK=0.3m)	1 to 5	F5.0	If LEOPT=3, this value will control the ice front thickness (m). *This value must be provided even if LEOPT is not 3.*
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-k Ice transport speed factor

Variable	*Column*	*Format*	*Description*
VFACTR (IEVOL) (If VFACTR is set at "99.", then the default value is automatically adopted: VFACTR=1.0)	1 to 5	F5.0	Transport of ice under a stationary ice cover moves at a velocity of "VFACTR" times the water velocity.

Variable	*Column*	*Format*	*Description*
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-l Ice cover porosity

Variable	*Column*	*Format*	*Description*
POROSC (IEVOL) (If POROSC is set at "99.", then the default value is automati-cally adopted: POROSC= 0.7)	1 to 5	F5.0	Porosity of the ice cover (user defined value, usually between 0.4 and 0.9)
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-m Slush ice porosity

Variable	*Column*	*Format*	*Description*
POROSFS (IEVOL) (If POROSFS is set at "99.", then the default value is automatically adopted: POROSFS=0.5)	1 to 5	F5.0	Porosity of the frazil and slush ice pans approaching the leading edge of the ice cover (user defined value, usually between 0.3 and 0.7)
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-n Slush ice pan thickness

Variable	*Column*	*Format*	*Description*
SLUSHT (If SLUSHT is set at "99.", then the default value is automatically adopted: SLUSHT=0.15m)	1 to 5	F5.0	User defined thickness of slush ice pans being carried in the open water zone (usually between 0.1 and 0.3 m)
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-o Cohesion of ice cover to riverbanks

Variable	*Column*	*Format*	*Description*
COHESN (If COHESN is set at "99.", then the	1 to 5	F5.0	Cohesion of ice cover to riverbanks (Pa)

default value is automatically adopted: COHESN=0.)			
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-p Number of user invoked ice bridges

Variable	*Column*	*Format*	*Description*
ICEBRGSW	1 to 5	I5	Number of "switches" or "lodgments" to be used in initiating ice cover(s)
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-q Ice bridge information

Variable	*Column*	*Format*	*Description*
RLOCBRG	1 to 5	I5	Cross section number at which ice bridge will form
DAYSBR	6 to 15	F10.0	Number of days at which ice bridge forms (after start of simulation)
BRIDTH	16 to 25	F10.0	Thickness of ice bridge that forms (m)
THERMD	26 to 35	F10.0	Thermal ice cover assumed at ALL cross sections downstream of section "RLOCBRG". If THERMD>0, a thermal ice cover will be imposed with a thickness equal to THERMD. If THERMD=0, no thermal ice assumed.
Header(j), j=1,10	36 to 75	10A4	"Reminder" description of variable

Data I-r Number of user load-shedding factors

Variable	*Column*	*Format*	*Description*
NSHEDF	1 to 5	I5	Number of "switches" or "load-shedding" factors to be used in varying bank, meander and bridge pier resistance to the ice cover(s)
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable
			Note: When NSHEDF is set to "0" then Data I-r-a is not required.

Data I-r-a Load-shedding factor information

Variable	*Column*	*Format*	*Description*
LOCSHED	1 to 5	I5	Cross section number at which ice cover load-shedding factor is applied to (default = 1.0)
VALUE	6 to 15	F10.0	Value of changed load-shedding factor (value starts and continues to last section until reset back to 1.0)

Variable	Column	Format	Description
			Note: single channel **load-shedding factor** is assumed to be 1.0 while an island with 2 **channels** is assumed to be 2.0.

Data I-s Simulation stop instruction

Variable	Column	Format	Description
ISTOP	1 to 5	I5	When ice cover advances to cross section "ISTOP", the simulation will stop.
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-t Output controls – General

Variable	Column	Format	Description
IPRTYPE	1 to 5	I5	Print type: - general printout only (no detailed force accounting) – detailed print (general + detailed force calcs)
LIMITOUT	2 to 10	I5	If IPRTYPE=1, there will be "LIMITOUT" cross sections with detailed output provided downstream of each leading edge.

Data I-u Output controls – time based

Variable	Column	Format	Description
TIMETRIGINC	1 to 5	I5	Number of time steps between output printing
NTOTHER	6 to 10	I5	If other specific times are required for output, indicate the number of the other print junctures.

Data I-v Output controls – increments in cross sections for general output

Variable	Column	Format	Description
SECTIONINC	1 to 5	I5	Print of results required at increments of cross section numbers (for general output option only). E.G. If SECTIONINC=10, general printout will be triggered every 10 cross sections (when printout occurs).

Data I-w Output controls – for specific time steps

Variable	*Column*	*Format*	*Description*
TIMETRIGGER(1)	1 to 8	I8	Timestep at which printout of ice results is requested
TIMETRIGGER(2)	9 to 16	I8	Next timestep at which printout of ice results is requested
Etc. – continue until "NTOTHER" time steps have been specified for out put	17 etc.	I8	Next times step, as required. *Note: if NTOTHER=0, then the I-u line of data is not included in TAPE5.txt*
+ other lines of data, if necessary to specify NTOTHER values of TIMETRIGGER			

Data I-x Output controls – based on increments in leading edge progression

Variable	*Column*	*Format*	*Description*
DISTRIGINC	1 to 5	I5	Increment in leading edge advancement that triggers output
NDOTHER	6 to 10	I5	If other specific leading edge locations are required for output, indicate the number of the other print locations.

Data I-y Output controls – for specific points of leading edge progress

Variable	*Column*	*Format*	*Description*
DISTRIGGER(1)	1 to 8	I8	Leading edge location (i.e. cross section number) at which printout of ice results is requested
DISTRIGGER(2)	9 to 16	I8	Next location at which printout of ice results is requested
Etc. – continue until "NDOTHER" time steps have been specified for out put	17 etc.	I8	Next location, as required. *Note: if NDOTHER=0, then the I-x line of data is not included in TAPE5.txt*
+ other lines of data, if necessary to specify NDOTHER values of DISTRIGGER			

Data I-z Ice generation method

Variable	*Column*	*Format*	*Description*
ICEGENMET-HOD	1 to 5	I5	If ICEGENMETHOD= 1– detailed calculation using heat loss estimated by Water Quality subroutines embodied in ONE-D. – Simplified method using user-defined heat

Variable	*Column*	*Format*	*Description*
			loss coefficient
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-zz Heat loss coefficient if ICEGENMETHOD=2 (see Section 4_)

Variable	*Column*	*Format*	*Description*
HLC	1 to 10	F10.0	If ICEGENMETHOD=2, HLC must be provided, in Watts per square metre per degree Celsius. Note that if ICEGENMETHOD=1, this line of data is not included in TAPE5.txt.
Header(j), j=1,10	6, 45	10A4	"Reminder" description of variable

Data I-aa Controls for specification of ice volumes inflowing to the study reach

Variable	*Column*	*Format*	*Description*
NTT	1 to 10	I10	Number of equal-length periods for which the total period, TTINT, will be broken, and a value of ice inflow provided for each
TTINT	11,20	F10.0	Total number of time steps at which ice inflow volumes are provided.

Data I-ab Controls for specification of ice volumes inflowing to the study reach

Variable	*Column*	*Format*	*Description*
USICEVOL(j, j=1,NTT)	1 to 80	Free format (requires blank or comma between values)	Ice volume inflowing during specific parts of the overall period TTINI (in cubic metres). See Section __ for more detailed description.
HEADER (I), I=1,20)	1 to 80	20A4	A header line that splits Data Type I from Data Type J.

J DATA GROUP
CONSTRAINTS AND MISCELLANEOUS INFORMATION FOR ICE

Data J-a Ice cover strength parameters

Variable	*Column*	*Format*	*Description*
ZZK1TAN	1 to 10	F10.0	K1 times Tan p, see Section _.
ZZK2	11 to 20	F10.0	K2 – see Section _.
Header (j, j=1,10)	21 to 60	10A4	"Reminder" description of this line's contents

Data J-b Method to estimate Manning n-value of ice under surface

Variable	*Column*	*Format*	*Description*
ICEnOPT	1 to 5	I5	Option number for n-value derivation: 1 – Method by Beltaos – Method by KGS Group – User specified ice n-value
Header (j, j=1,10)	11 to 50	10A4	"Reminder" description of this line's contents

Data J-c Manning n-value information for each cross section

Variable	*Column*	*Format*	*Description*
IX	1 to 5	I5	Cross section number
FACTOR1	6 to 10	F5.0	Coefficient for Beltaos Method
FACTOR2	11 to 15	F5.0	Manning n-value for KGS Method for 8-m thick ice
FACTOR3	16 to 20	F5.0	User-specified n-value
CNBED	21 to 25	F5.0	Manning n-value for riverbed
Header (j, j=1,13)	26 to 70	13A4	"Reminder" description of this line's contents

Data J-d Border ice prediction method (Refer to Section 2.2.2)

Variable	*Column*	*Format*	*Description*
IBORD	1 to 5	I5	Option number for calculation of border ice advancement user defined – as a function of days elapsed in the simulation Modified Newbury method Matousek method
Header (j,	6 to 80	18A4	"Reminder" description of this line's contents

Variable	*Column*	*Format*	*Description*
j=1,10)			

Data J-e Time to start Border Ice Generation

Variable	*Column*	*Format*	*Description*
DAYBORDSTART	1 to 10	F10.0	Start of border ice generation, in days elapsed since start of simulation
Header (j, j=1,10)	11 to 78	17A4	"Reminder" description of this line's contents

Data J-f Border ice breakup triggers

Variable	*Column*	*Format*	*Description*
BORDUPBRK	1 to 10	F10.0	Rise in water level that would trigger border ice breakup (m)
BORDDOWNBRK	11 to 20	F10.0	Decrease in water level that would trigger border ice breakup (m)
Header (j, j=1,10)	6 to 80	15A4	"Reminder" description of this line's contents

Data J-g Border ice parameters for Newbury method (if IBORD=2)

Variable	*Column*	*Format*	*Description*
IX	1 to 5	I5	Cross section number
BCF1	6 to 10	F5.0	Newbury Coefficient (see Section _)
BCF2	11 to 15	F5.0	Newbury Coefficient (see Section _)
BRDT	16 to 20	F5.0	Border ice thickness (m) ; if BRDT=0, then default value of 0.2 m is adopted.
Header (j, j=1,10)	21 to 80	15A4	"Reminder" description of this line's contents

Data J-h Border ice parameters for user-defined method (if IBORD=1)

Variable	*Column*	*Format*	*Description*
IX	1 to 5	I5	Cross Section number
BCF1	6 to 10	F5.0	Coefficient for user-specified border ice advancement
BRDT	11 to 15	F5.0	Border ice thickness for user-defined method; if BRDT=0, then default value of 0.2 m is adopted.
Header (j, j=1,16)	16 to 79	16A4	"Reminder" description of this line's contents

Data J-i Border ice parameters for Matousek Method (if IBORD=3)

Variable	*Column*	*Format*	*Description*
BRDT	1 to 10	F10.0	Border ice thickness for Matousek Method (all cross sections); if BRDT=0, then default value of 0.2 m is adopted.
Header (j, j=1,17)	11 to 78	17A4	"Reminder" description of this line's contents

Data J-j Controls for Ice Cover Melting

Variable	*Column*	*Format*	*Description*
MELTOPT	1 to 5	I5	Option for melt calculations: - User-define heat transfer coefficient from water to ice - Algorithm devised by original RIVICE developers
MELTSTART	6 to 10	I5	Time step number at which the calculations of ice cover melting will commence
Header (j, j=1,10)	11 to 78	17A4	"Reminder" description of this line's contents

Data J-k Heat transfer coefficient (if MELTOPT=1)

Variable	*Column*	*Format*	*Description*
HWI	1 to 10	F10.0	Heat transfer coefficient from water to ice
Header (j, j=1,10)	6 to 80	15A4	"Reminder" description of this line's contents

7.2 NOTES ON SELECTION AND INPUT OF ICE PARAMETERS

Understanding the implications of selecting input parameters is a critical part of effective use of RIVICE. To assist the user in thoroughly understanding the capability of the software, a series of notes and hints have been assembled and are organized in parallel with the input variable types I and J, whose formats are listed in Section 7.2. Some Data Types are reasonably self-explanatory and notes are not warranted and have not been included below.

Data I-a: The original strategy that was envisaged for RIVICE was to support two possible modes of ice evolution, one for formation, and the other for spring break-up (see Section 2.2 for further explanation). This has not been carried through to this version of RIVICE. Only one value of IEVOL is allowed (a value of "1"). This does not necessarily mean that the program cannot be used to address these two modes, but rather the means of doing so is embodied in the flexibility of defining the various parameters that control the ice accumulation. A good example would be the selection of the value of cohesion in the ice pack. The simulation of a spring ice jam would almost always be with a value of cohesion of zero, whereas the simulation of a formation jam under sub-zero air temperatures might include a value for cohesion that is greater than 0 Pa.

Data I-b: The three options to represent the process of ice deposition under an established ice cover are described in Section 2.2.6. By far the most common method will be the use of a critical velocity above which deposition will not occur. The two optional methods (Meyer Peter bed-load sediment analogy, and the use of a densimetric Froude number) are much more experimental and the user is advised that there is no widespread usage of either of these techniques and the results may or may not be representative of real ice covers. Note that there is an alphanumeric Data that is allowed at the end of this line of data. It is meant to be a reminder for the user when modifying each line of data, without the need for referral to the User's manual. This information is read as the variable "HEADER". It is also used on many of the subsequent lines of ice data.

Data I-c: If ice deposition is based on a maximum velocity, the user must specify the value for the program (VDEP). This value is believed to range from as low as 0.7 m/s for frazil/slush ice particles to possibly as much as 1.5 m/s for spring ice jams where transported ice is in the form of ice floes formed by the break-up of a previously intact ice cover.

Data I-d: If the user wishes to use the Meyer Peter bed-load sediment analogy, the critical Data variable for this technique is the average size of the ice particles that will be in transport under the ice cover (variable name DIAICE). There is almost no credible information on this element from field investigations, and the user must resort to experimenting with a range of values of DIAICE, until the results best represent the situation that he/she is attempting to simulate.

Data I-e: Two options for representing the process of erosion of the ice cover by high velocity are described in Section 2.2.6. The most common method will be the selection of a critical velocity above which erosion of the ice cover would occur. The tractive force approach is less well used and the precise threshold value of the tractive force that would initiate ice cover erosion is not known. However, a value in the range of 1.0 M/S is believed to be a reasonable estimate.

Data I-f: The minimum velocity that would initiate erosion is not well known and is almost certainly variable depending on the type of ice cover involved. For example, erosion of slush ice deposits soon after they have formed might be expected to erode at velocities as low as 0.9 m/s. On the other hand, erosion of consolidated and jammed ice covers might be as high as 2 m/s. It should be noted that velocities in this high range have a high potential to cause difficulties with stability of the numerical solutions within RIVICE, and the user is advised to use caution when erosion is observed to have occurred in the output under conditions of high velocities exceeding 1.5 m/s.

Data I-g: Tractive forces that would initiate erosion are not well defined. It is expected, however, that a tractive force in the range of _ to _ would likely be a representative limit.

Data I-h: The value of LEOPT determines which method of estimating the stability of the leading edge of the ice cover will be used. The three methods are described in Section 2.2.5. There is some flexibility in the algorithms that are used, as follows:

- LEOPT=1, then the method is that proposed by Pariset and Hausser. The only variation in the values applied can be forced by changing the porosity of the ice approaching the ice cover (PORSFS – see Data I-L). The higher the porosity, the lower will be the approach velocity at which ice is forced to submerge at the leading edge. A porosity between about 0.5 and 0.8 is believed to best represent leading edge conditions.
- LEOPT=2, then the user has no capability to adjust the algorithm, as it is coded in RIVICE in the form that Ashton devised (see Section 2.2.5 for the variables in the algorithm).
- If LEOPT=3, then the leading edge is forced to be equal in thickness to the specified value by the user (Data I-i). All incoming ice accumulates at the leading edge, and no ice is allowed to submerge and travel under the downstream ice cover if LEOPT=3.

Data I-j: The value of VFACTR affects the speed at which ice in transport under the ice cover moves. There is no known field data that could support a selection of this variable, and this factor is included as a means to permit numerical sensitivity tests and future research. All tests of RIVICE have used a value of 1.0 and no experiments have been launched to date on values less than 1.0.

Data I-k: The value of POROSC is used primarily to compute the volume of the ice cover that is formed from an incoming quantity of solid ice. Typical values for POROSC are believed to range from 0.5 to 0.8). Note that this is intended to be the porosity of the ice cover ***after*** it has formed.

Data I-l: The value of POROSFS is used primarily in conjunction with SLUSHT (Data I-m) to estimate the surface area of slush ice pans that insulate the surface of the open water. It also can be varied to affect the leading edge algorithm in the case where LEOPT=1 (see Data I-h).

Data I-p: There will be as many lines of Data Type I-p as has been defined in Data I-n by the variable NBRDGSW. The current limit of ice initiations (or ice bridges) is 5.

Data I-r: This is the first of a series of Data that control the type of tabular intermediate output that stored as results during a simulation.

If IPRTYPE=0 then only general output is requested, without any details on estimated forces within the ice cover. Output for this general type includes:

- The time of the data being generated (in days and time steps since the start of the simulation)
- The number and locations (leading edge, training edge) of ice segments
- Cross section number
- Location in metres from the upstream end of the reach
- Discharge (m^3/s)
- Water level (m)
- Total top width (m)
- Total border ice width (m) (left plus right sides)
- Average velocity (m/s)
- Manning n-value of river bed
- Manning n-value of ice cover under surface
- Composite Manning n-value of bed/ice
- Thicknesses of ice cover (m). If this vale is greater than 0.0, then there is a full ice cover

extending across the river at this location. If the thickness is zero, there is open water potentially generating ice at this location, and there can be border ice.

- Ice volume in transit (m^3/s). In open water this is the slush ice being carried with the flow, and where the ice thickness is greater than zero, it represents ice being carried with the flow under the ice cover. Note that this is an estimate based on the volume of ice currently at this cross section, and the velocity at this location.

If PRTYPE=1, then general output is requested, plus detailed output of ice forces and other related information. This will include:

- The first six bulleted items listed above
- Cross sectional area available for flow (m^2)
- Ice thickness (m)
- Froude Number of flow at the leading edge
- Volume of ice in transit under the ice cover at this location (m^3)
- Hydraulic thrust at the leading edge
- Drag force of the flow under the ice cover
- Component of weight of the ice cover acting down the slope of the river
- Load shed from the ice to the river banks
- Residual force within the ice cover than must be resisted by the internal strength of the ice cover
- The minimum thickness required at this location to resist the hydraulic loads without causing a shove. This is a useful indicator that shows that if the value of ice thickness is very close to the value specified as the minimum stable thickness, then the ice cover evolution has been governed by the hydraulic forces, rather than, for example, the thickness that forms at the leading edge, or deposits of slush ice that have been caused by submergence of incoming ice at the leading edge.

Note that the value of LIMITOUT controls the number of cross sections downstream of each leading edge that detailed output will be provided. This allows the user to view the most critical zone of ice evolution near the leading edge, without triggering output of many more cross sections where the ice is not evolving.

Data I-u: This line provides specific time steps at which output is requested, in addition to the increments in time stipulated by TIMETRIGINC in Data I-s. There can be as many as 20 TIMETRIGGER's, and since there can only be a maximum of 8 on each line, there could be as many as 3 lines of Data I-u data.

Data I-v: This line controls output that may be rested at specific junctures of ice front advancement, and supplements the data output that is triggered on the basis of time alone.

Data I-x: This Data line(s) provides the opportunity to define specific ice front locations at which output is required. It is a list of values of DISTRIGGER, and the dimension limits this to not more than 20 locations. Given the format requirements, this could require up to three separate lines of data. The distances defined must be equal to the chainage at the location specified. For example if the spacing of cross sections has been selected to be 20 m, and a cross section chainage of 9880 m exists and is desired as a location for output, then that precise value must be specified. Selection of say 9881m will not trigger output as the leading edge of the ice cover advances past this location. ***A value not equal to a specific cross section chainage will be ignored.***

Data I-z: This Data is required if the option for a simplified calculation of heat loss from the open water surface is requested (ICEGENMETHOD=2). It defines the assumed heat loss coefficient between the water surface and the air (in watts per square metre per degree Celsius). Note that if ICEGENMETHOD is specified as "1" in Data Type I-y (i.e. detailed calculation of heat loss), then this line of output will be ignored, but still ***must be included***. Also note that if ICEGENMETHOD=2, it is assumed that the water temperature is zero degrees at all locations where open water exists. The simplified method does not permit the cooling of warm inflowing water to the point of freezing. If that process is desired by the user, then he must select ICEGENMETHOD=1.

Data I-aa: The two values specified in Data Type I-aa permit relatively easy means of specifying the volume of incoming ice into the reach that the user would like to simulate. If desired, this source of incoming ice can be the only Data of ice (air temperatures could be assigned positive values), and can be used to represent the accumulation of a spring ice jam at a pre-selected lodgement location downstream. The first value, "NTT", indicates the number of portions of equal time that the total period "TTINT" will be broken. For example, if NTT=3 and TTINT is 30,000 then there will be three periods of time for which ice volumes will be defined, each being 10,000 time steps in length.

Data I-ab: This line or series of lines contain each value of ice inflow at the upstream end of the reach, for each of the periods of time that have been identified in Data Type I-aa. The value of each is in cubic metres. For example, if a value of 60 cubic metres is provided, and the time step is 30 seconds in length, then the value of USICEVOL will represent a steady inflow of 2 m^3/s of ice.

Data J-a: The parameters supplied on this line relate directly to the values stated in Section 2.2.7 for the ice strengths.

ZZK1TAN: $K_1 * \tan \varphi$

ZZK2: K_2 (coefficient greater than or equal to 1.0, comparable to a Rankine passive coefficient in soil mechanics)

These values are applied to all cross sections with a full ice cover and do not vary during the entire simulation. Note that if an ice bridge has been requested, and a thermal ice cover is requested to exist downstream of that location, then no shoves are possible in that downstream zone.

Data J-c: This line provides details regarding the method of Manning n-value estimation that has been requested by the user. It is possible through this data type that the parameters for n-value estimation can be separately defined for each and every cross section, if necessary. The user simply provides one line of data for each change in the parameters that is desired. So, for example, if there are 500 cross sections and the user would like one set of n-value parameters to be used in the upstream 200, and then a different set for the downstream 300 cross sections, he would only have to provide the values of the parameters for the cross section 200, and 500. The ones in between would be automatically set (i.e. 1 to 200 would be use the first parameters, and 201 to 500 would use the second set of parameters).

The parameters supplied through each line of Data Type J-c are:

IX: cross section number (all cross sections between the previously defined cross section and this one will be assigned the values provided).

FACTOR1: Coefficient in Beltaos method of n-value estimation (usually about 0.4 to 0.6). Refer to Section 2.2 8.

FACTOR2: Coefficient in KGS Method (actually represents the estimated n-value for an 8-m thick ice cover). Refer to Section 2.2 8.

FACTOR3: Manning n-value selected by the user for this cross section, and all the ones previous, back to the last defined cross section number. Refer to Section 2.2 8.

Regardless of the method that is selected, values for all of these "FACTORS" should be provided.

CNBED: The n-value for the riverbed for this and previous cross sections.

Data J-g to J- i: Depending on the type of border ice calculation that has been selected, the user must provide the type of data that pertains to that method in the way prescribed for that data type. For example, if Data J-d has specified IBORD to be "2", then the Data must include Data Type J-g, and exclude Data Types J-h and J-i.

8. RIVICE OUTPUT FILES

8.1 STRUCTURES / FILES

As shown in Figure 1-1, a total of 20 output files are generated by a single run with the ONE-D program. Some of these files are very useful, and some are unnecessary to keep. The following paragraphs describe each file and its contents:

TAPE6.TXT

General Points

This ASCII output file begins with a listing of all the Data as read from TAPE5.TXT by the ONE-D program. The user can control the level of detail in part of this listing to some extent. By selecting one of three options for the value of variable IDTABL(K) on Data B-b, the user can obtain different quantities of hydraulic tables for both Data and interpolated cross sections. This variable is set individually for every reach. The Data echoed in this part of TAPE6.TXT is in a more readable form than the TAPE5.TXT file, and contains warning and error messages for certain data anomalies. It also contains a re-statement of the ice parameters that have been selected by the user.

The second part of the TAPE6.TXT file contains the following:

- Intermediate calculation results for equivalent Manning's n estimates for those reaches which use entrance and exit loss coefficients
- Echo of the TAPE51 - TAPE54.TXT Data files for the first time they are read, if and when sea dam changes occur in the model that require these files
- QFCS discharge calculation results for each time step.
- Intermediate results of the ice cover simulation as requested through Data Types I-r through I-x (see further explanations in Section 7.3 above.

These intermediate results are interspersed with each other depending in the order in which they are calculated by the program, i.e., they appear in order of the time step in which they occur. The end of the TAPE6.TXT file is the most likely location for error messages to appear when a run terminates prematurely.

TAPE6.TXT files are usually quite large, typically varying from 200 Kb to 15 Mb, depending on the size of the model and the number of time steps of the run that generated it. After the initial model building phase, the echo of the Data is of little use, unless a problem occurs and the user wishes to ensure that the program is reading the Data correctly.

Output of Intermediate Ice Results

All output that shows the evolution of the ice cover is located in Tape6.txt. The amount of output is controlled directly by the user through the parameters that are listed in Input Data.

TAPE61.TXT

This ASCII output file contains listings of hydrographs for all the locations requested in the G Data Group. One line of data is provided for each time step, and that line includes the time, water surface elevation, depth, discharge, velocity and Manning's n. The hydrograph listings are followed by listings of profiles of every reach in the network for all the time steps requested in the G Data Group. One line of data is provided for each mesh point, and the data includes the distance from the upstream end of the reach, water surface elevation, depth, discharge, velocity and Manning's n. The size of the TAPE61.TXT file is entirely dependent upon the quantity of hydrographs and profiles requested by the user.

TAPE621.TXT, TAPE622.TXT, THROUGH TAPE630.TXT

The data for each pump station in a model is written to one of these ASCII files, therefore there is a limit of ten pump stations per model. This limit can be exceeded, if necessary, with some simple changes to the FORTRAN source code. Should the need occur to model more than ten pump stations, the user is advised to contact Environment Canada for these changes (see contact name in the Introduction in Volume 1).

For each pump station in the model, a file is produced that contains a table with the following data for each time step:

- water level at the intake
- water level at the outlet
- water level at the remote switch site (if one exists)
- mode of pump station operation (normal mode or storm mode)
- status of each pump
- discharge for each pump

Of the total of ten files created, those which are not used (when there are fewer than ten pump stations) will be empty and may be erased by the user. It is anticipated that the next version of ONE-D will be able to create the appropriate number of pump station files automatically, without any empty files and a greatly increased limit on the number of pump stations.

The size of each pump station file will depend solely on the number of time steps involved in a model run.

TAPE10.TXT

This is a large binary file that contains all the computed water levels and discharges at every mesh point in a model network for every time step in a model run. It does not contain any other hydraulic data, such as velocities. The data is packed into the file with no internal map of the data structure. It is necessary to know the sequence of reaches and the number of mesh points in each reach in order to locate the mesh point to which each elevation-discharge pair belongs.

The structure of the TAPE10.TXT file is organized in repeating blocks, with the major repeating block being the time step. Within each time step, the data is organized by reach, in the order that each reach appears in the reach-node connectivity table. Within each reach, the data is listed by mesh point, starting at the upstream end of each reach. The data simply consists of water surface elevation, Z, followed by discharge, Q, written in single precision for each consecutive mesh point. At the end of each reach, an end of record marker is written to the file.

The option exists in the ONE-D program to not write any data to the TAPE10.TXT file. This may be acceptable for small models where all the necessary information can be obtained from the requested hydrographs and profiles tabulated in TAPE61.TXT. However, as shown in Figure 1-1, TAPE10.TXT is a necessary link to the post-processors, REMAT and BUILD. It can also be used for "warm start" runs (see Section 3.1, "Initial Conditions"), from any time step of an earlier run.

If a run terminates prematurely, any output computed up to the time of failure is generally written to this file. This is a very important feature, since it allows the post-processors to be used to help analyze the cause of the failure.

The size of a TAPE10.TXT file is proportional to the number of mesh points multiplied by the number of time steps in a run. For long runs with a large model, it is not unusual for the TAPE10.TXT file to exceed 20 Mb in size.

TAPE11.TXT, TAPE12.TXT, THROUGH TAPE15.TXT

These are binary files which are used during execution to hold data temporarily. They serve no useful function after a run is completed and may be erased or ignored, as each subsequent run will overwrite existing files by these names.

TAPE16.TXT

This is a binary Data file that is used only for a "warm start" run. If the file does not exist prior to a ONE-D model run, an empty file by this name is created, therefore it is also an output file. If it does exist, then it is left unaltered by subsequent runs. It contains water level and discharge data for every mesh point in a model network which will become the initial conditions for a run if a TAPE16.TXT warm start is specified. The file is created by renaming a TAPE17.TXT file created by an earlier ONE-D run to this name. For additional information, please refer to "Initial Conditions" in Section 3.1, and to the TAPE17.TXT description below.

TAPE17.TXT

This is a binary hydraulic only output file which contains water level and discharge data for every mesh point in a model network for the last time step of a run. It is designed to be used as a source for initial conditions for a subsequent "warm start" run, starting at the point when the earlier run stopped. Unlike the TAPE10.TXT warm starts, a run must terminate normally for a TAPE17.TXT file to contain any data. For additional information, please refer to "Initial Conditions" in Section 3.1, and to the TAPE16.TXT description above.

FILENAME.TXT

Individual files for hydrographs and profiles are generated by this routine which is available in **TAPE6** printer output. The routine provides an option for the same hydrographs and profiles to be produced as separate files for each location requested in **TAPE5** input. The format of the file name is as follows: "**H_01_000600.txt**" or "**P_01_000050.txt**"

First character "**H**" or "**P**" are for hydrograph or profile; first two digit numbers is the reach number; second number is the location or "distance" along the reach for hydrographs or "time-step" for profile along the specific reach. These files can be further manipulated to produce individualized graphical output such as in EXCEL or other similar plot capable packages.

9. A RECENT CASE STUDY

Simulating the effects of dredging on ice jam flooding along the Lower Red River

9.1 INTRODUCTION

Ice jams are a common occurrence during the onset of spring flooding along the Lower Red River, the most-downstream reach of the Red River between Winnipeg and Lake Winnipeg. Ice jams have occurred in this area for all of recorded history (Acres, 2004; Farlinger and Westdal, 2010) and are a frequent problem in regards to local flooding. Trends show that the spring flood hydrographs begin earlier and rise steeper than in past decades causing break-up to be more severe and prone to ice jamming (Lindenschmidt et al., 2010).

In response to this tendency of increased frequency and severity of ice jamming and ice jam flooding, provincial and municipal government agencies have implemented hydraulically-operated amphibious excavators (Amphibex) to reduce the risk of ice jams in this area (Topping et al., 2008). The ice is pre-cut with trenches using tracked vehicles equipped with rotary blades and saws to allow easier breakage of the ice.

Although the Ice Jam Mitigation Program has reduced the frequency of ice jam occurrences and the magnitude of ice jamming flooding, within the Selkirk area, ice jams still occur along the most downstream portion of the reach before it empties into Lake Winnipeg. This area is the river's delta where the river is very flat-bottomed and sallower compared to the upstream stretch. Dredging the river bed in this area to increase the depth has been proposed to reduce the occurrence of ice jamming and its impacts.

Dredging of the Red River north of Lockport started in the late 1800s and continued until 1998, when the Government of Canada reduced its dredging program. The Government of Canada eliminated its involvement entirely in 1999, and there has been no further dredging carried out since.

9.2 STUDY SITE

The Lower Red River is the most downstream reach of the Red River extending from the Assiniboine River confluence in Winnipeg (The Forks) to the Red River outlet at Lake Winnipeg (see **Error! Reference source not found.**). The total drainage area of the Red and Assiniboine river watersheds is approximately 287,500 km^2.The average flow at Lockport is 244 m^3/s where maximum and minimum flows of 4330 and 14 m^3/s, respectively, have been recorded.

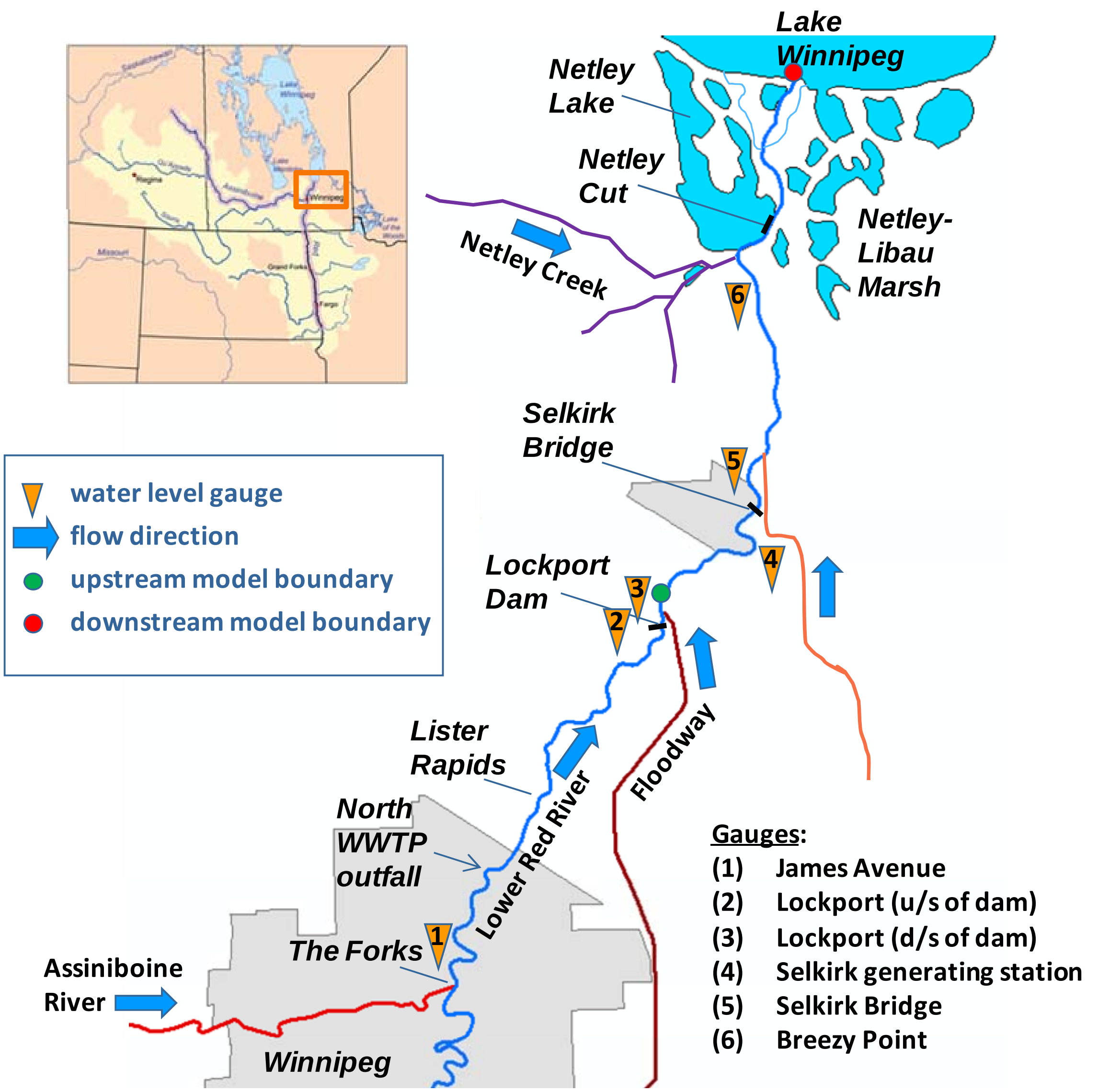


Figure 1: The Lower Red River between The Forks and Lake Winnipeg

A lock and dam is situated at Lockport, which was built in 1910 to allow navigation along the river between Winnipeg and Lake Winnipeg. The dam has steel curtains that dam the river for navigation and roll up to allow flood waters from the spring freshets to pass. The lock and dam was built to allow navigation over a series of five rapids including an approximate 4 m drop in elevation around Lister Rapids. Just downstream of Lockport is the outlet of the Floodway, a channel that diverts spring floodwaters from the Red River south of Winnipeg to protect the city of potentially high flooding. Between Selkirk and Lake Winnipeg, the river flows through a delta system called the Netley-Libau Marsh. The marsh is very flat and consists of many small bodies of water interconnected by a network of channels with the Red River. A 400 m long cut short-circuits water from the river into Netley Lake.

A longitudinal profile of the river's thalweg and ice cover level, typical at the end of winter, is shown in

Figure 2. Generally at the end of winter, the water level gradient between The Forks and Lister Rapids is low (≈ 0.00005 m/m). The river bottom becomes steeper between Lister Rapids and Lockport and the water level, too, at the end of winter, is steeper (≈ 0.00015 m/m) compared to the rest of the river stretch. Due to backwater effects from Lake Winnipeg and water level gradient along the most downstream portion of the river, between Lockport and Lake Winnipeg, can be almost flat (< 0.00001 m/m).

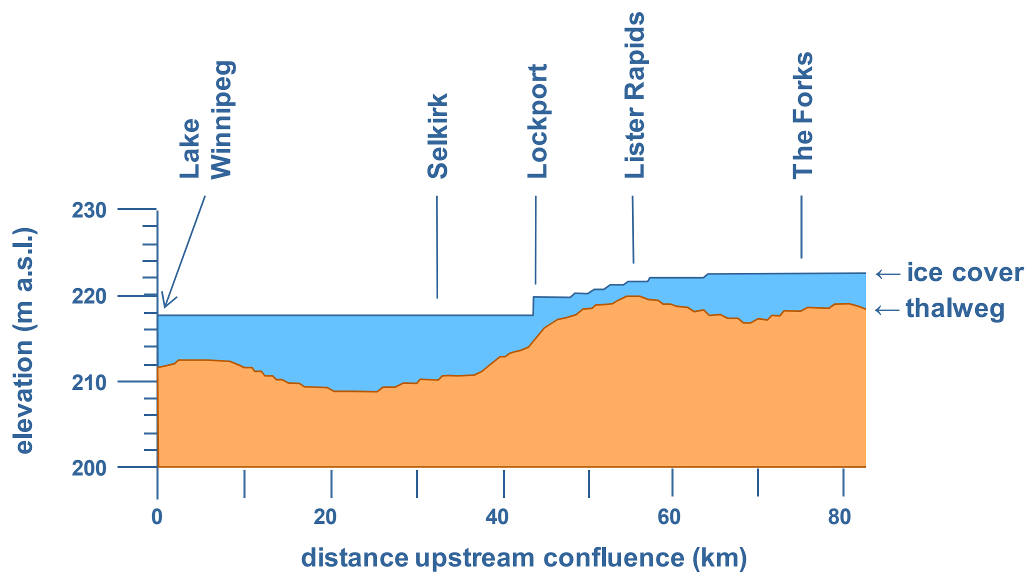


Figure 2: Longitudinal profile of the Lower Red River's thalweg and typical winters' end ice cover level (modified from Geological Survey of Canada http://gsc.nrcan.gc.ca/floods/redriver/geomorphology e.php)

9.3 LOWER RED RIVER ICE

The ice cover season along the Lower Red River typically extends from November to April. The ice cover is generally smooth, and once formed, tends to remain in place through the entire winter and has been measured to be up to 1 m thick in this region. Spring flooding is frequently exacerbated by mechanical ice breakup and ice jamming, especially during early and rapid melt events. Since recorded history, ice jams have plagued this area. The river at Selkirk and north of Selkirk are particularly prone to ice jam flooding. Historical newspaper articles indicate that serious ice jams occurred on the Red River near Selkirk as early as the mid to late 1800s (MB, 2010).

9.4 BREAKUP OF ICE ALONG THE LOWER RED RIVER

Ice breakup has been found to usually occur in the 990 m^3/s to 1420 m^3/s range. Historically, locations most prone to ice jamming are at the Selkirk Bridge, Sugar Island, PTH 4 Bridge and downstream of the PTH 4 Bridge. In the Breezy Point to Netley Creek area, breakup likely occurs at somewhat higher flows (up to 2690 m^3/s)) as the ice in this area is typically more competent "lake" ice (Acres, 2004).

Typically, the ice cover initially opens at the North Perimeter Bridge and over the next few days ice moves in the reach between the North Perimeter Bridge and Selkirk. An early rain event may exacerbate this situation. On occasion, the ice jams along this stretch and causes local flooding, as was the case along River Road in the spring of 2009. The ice movement is arrested at Selkirk to form a jam. This may occur first at the golf course, and progress downstream to the Selkirk Bridge and Sugar Island. This usually causes flooding of the east approach to the Selkirk Bridge requiring the bridge to be closed to traffic. The jam pushes past Sugar Island to the PTH 4 Bridge.

Parallel to these events, the ice cover may break up north of PTH 4 Bridge and cause jamming at various points, typically along McIvor Lane and at the Netley Creek confluence. Ice then moves further downstream and its initial surge is diverted into Netley Lake through Netley Cut. Jamming in this area very often is accompanied by water backup into Netley Creek causing local flooding at Petersfield. Recent years of severe ice jamming with flooding are 1996, 2004, 2007, 2009, 2010 and 2011. Attention will be focused on the ice jamming during the 2010 spring break-up of the ice cover along the Lower Red River.

2010:

Artificial ice cutting and breaking was carried out 1. – 20. March 2010. Ice cutting was carried out from just south of Selkirk to Netley Marsh. Ice breaking took place between Netley Creek and Netley Cut, at Netley Lake, McIvor Lane and PTH 4 to Selkirk Golf course.

A rainfall event occurred on 10. March which was followed by a week of above 0°C daytime temperatures. Snowmelt occurred rapidly with no snow recorded on the ground at the Oakbank weather station by 18. March. The increased runoff caused the ice cover to open up at North Perimeter Bridge and the ice cover break-up progressed downstream until its front reached St. Andrews on 16. March and south Selkirk on 23. March. Figure 3 provides a SPOT-5 satellite image with the ice accumulation front at St. Andrews and Figure 4 shows a RADARSAT-2 satellite image with the ice jam at south Selkirk.

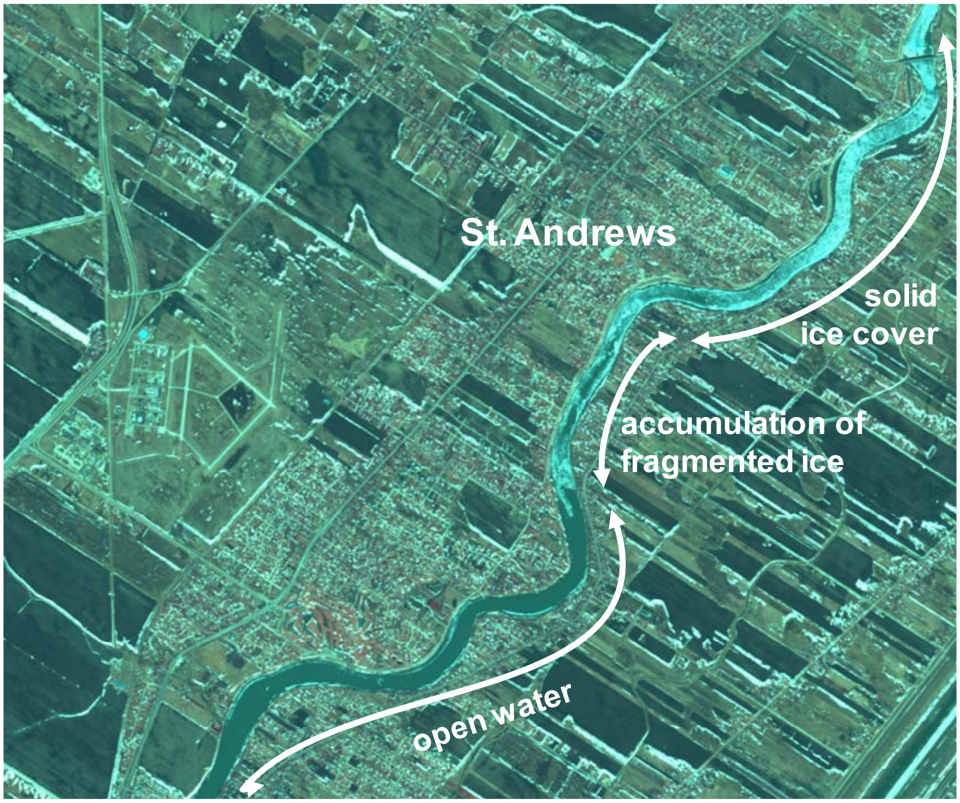


Figure 3: Multispectral SPOT-5 image of ice accumulation front at St. Andrews on 16. March 2010 (SPOT-5 image © 2010 CNES, Licensed by Iunctus Geomatics Corp, www.terraengine.com).

The ice cover shown in the RADARSAT-2 imagery had a smooth texture consisting predominantly of columnar ice (Lindenschmidt et al., 2011). A RADARASAT-2 image acquired on 6. March along the same river stretch allowed ice thicknesses to be calculated from the image signals (Lindenschmidt et al., 2010).

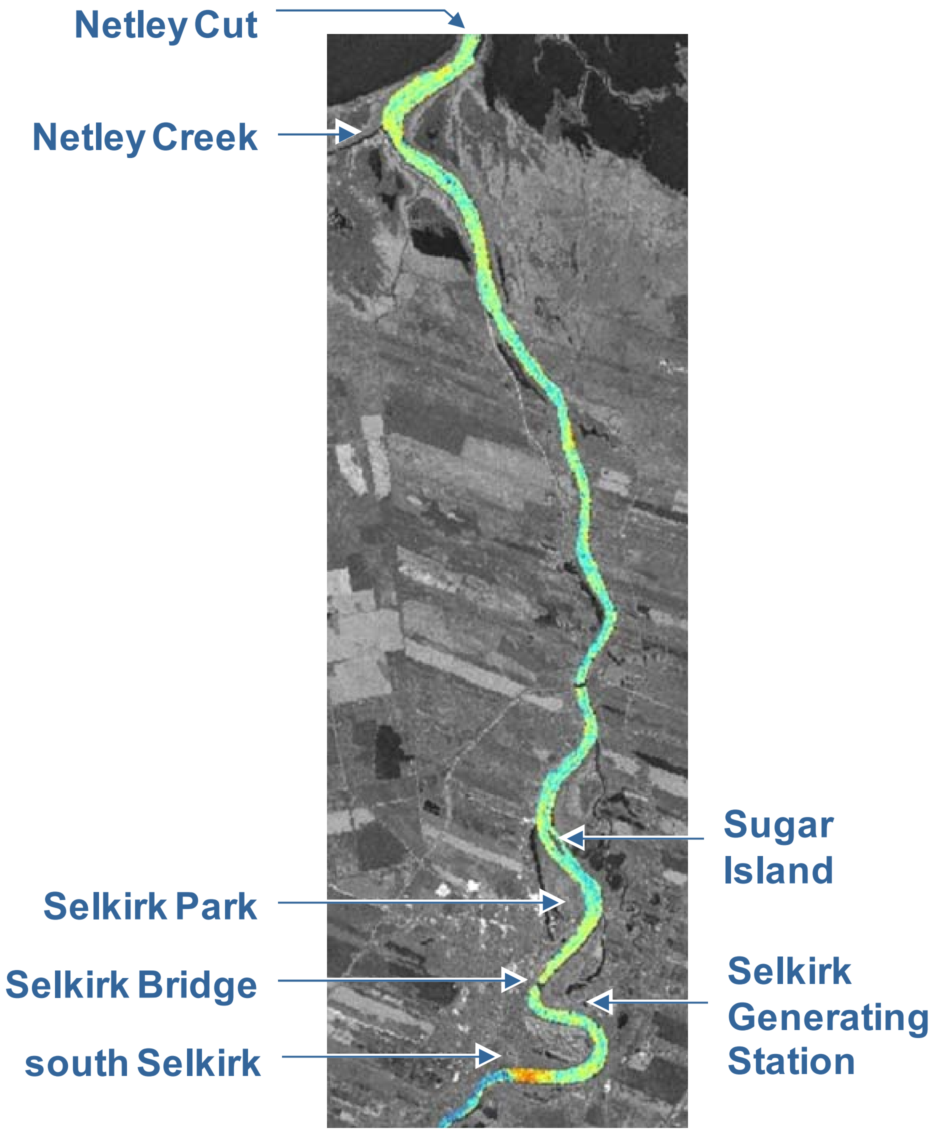


Figure 4: RADARSAT-2 image of the Red River ice cover between south Selkirk and Netley Cut on 23. March 2010. The bright red and orange signal at south Selkirk indicates the ice jam (RADARSAT-2 Data and Products © MacDonald, Dettwiler and Associates Ltd. 2010 – All Rights Reserved / RADARSAT is an official mark of the Canadian Space Agency).

he ice jam at south Selkirk released on 24. March and the ice cover continued to break-up until Selkirk Park. Ice jamming occurred at this location and remained in place until 27. March. During this time, fragmented ice from upstream accumulated at the ice jam, whose front juxtaposed upstream past the Selkirk Bridge and almost reaching the Selkirk Generating Station. A SPOT-5 image of ice jam extension from its toe at the park to its front at the generating station was captured on 26. March and is shown in Figure 5. This is the ice jam that serves as one of the modeling case studies described below.

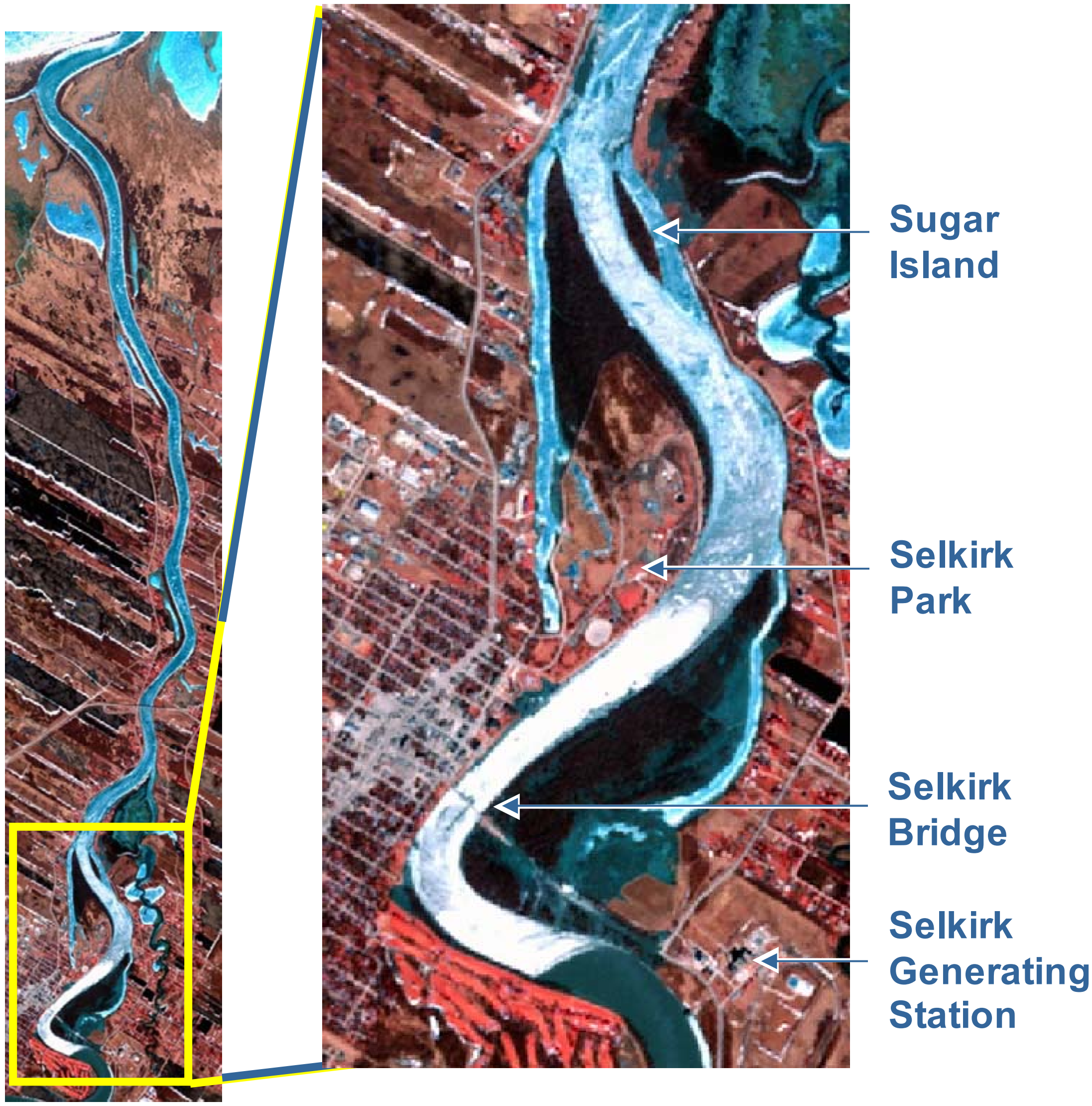


Figure 5: SPOT-5 image of ice jam at Selkirk on 26. March 2010 (SPOT-5 image © 2010 CNES, Licensed by Iunctus Geomatics Corp, www.terraengine.com).

The ice jam at Selkirk Park released on 28. March and the ice flowed downstream to jam again for a short period at the PTH#4 Bridge. By 29. March, this jam released and the ice cover front moved downstream to cause a jam at Netley Cut. The river reach between the Netley Creek confluence and Netley Cut is particularly susceptible to ice jams for several reasons:

i) tributary sediment export - Netley Creek drains a large agricultural area and much sediment is deposited into the Red River just downstream of the creek outlet into the Red River. An increase of up to 4 to 5 m in bed elevation is consistently measured in the flow direction as Red River water passes the Netley Creek confluence.

ii) sharp meander - the meander of the Red River at the Netley Creek outlet is very tight which can constrict the flow of water and ice during spring break-up of the ice cover.

iii) low slope - water level gradient along the most downstream portion of the river is almost flat (< 0.00001 m/m).

An aerial photograph of the ice accumulation at Netley Cut is shown in Figure 6. The photo shows ice fragments from the jam spilling through Netley Cut into Netley Lake. The ice jam produced the highest stages on record along the east bank between Selkirk and Breezy Point. Minor flooding occurred at Petersfield due to the ice jam backing water into Netley Creek. This ice jam is the second modelling test case described below. By 31. March, all the ice had cleared from the river up to its confluence at Lake Winnipeg, which usually maintains its ice cover into May.

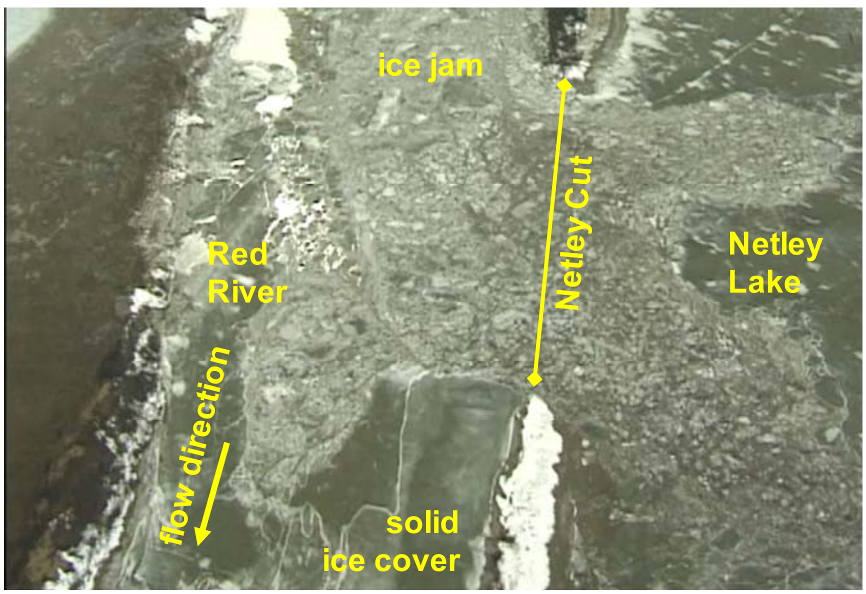


Figure 6: Ice jam at Netley Cut on 29. March 2010. Some of the fragmented ice has spilled through the cut into Netley Lake.

9.5 JUSTIFICATION FOR COMPUTER MODELLING USING RIVICE

To better understand the formation and behaviour of ice jams along the Red River, the Manitoba Government committed to expand its computer modelling repertoire: *"Considerable work is underway through Manitoba Water Stewardship to understand the mechanisms of ice formation on the Red River. Computer models are being developed to simulate the evolution and behaviour of ice jams along the Red River." (MB, 2010, p.6).* Due to the long extent of the Lower Red River (~75 km) required a one-dimensional approach (variables change longitudinally and are averaged at each cross-section) to be taken for such a modelling exercise in order to reduce computational expenditure and minimise data input. The most downstream portion of the reach flows through a delta, which caused particular challenges in modelling ice jams in this area. Those challenges and solutions to overcome them will be addressed in this paper.

Comparable studies of one dimensional ice-jam modelling in river deltas are sparse in the literature. One prevalent study is ice jam modelling of the Peace and Athabasca River delta (Beltaos, 2003). The papers highlights limitations in modelling such an area with low-lying topography and river banks, in particular "floodplain truncation and consequent neglect of overbank flows at high stages" (p. 3691). In addition, "distributed flow sinks due to overbank flow [were] ignored because there [was] no known method to quantify such sinks and the overbank topography [was] not known in sufficient detail." (p. 3691). "Neglect of such flow withdrawals from the main river can have significant impacts on modelling results, and this limitation has already been identified by Demuth et al. (1996)" (cited in Beltaos, 2003, p. 3696).

9.6 DATA FOR MODEL SETUP

The model extended from its upper boundary just downstream of the Lockport Dam and the Floodway confluence (see **Error! Reference source not found.**) to its lower boundary at the Red River confluence at Lake Winnipeg. Cross-sections of the river bed were available with average 250 m spacing from a bathymetric survey, but extended only from the upper model boundary to just upstream of the Breezy Point gauge. Additional bathymetric soundings from the Netley Creek confluence, Delta Forks and Red River confluence at Lake Winnipeg were made available by Public Works and Government Services Canada. An example of river bed elevation contours extracted from the soundings data acquired at the Netley Creek confluence is shown in Figure 7.

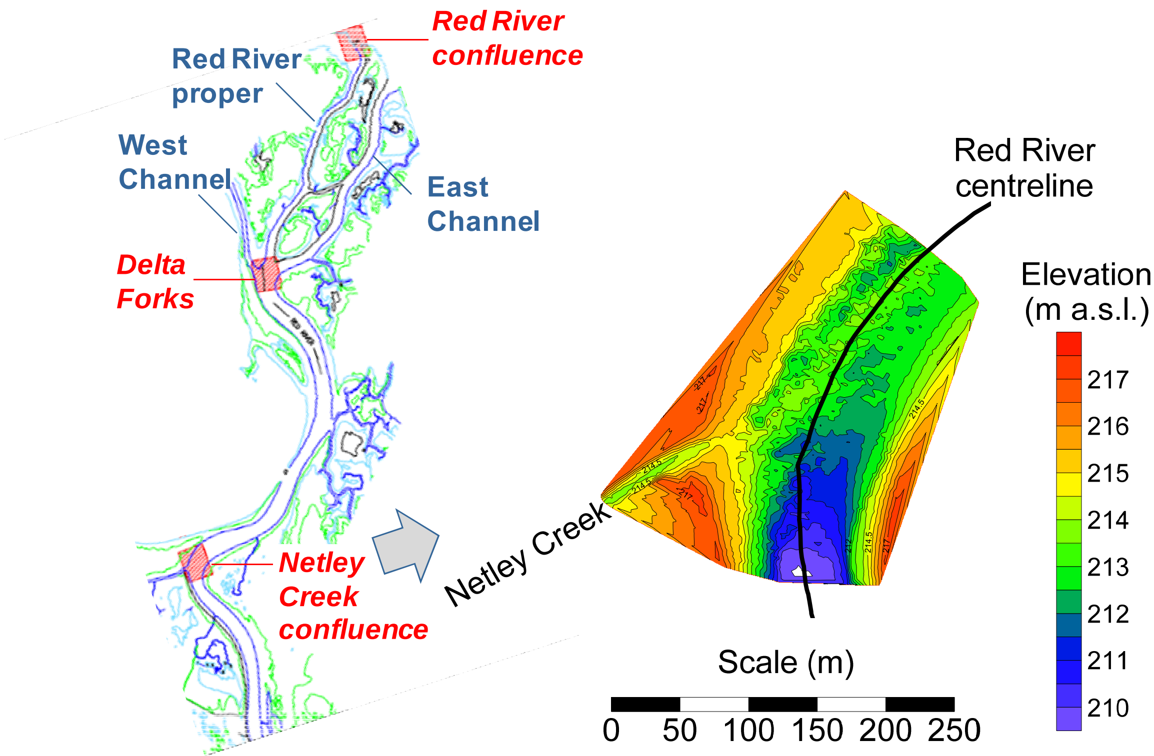


Figure 7: Location of soundings with a zoom and contour bathymetry at the Netley Creek confluence (data purchased from Public Works and Government Services Canada)

All gauge locations are indicated in **Error! Reference source not found.**. Discharges recorded at the Water Survey of Canada gauging station Red River at Selkirk located near the Selkirk Bridge were used for the upstream model boundary (see Figure 8). Discharge readings between 24. and 31. March are erratic due to ice run and jam activity. A polynomial function was fit to the hydrograph from which model input discharges were extracted. Water levels recorded at Gimli on Lake Winnipeg were used for the downstream model boundary. Water levels of several gauges along the river were used to re-construct the occurrence of ice jams and to calibrate the model. These include the gauges at Lockport, both upstream and downstream of the dam, Selkirk generating station and Breezy Point.

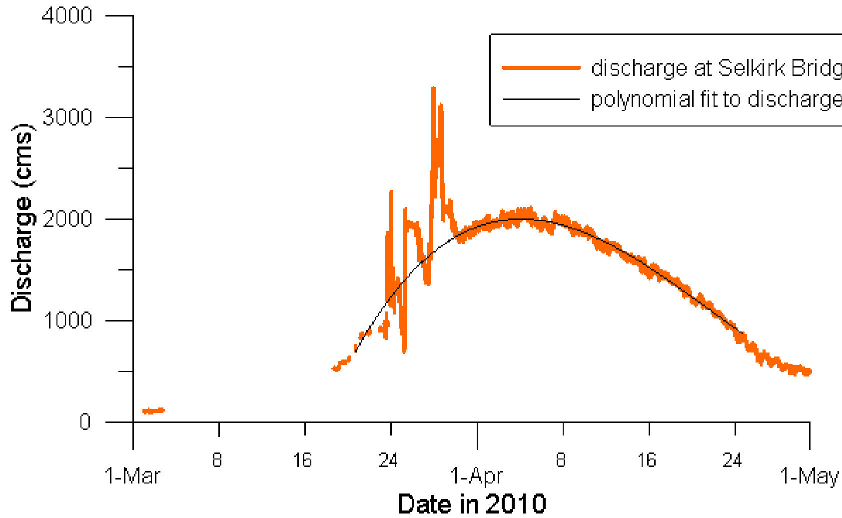


Figure 8: Discharge at Selkirk Bridge flow gauge.

Ice thicknesses were extracted from RADARSAT-2 satellite imagery (see Lindenschmidt et al., 2010, for a description of the methodology). A longitudinal profile of the average ice thicknesses of the Lower Red River is provided in Figure 9.

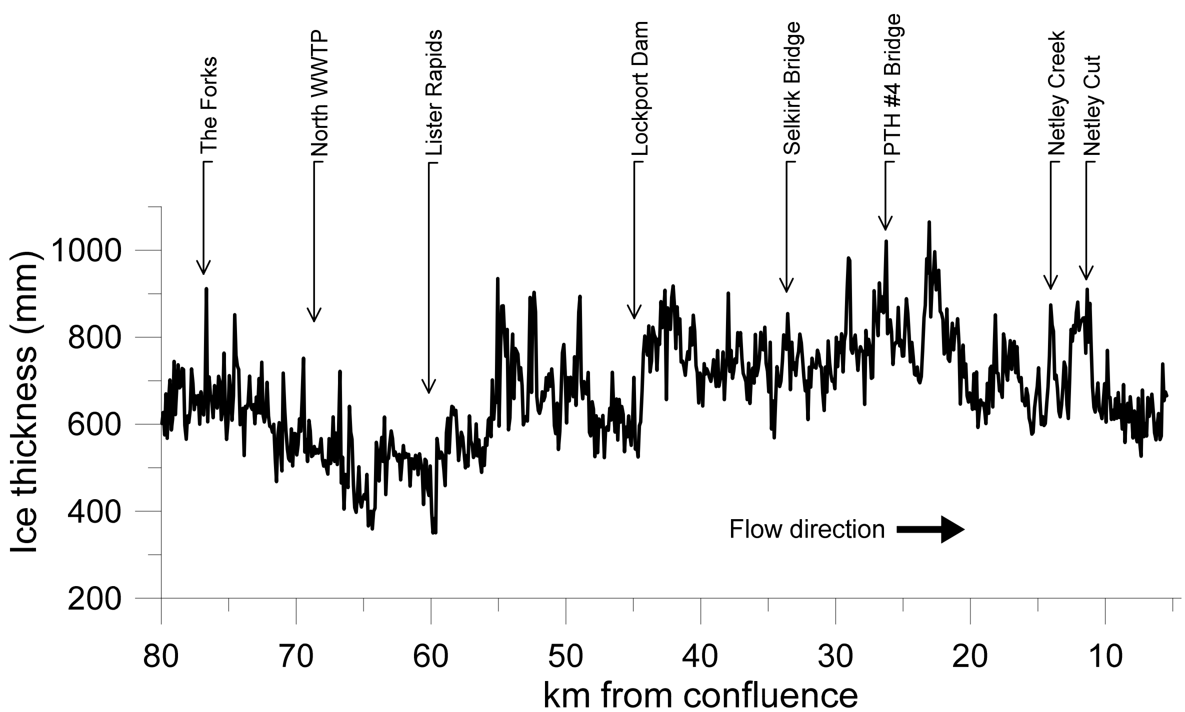


Figure 9: Longitudinal profile of average ice thicknesses of the Lower Red River between The Forks in Winnipeg and just upstream of the river's confluence at Lake Winnipeg.

9.7 RESULTS AND DISCUSSION

Ice jam at Selkirk

Figure 10 shows, for a discharge of 1000 cms, the longitudinal water level profiles along the modeled river stretch for three cases: open water conditions, an ice cover only and an ice cover with an ice jam at Selkirk. The model was first run under open water conditions without any ice on the river stretch, until a steady state was achieved (blue line). An ice cover was then inserted in the model during the simulation. The model was then allowed to continue to run until a second steady state condition was attained resulting in an increased water level profile due to the backwater effects caused by the flow under ice (red line). A flow of ice was then inserted that lodged at the ice cover front and formed an ice jam. The volume of ice corresponds to the amount of ice that broke up between Lockport and Selkirk. The simulation was allowed to persist until another steady state was achieved. The resulting profile of the backwater levels (black line) and the thicknesses of the ice cover and ice jam (black infills) are included in the figure. Notice that the ice jam and its backwater effects are well within the banks of the river.

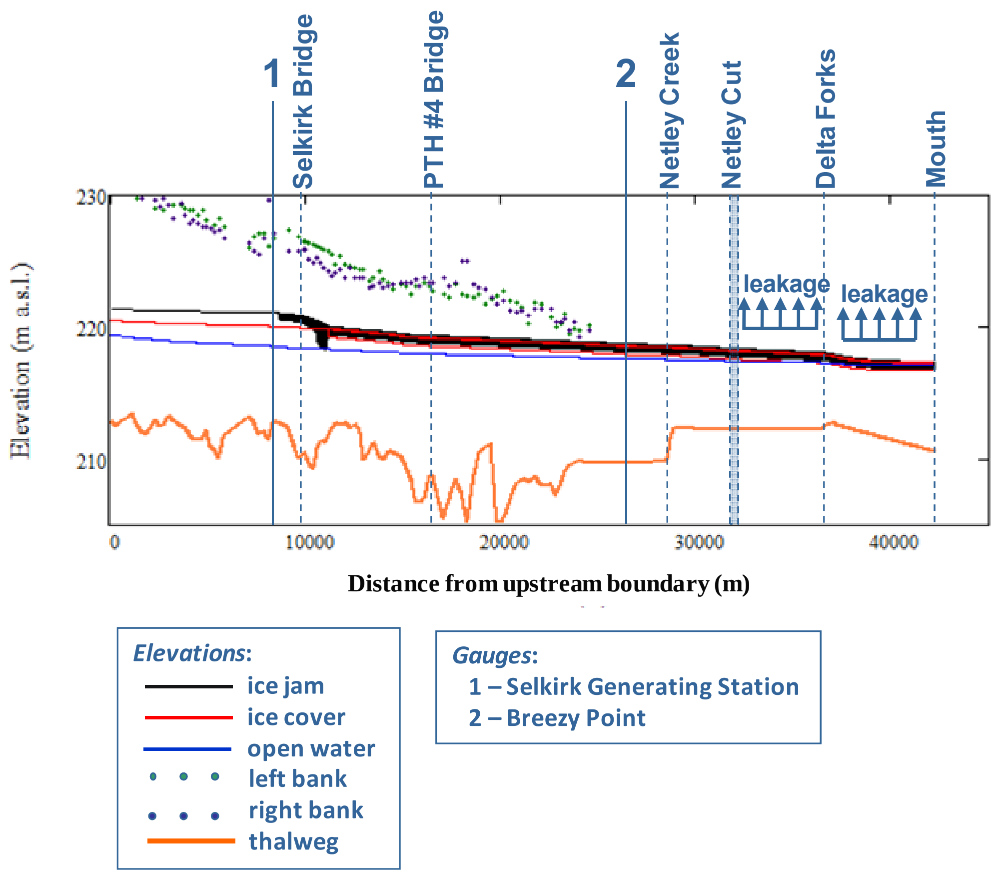


Figure 10: Simulated longitudinal profiles for open water conditions, ice cover and ice cover with ice jam at Sugar Island on 25. March 2010.

The simulations were repeated for increasing discharges on successive days, and the simulation results are juxtaposed on corresponding hydrographs in Figure 11. There is very good agreement between model results and recorded water levels.

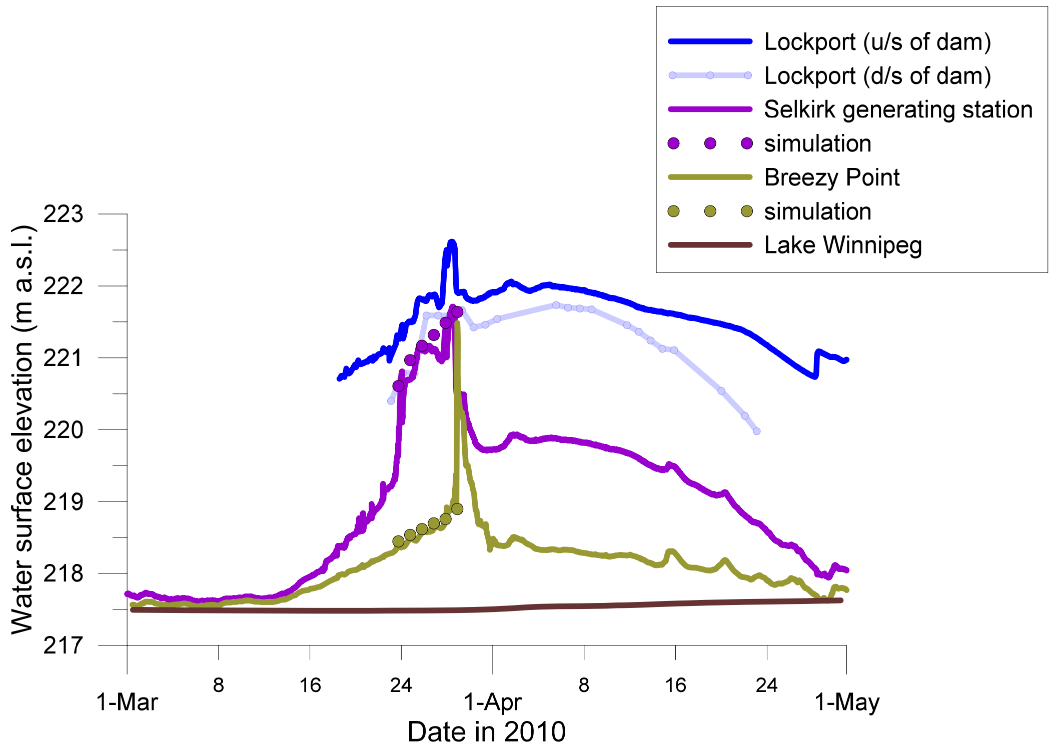


Figure 11: Recorded water levels during the spring flood of 2010. Simulation results pertain to the ice jam at Selkirk Park (data source: Water Survey of Canada and Manitoba Water Stewardship).

Water level readings from the Breezy Point gauge were drawn upon to determine model simulation outcome along the ice cover downstream from the ice jam. An initial simulation with the river modelled as a single channel to its mouth at Lake Winnipeg resulted in over-estimating the water levels with an intact ice cover at Breezy Point. This is due to the constricted cross-sections of the Red River proper in its delta in the Netley-Libau Marsh area. In actuality, the flow of the river fans out into several side channels between Netley Cut and Lake Winnipeg (see **Error! Reference source not found.**). Many of the shallow lakes in the marsh are also interconnected providing additional paths for water to flow from the Red River to Lake Winnipeg. Hence, for our one-dimensional model setup, a diffuse lateral abstraction was inserted between Netley Cut and Lake Winnipeg to represent the leakage of water away from the Red River proper. Calibration of the simulated water level to the gauge readings at Breezy Point resulted in a water leakage of 65% of the total inflow at the upper boundary. This is in line with leakages reported from other rivers in their deltas (e.g. Mackenzie River).

Ice jam at Netley

In the evening of the 28th of March 2010, the ice jam at the Selkirk Bridge released, as shown in Figure 12 by the abrupt drop in water level recorded at the Selkirk generating station. The jave caused a sharp water level rise at the Breezy point gauge approximately six hours later and the ice run was arrested at the Netley Cut where an ice jam was established until mid-day of 29. March 2010.

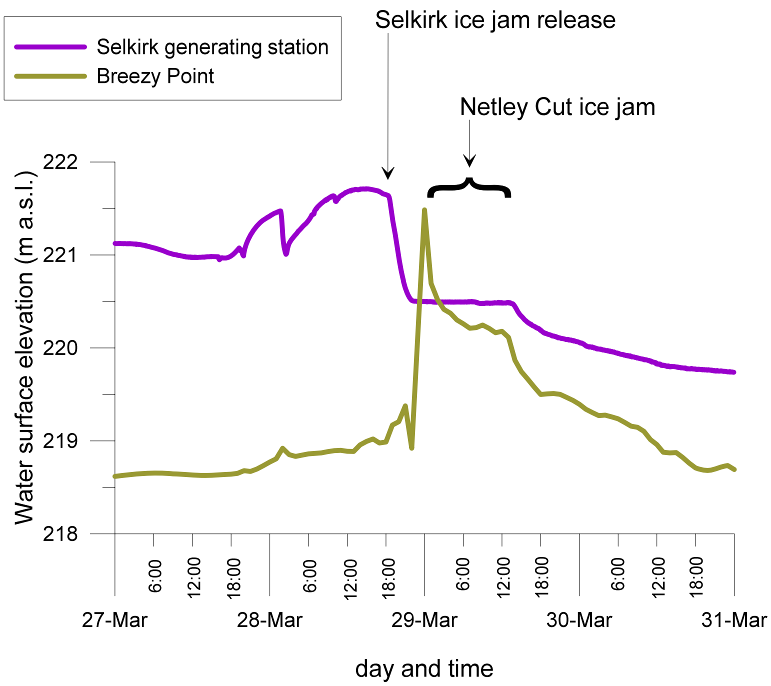


Figure 12: Water levels recorded at Selkirk generating station and the Breezy Point gauge during the ice jam release at Selkirk Bridge and the establishment of another ice jam at Netley Cut.

The model calibrated for the Selkirk ice jam was used as a basis for the model of the Netley Cut ice jam. The flow upstream of the jam release was approximately 1750 (see Figure 8), which was inserted as the upstream boundary flow. The water level at the downstream boundary at the Red River mouth at Lake Winnipeg remained unchanged. The volume of ice contributing to the Selkirk ice jam plus the ice that covered the river between the Selkirk Bridge and Netley Cut was input to the model.

Approximately 5 km upstream from the Netley Creek confluence, the widening of the the floodplain adjacent to the Red River begins. This floodplain extends downstream into the Netley-Libau Marsh area, which forms the delta of the Red River. The river's banks are very low and the surrounding topography is also very flat the low lying. Hence, this floodplain is very prone to ice jam flooding. The concept is depicted in Figure 13 (a) and (b) in which an increase in discharge will cause the ice cover to rise and break up. If the flow is a surge, which is the case when an upstream ice jam releases (e.g. at Selkirk), the in-channel storage capacity is quickly exhausted and the water spills into the surrounding floodplain or backs up into adjacent tributaries.

In the model, the width of the ice cover spans across the top of the entire cross-section provided as input. Only in-bank cross-sections are used since the ice cover width is contained within the river banks. The model assumes vertical walls extending upward from the leftmost and rightmost points of each cross-section to contain increased discharges, as shown in Figure 13 (c). The water levels become too high, though, for an upstream ice jam release since water spillage into the large floodplain is not simulated. The water level profiles for this particular case for the Netley Cut ice jam is shown in Figure 14 (top panel). The back water level just upstream of the ice jam front (black line) is almost 3 m higher than the backwater level cause by the downstream ice cover alone (red line) and ~2.5 m above the left and right bank elevations.

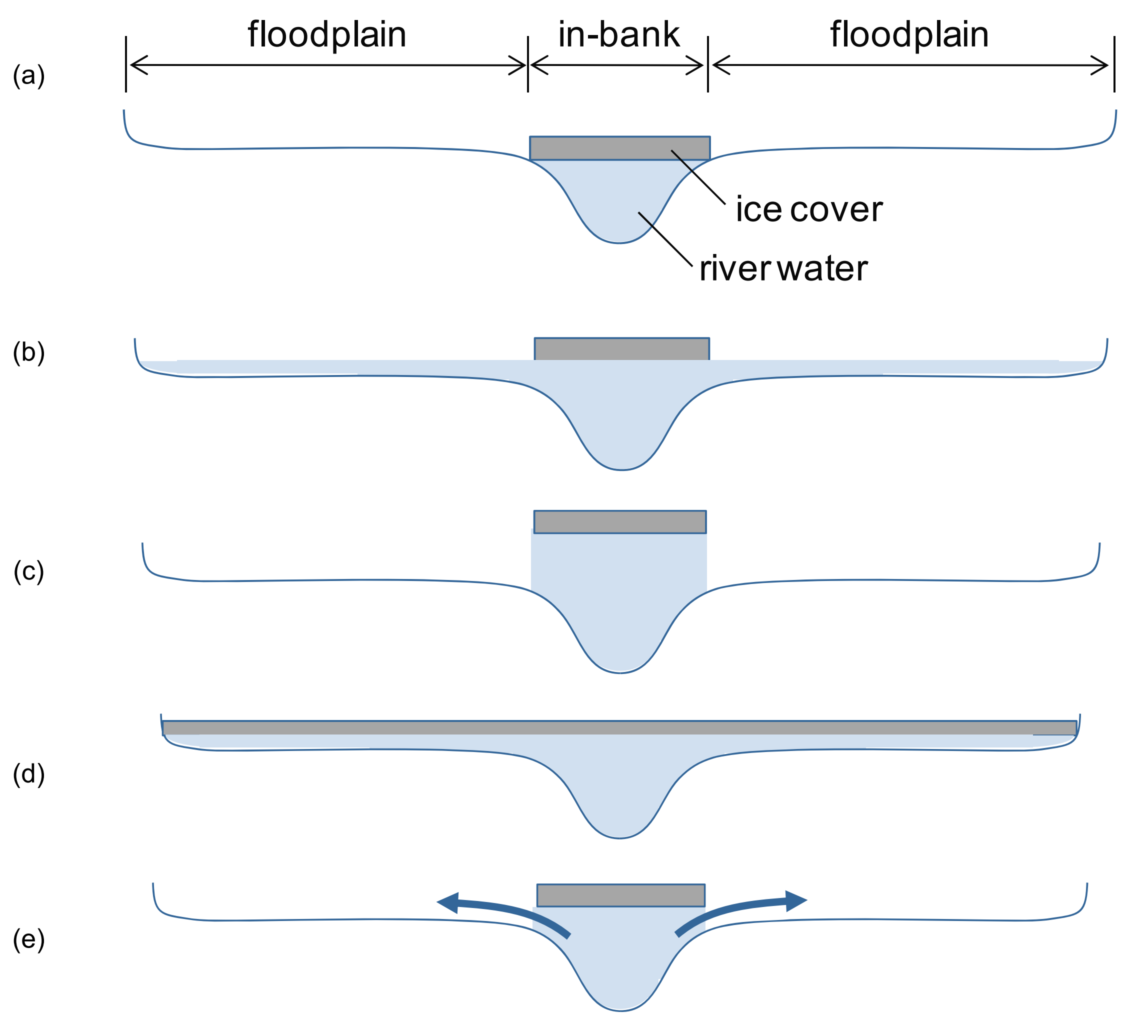


Figure 13: Conceptualization of an ice covered river flooding its floodplain: (a) river ice cover before breakup, (b) with increased flow, water spills into floodplain, (c) basic model setup with in-bank cross-sections causing backwater levels to rise too high, (d) alternative model setup including the floodplain in the cross-sections; ice cover width spans across entire floodplain which dampens the effect of ice jam backwater, (e) model adaptation with a diffuse abstract of main channel water equivalent to the volume of flood water in the floodplain.

Extending the cross-sections in the model to include the floodplain, however, would also extend the ice cover width to the edges of the floodplain, as shown in Figure 13 (d). This, too, veers from reality.

The approach taken here to adapt the model to simulate more realistic ice jam flooding in this river section within a large floodplain is to incorporate a diffuse abstraction of water from the river along the floodplain upstream of the ice jam, as depicted in Figure 13 (e). The volume of water removed from the main channel represents both flood water spillage into the floodplain and leakage of main channel water into side channel storage and diversions. Resulting water level profiles are shown in Figure 14 (bottom panel). The amount of water abstracted along the 5 km stretch upstream of the ice jam from was varied until the backwater level coincided or was just above the most downstream left and right bank elevations available. This resulted in a total abstraction equalling 1/3 of the upstream boundary flow. The leakage along the downstream ice covered portion of the river was reduced by half in order to avoid a drop in the downstream ice cover. This decrease in leakage is justified due to the reduced flow under the ice jam and the reduction in hydraulic head after the Selkirk jam release.

Another approach that may be implemented, and warrants future study, is inserting a tributary just upstream of the ice jam which has a storage capacity equivalent to the volume of water flooding the floodplain. The flow through the tributary would only be activated on the onset of jamming. Another approach described in the literature is taking "tributary flow reversals ... into account by trying different plausible outflow amounts and settling on the set that gave satisfactory ice-jam profiles and water-level predictions" (Beltaos 2003, p.3696). Sequential model applications in sub-reaches between reversed-flow tributaries have resulted in satisfactory reproductions of peak water levels (Beltaos 2003, p.3700).

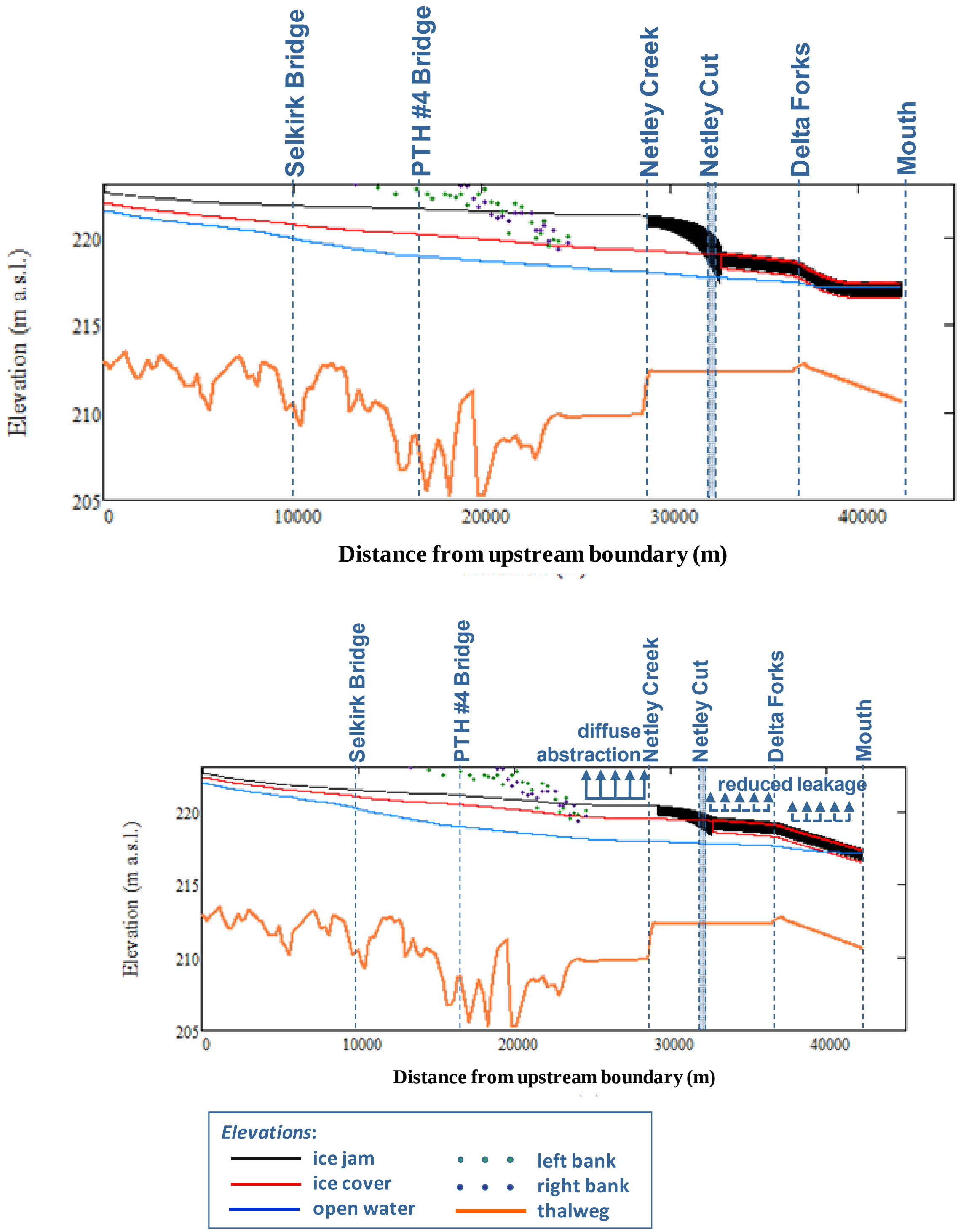


Figure 14: Simulated longitudinal profiles for open water conditions, ice cover and ice cover with ice jam at Netley Cut on 29. March 2010; *top panel*: simulations without incorporating flood water spillage into floodplain; *bottom panel*: simulations with a diffuse abstraction of water equivalent to the volume of flood water spilling into the floodplain.

Scenario: dredging of the lower reach

Based on the Netley Cut model setup described above, a scenario was carried to simulate the reduction in backwater staging due to dredging the river bed. Dredging would be feasible along the lowest reach of the studied river section between Netley Creek and the mouth of the Red River.

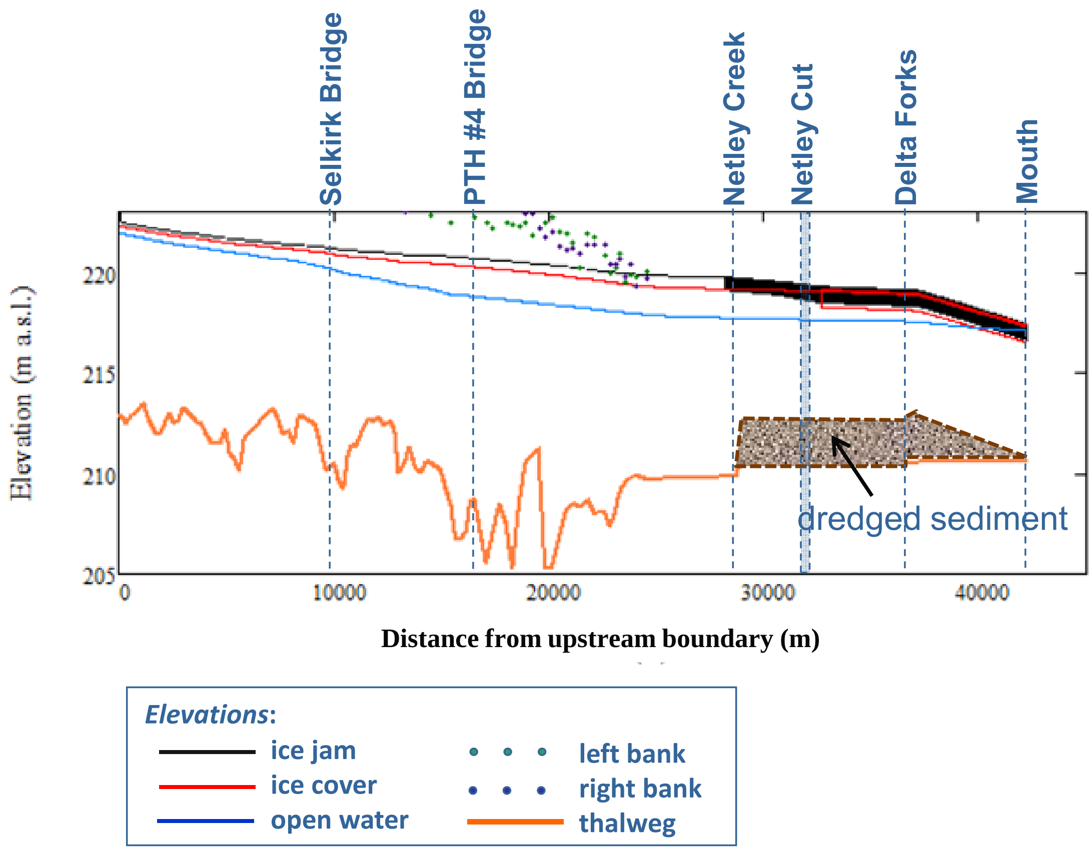
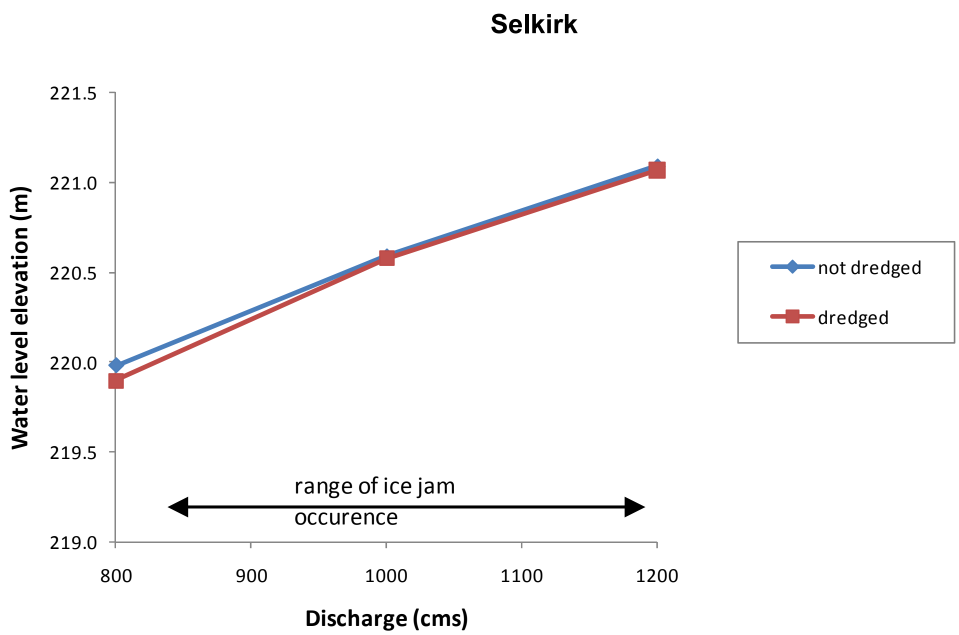


Figure 15: Simulated longitudinal profiles for open water conditions, ice cover and ice cover with ice jam for the dredging scenario.

The results of the scenario simulation are provided in Figure 16.The drop in ice jam backwater level due to dredging was calculated to be at

- Selkirk Bridge = 0.15 m
- Netley Creek = 0.6 m

In Figure 16 (bottom panel), the model results show that for discharges less than 1200 cms, a 3 metre removal of sediment would reduce the backwater level by 0.6 metres at Netley Creek, if an ice jam occurred at Netley Cut. This corresponds to a < 0.15 metre reduction at Selkirk, as indicated in Figure 16 (top panel). Additionally, the 0.6 metre reduction at Netley Creek occurs at lower discharges. At higher discharges (>1200 cms) when there is overbank spillage into the floodplain and Red River water backup into Netley Creek (a situation which is more common during ice jamming), then the reduction becomes less than 0.6 metres and would hypothetically approach a negligible amount at even higher discharges (approaching 2000 cms).



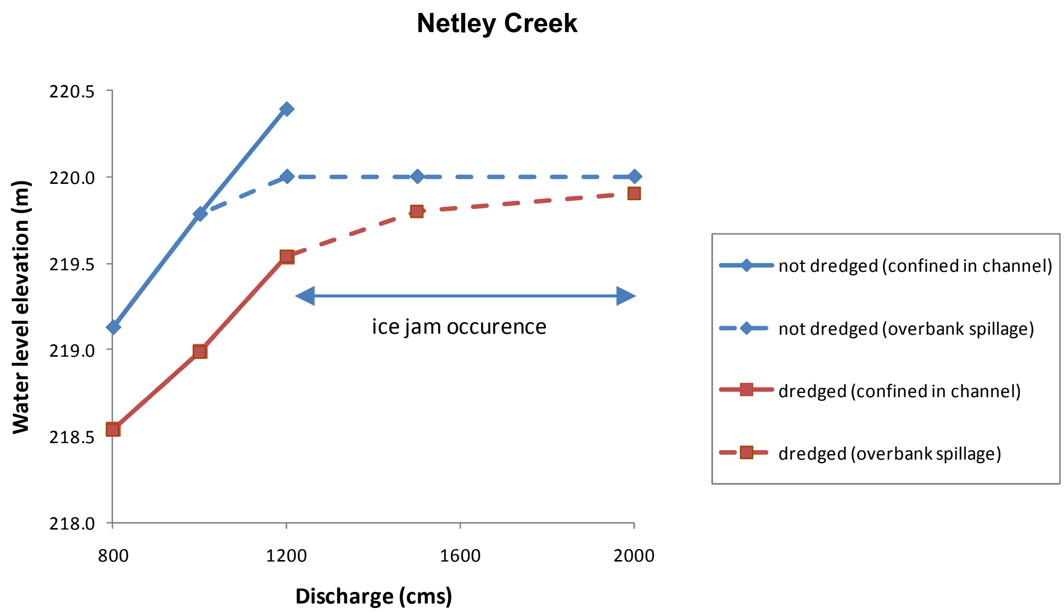


Figure 16: Backwater elevations at Selkirk (top panel) and Netley Creek (bottom panel) due to an ice jam at Netley Creek for different discharges and dredging scenarios. Note the difference in scale of the x-axes. The dashed lines for the Netley Creek graph are hypothetical extended based on expert opinions.

9.8 CONCLUSIONS

Rivers flowing through low-lying areas such as river delta poses particular challenges for ice jam modelling. One solution using main channel abstraction to represent bank overspill into a floodplain and leakage into side channels proved successful. Under-ice leakage from the Red River main channel was estimated through calibrated to be as high as 65% of the upstream discharge. Leakage decreased as the ice jam front progressed further downstream, thereby reducing the head differential between water levels upstream of the ice jam and the downstream ice-covered leakage area. Ice jam backwater spilling over low-lying river banks into the floodplain maintained this reduced head differential.

With regards to the scouring/deposition behaviour of the studied river section, more scouring occurs in the upstream stretch (between Lockport and the confluence of Netley Creek), whereas the downstream portion is a more sediment depositional section (between the Netley Creek confluence and Lake Winnipeg). Accreted sediment in the downstream reach is periodically washed out, especially in high flow years. The dredging scenario revealed that a small reduction in backwater levels (0.15 m) occurs at Selkirk with a dredged lower Red River reach.

All results were obtained from simulations with fixed values for the volume of incoming jamming ice (upper boundary condition) and with the same high water level of Lake Winnipeg (lower boundary condition). These values were adopted from the Selkirk ice jam calibration. A more encompassing assessment would require further simulations, in the framework of a Monte Carlo analysis, in which the following parameters are varied: (i) volume of jamming ice, (ii) dredging depth (1 m and 2 m) and (iii) Lake Winnipeg water levels.

Simulating ice jams in the Red River delta area with a one-dimensional modelling approach was successful and proves to be a useful tool to investigate various scenarios for ice jam flood management. Additional important questions for future work include:

(i) Where efforts of artificial cutting and breaking of the ice cover should be concentrated and avoided to reduce the hazard of ice jam flooding?

(ii) What impact would closing off the Netley Cut have on ice jam flooding risk?

9.9 REFERENCES

Acres (2004) Manitoba Floodway Expansion Authority - Floodway Expansion Project - Project Definition and Environmental Assessment. Preliminary engineering report Appendix L: Environmental baseline studies - water regime effects. Final report.

Beltaos, S., Pomerleau, R. and Halliday, R.A. (2000) Ice-jam effects on Red River flooding and possible mitigation methods. http://www.ijc.org/rel/pdf/icereport.pdf

Beltaos, S. (2003) Numerical modelling of ice-jam flooding on the Peace-Athabasca delta. *Hydrological Processes* **17**: 3685-3702.

Demuth MN, Hicks FE, Prowse TD, McKay K. 1996. A numerical modelling analysis of ice jam flooding on the Peace/Slave River, Peace–Athabasca delta. Peace–Athabasca Delta Technical Studies—P-ADJAM, Sub-component of Task F.2: Ice Studies. In: Peace–Athabasca Delta Technical Studies Appendices: I, Understanding the Ecosystem, Task Reports. National Hydrology Research Institute Contribution Series CS-96016, Saskatoon, Canada.

Farlinger and Westdal (2010) Red River Floodway public consultation on the rules of operation. Volume 1: Report on the public consultation. October 2010. http://www.gov.mb.ca/waterstewardship/floodway_rules_of_operation/documents/report_and_appendices.pdf

KGS (2010) RIVICE software – user manual. Draft revision A, January 2010.

Lacroix, M.P., Prowse, T.D., Bonsal, B.R., Duguay, C.R. and Ménard, P. (2005) River Ice Trends in Canada. Proceedings 13th Workshop on the Hydraulics of Ice Covered Rivers by CGU HS Committee on River Ice Processes and the Environment (CRIPE), Hanover, New Hampshire, September15-16, 2005.

Lindenschmidt, K.-E., Syrenne, G. & Harrison, R. (2010) Measuring Ice Thicknesses along the Red River in Canada using RADARSAT-2 Satellite Imagery. *Journal of Water Resource and Protection* 2(11): 923-933. http://www.scirp.org/journal/jwarp/

Lindenschmidt, K.-E., van der Sanden, J., Demski, A., Drouin, H. and Geldsetzer, T (2011) Characterising river ice along the Lower Red River using RADARSAT-2 imagery. CGU HS Committee on River Ice Processes and the Environment (CRIPE), 16th Workshop on River Ice, Winnipeg, Manitoba, September 18 – 22, 2011. http://cripe.civil.ualberta.ca/Downloads/16th_Workshop/Lindenschmidt-et-al-2011a.pdf

Michel, B. (1971) Winter Regime of Rivers and Lakes. U.S. Army Corp of Engineers. Cold Regions Research and Engineering Laboratory, Hanover, NH, Publ. No. AD 724121.

MB - Manitoba Government (2010) Red River Floodway operating rules public review report. 1. November 2010. http://www.gov.mb.ca/waterstewardship/floodway_rules_of_operation/documents/report_on_floodway_operating_rules_public_review_nov_1_2010.pdf

Topping, S., Warkentin, A. and Harris, J. (2008) Experience with dispersing ice jams in Manitoba. 19th IAHR International Symposium on Ice "Using new technologies to understand water-ice interaction". Vancouver, B.C., Canada, July 6 – 11, 2008. http://web2.clarkson.edu/projects/iahrice/19th%20IAHR%20Ice%20Symposium-Volume%201-revised.pdf

10. BUILDING AND DEBUGGING A RIVICE MODEL

10.1 DEVELOPMENT PHASE

Development of a RIVICE model can be a relatively simple process, once a user has gained familiarity with the program. Creation of a large or complex model is always challenging, even for the most experienced users. The most efficient way to build a large model is to progressively add small, logical components to the overall network, debugging each piece as it is added. This allows a user to isolate problem sources before they become intermixed with other new data. It also offers the likelihood of having fewer problems at one time.

The maximum size of any component should be related to its complexity and the level of knowledge and experience of the user.

10.2 PLANNING HINTS/TIPS

STABLE INITIAL CONDITIONS

An unbalanced set of initial conditions can cause very rapid failure near the beginning of a run. This occurs when the model computes high discharges while simulating a system that is in the process of balancing itself hydraulically. When the flows approach critical conditions, numerical instabilities develop and grow very rapidly, leading to premature termination of the run. Stable initial conditions are therefore especially important for steeper reaches and/or shallow depths.

At first, the user should establish simple initial conditions that avoid unstable situations, in order to establish a running model. Once a model can be started up without an immediate failure, it can be used to develop a set of stable initial conditions for some higher or lower set of discharges. To achieve this, the model should be started with any set of initial conditions that do not trigger immediate failure, then the discharges or water levels should be varied slowly to the target values, and finally these parameters should be held constant until a steady-state condition is achieved. This stable set of water levels and discharges can be entered to the model as starting values using one of the "warm start" techniques described under "Initial Conditions" in Section 3.1.

An alternative technique for developing an approximately balanced set of initial conditions is offered by the COORD2 program, but requires the user to estimate the water surface slope (variable SL(K) on Data 2-c). Sometimes the average riverbed slope can be used here, but in some cases the stable water surface slope can vary significantly from the average bed slope. Therefore, this technique may not always result in a stable set of initial conditions.

MODEL TESTING

It is necessary to provide boundary conditions, even for the simplest model. The use of constant discharge and water level boundary conditions provides a simple environment in which to test new components of a developing model. However, in some cases it is prudent to test a partly-constructed model over the full range of expected water levels and discharges to flush out any situations that will generate instabilities while the model is still small. Use of the BROWSE post-processor is recommended to review the simulated hydrographs at every mesh point to find any unusual or unstable conditions.

Sensitivity tests of other parameters are also recommended, to determine the effects of changing them. These tests give the user a feel for the responsiveness and behaviour of the modelled system. Although the calibration phase will involve strict comparisons with recorded data, some occasional checks against independent data are highly recommended, because early detection of large discrepancies will simplify the calibration later on, and might uncover some data errors.

SEQUENCE OF MODEL DEVELOPMENT

As the basic network of the model grows, additional boundary conditions and lateral inflows can be added. After the main network is complete, tributaries should be added. Then other reaches that will only be connected to the main network with QFC structures should be added. This should be followed by the addition of QFC structures, and any remaining boundary conditions, lateral inflows, control structures and special features such as sea dams.

Sometimes a problem can occur in a part of the model that ran "clean" earlier, and the source of the problem cannot be readily diagnosed. In this case, it is often useful to break out a small model of the suspected problem component and test it under the same conditions that it is subjected to in the large model. When isolated in small pieces the model behaviour can be much more easily understood, and problem sources usually reveal themselves. If the small part runs flawlessly in isolation, but fails when connected to the system, the user should start with the small model and add components one at a time working toward the large model until the problem appears. This procedure is usually sufficient to identify the cause.

TIME STEP

A formula for estimating an initial time step is provided in Section 3.1. Some types of problems (discussed below) can be eliminated by reducing the duration of the time step. For this reason a trial run with a very small time step is sometimes helpful. The longest time step for which the answers are not significantly affected is usually desirable for large models on computers with slow processors or limited disk space. If the model is very simple, or the run duration is very short, then there is little disadvantage to using smaller than necessary time steps.

REACH LENGTH AND MESH SPACING

Mesh space length also has a direct effect on the execution time and file storage requirements, and can be optimized using a technique similar to the time step optimization described above. Short mesh spaces allow the model to simulate curved water surface slopes with greater resolution. This is important when modelling situations where a rapid slope change occurs, such as near a sharp constriction or bed slope change. Sometimes it is wise to subdivide reaches at locations where significant energy slope changes are expected, and provide a smaller mesh space for the reach in the region of the greater slope variation. This is the rationale behind including transition reaches near sharp constrictions in the network.

10.3 DE-BUGGING HINTS - GENERAL HYDRAULIC CONDITIONS

This section contains a collection of hints and techniques. This is not an exhaustive list of all possible problems and errors and their solution procedures. Some advice for debugging has already been presented in the preceding sections, as it is difficult to separate the development of a model in stages with the debugging process. A "bug" is anything in the data or the program that causes premature program failures (crashes), numerical instabilities, errors or unrealistic simulation results.

ERROR MESSAGES

The most obvious bugs are those which result in crashes. These are sometimes accompanied with error messages and sometimes are not. The error messages, if they exist, may originate from the compiler as run-time error messages, or they may be from the ONE-D program. One very useful feature of the ONE-D program is that, even though it may fail prematurely, it saves the computed data up to the crash point in the simulation. This is vital for determining the cause of the problem.

Run-time error messages from the compiler will be written to the screen. These are very specific messages that state why the computer aborted the execution, such as "Invalid exponentiation", and give no explanation of what led to that condition, such as a water level being computed as falling below the bed of a channel.

Compiler error messages dealing with invalid numeric Data will state the filename (usually TAPE5.TXT) and the location (record number) in the file where the invalid data was first recognized. This indicates that data is either missing or unnecessary data exists in the file, or a counter, such as the number of cross sections in a reach, is not correctly set in the data.

RIVICE error messages are few in number. The program does little checking of data to ensure the data is within realistic limits, or whether it is consistent with other data when it should be. Therefore the onus is very much on the user to check all Data thoroughly, and to ensure that, when a particular piece of data is changed, all related necessary modifications will be done completely and precisely.

RIVICE error messages are usually found at the end of the TAPE6.TXT file, but they may on occasion be printed before the end of the file. The user should search the file for the word "ERROR" to find any earlier messages. On occasion, error messages are found in the TAPE61.TXT file.

When a run crashes with no useful hints contained in the compiler error message, and no ONE-D error message can be found in any of the output files, clues to the approximate source of the problem can sometimes be found by examining the point that the printing of information to the TAPE6.TXT file stopped.

NUMERICAL INSTABILITIES - HYDROGRAPH OSCILLATIONS

The majority of crashes are related to either Data errors or the development of numerical instabilities. These occur when the water levels and discharges begin to oscillate from one time step to the next, or from one mesh point to the next. If the oscillations grow with time, eventually the water level at some point in the model will be calculated to occur below the bed of the channel. When this happens, the program attempts exponentiation of a negative value, which is a fatal error condition that terminates the run.

In the Serpentine-Nicomekl study, the following techniques were developed to solve the problem:

- Model the culvert as a reach. This is the best solution as long as the flows never approach a Froude Number of 1.0 (critical depth).
- Reduce the time step. This is a good solution provided that execution times and disk storage capacities will still be satisfactory.
- Artificially increase the local dead storage near each end of the culvert. This may work, and in some cases may have no significant effect on the results, however the user must be aware of the implications that such changes will have on the results. Sensitivity tests may be warranted.

NUMERICAL INSTABILITIES - "SAW TOOTH PROFILES"

The "saw tooth" is a phenomenon normally associated with a hydraulic condition that is related to boundary conditions internal to the model. For instance, if the area of the upstream reach meeting the area of the downstream reach are not the same but the Manning "n" is, this will result in a location where the generation of the saw tooth commences. This can also occur is the area are the same but the manning coefficients in the adjacent reaches are not. The greater the difference the greater the oscillation will be.

The result of such situations, as described above, commonly occurs in RIVICE and ONE-D simulations is one where the water level is calculated to oscillate unreasonably from one mesh point to the next. Usually these profile oscillations occur within a reach and do not transmit to adjacent reaches connected by nodes. Typically, the amplitudes of the oscillations are very consistent from one mesh point to the next and the term "saw tooth profile" was coined to describe this phenomenon. The amplitude of all the oscillations grows or diminishes with time. Under severe profile oscillations, the run fails when the water level is computed to fall below the bed of the channel.

Once ice is involved and the roughness changes rather dramatically, such as at the leading edge, this characteristic of saw tooth profiles will result in the calculated discharges and water levels at the odd-numbered mesh points can be at times not too unreasonable. The wild estimates always occur on the even-numbered mesh points. Since the nodes are always odd-numbered mesh points (provided the user has conformed to the requirement of using only an odd number of mesh points in all reaches), the erroneous values are not passed on to the adjacent reaches through the nodes.

The cause of these profile oscillations is rooted in the finite-difference solution scheme. They can be normally found to occur under the following conditions:

- flow conditions approaching critical depth (i.e., high slopes or velocities, shallow depths, M-2 profiles)
- distorted hydraulic table data which represents impossible channel configurations
- sudden transitions of hydraulic parameters with elevation, such as at the underside of a bridge deck
- large changes in Manning's n values with elevation

These problems are more difficult to resolve than the hydrograph oscillations. In general, reducing time step duration and/or mesh space length is not usually sufficient to eliminate the saw tooth profile, although sometimes the mesh space length has an unpredictable effect on the amplitude of the oscillations. A few steps that can be taken to resolve the saw tooth profiles are listed below:

- Check all hydraulic tables for cross sections in the region of the saw tooth profile, especially near the point where the oscillations have the largest amplitude, and revise any unrealistic values.
- If flow conditions really are supercritical, then add a control structure in the system at the appropriate location.
- Add dead storage in the reach at the water levels where the problems are the worst (often this is at shallow flow depths). The user must be aware of the implications that such changes will have on the results. Sensitivity tests may be warranted.
- If the problem occurs near a high point in the profile, add an artificial slot in the bed of the channel throughout the problem area, provided its presence will not affect the key results for which the model is being developed. For example, if the problems only occur at low flows and the model is developed to find peak flood levels, a small artificial slot may not affect the final flood level estimates significantly.

Add transition reaches with small mesh spaces between cross sections with very different hydraulic parameters at the same elevation. In some instances, this has a beneficial effect; in others, however, this can actually make the problem worse.

In order to manage the development, calibration and final runs of a large RIVICE model, it is important to develop systems for organizing data files. As shown in Figure 1-1, a large number of files are involved in each of the modelling stages. Logical naming conventions and use of well-planned directory structures are fundamental requirements for a successful and efficient modelling project.

Documentation of runs, the changes made and the effects observed is also important in the development, calibration and sensitivity testing phases. The run log can take a variety of forms, and it inevitably becomes a valuable reference throughout a modelling project.

When testing a change, it is wise to only change one parameter at a time, then to observe the effects and document them. Multiple changes are only productive when the user has gained enough experience to accurately anticipate the effect that each change will have. If unexpected results occur, it is mandatory for the user to unravel why they are happening before proceeding. Sometimes a model predicts very reasonable, but completely unexpected, results. More often, unusual results are caused by modelling errors.

10.4 DEBUGGING HINTS – ICE CONDITIONS

The software has been developed to be as "bullet-proof" as practical within the time available for development and for the foreseeable common errors that might develop. However, there are common faults that should be avoided, and are listed and described below. The intent is that this list will be expanded as further understanding of the model capability develops.

1. High velocities under the ice cover, greater than about 1.5 m/s will have a significant potential to cause instabilities in the hydraulic calculations. Caution should be used in evaluating any results of RIVICE, but in particular if the velocities are high.
2. Velocity excessive at the location where an ice bridge is prescribed. This will almost certainly cause errors and program difficulties. The user should be careful that when an ice bridge is prescribed that it is capable of forming and remaining in place under the conditions that prevail.
3. The boundary condition at the downstream end should be sufficient to allow the ice cover to progress without causing the release of ice under the ice cover and out the downstream end of the study reach. Once this occurs, further progress of the ice cover will be impossible unless the inflow decreases.

There are several practical suggestions that should be considered by users:

1. The most important foundation for a good representation of river ice conditions is a set of prototype observations with water levels, ice cover progression rates, etc.
2. The user must understand the dominant ice processes that drive the ice formation that is being represented. For example, is the ice formation driven by ice deposition in a hanging dam? If so the user must pay special attention to selecting the critical velocity for deposition. Given the wide range in this value from site to site, a sensitivity study would be appropriate.
3. If shoving of the ice cover is a dominant process, the user must pay special attention to the interactive nature of Manning n-value, ice strength parameters and observed water surface profiles.
4. "Pilot runs" can be attempted to try to understand the dominant ice processes, if they are not obvious from the river observations available. After these initial trials, the user can then focus on the refinement of the appropriate parameters. Sensitivity analyses are recommended to be used extensively.

11. EXAMPLE SIMULATIONS OF ICE CONDITIONS

11.1 SIMPLE HYPOTHETICAL CHANNEL

This was a case used by the Committee on River Ice Processes and the Environment (CRIPE) in 2001 and 2002 for their first test of a variety of proprietary and non-proprietary programs that compute profiles of river ice jams. It is described in a CRIPE River Ice Workshop paper (Carson et al, 2001).

The key parameters are as follows:

- Constant channel width of 800 m.
- Bed slope 1 per 1 000.
- Reach length under study – 30 km.
- Rectangular cross sectional shape, infinitely high vertical banks.
- Constant water inflow of 3 000 m3/s.
- Constant inflow of ice 12 m3/s.
- Controlled downstream water level with 18 m of depth at the downstream end of the channel.
- Bed roughness defined by Manning 0.027.
- An additional thermal ice cover extending from the downstream end of the reach to a location 10 km upstream, with a fixed thickness of 0.6 m (and assumed to be rigid and unbreakable).
- Ice under surface roughness defined by Manning's n-value of 0.08.
- Leading edge stability based on a user defined thickness of 0.6 m (constant throughout the study reach).

Figure 11-1 shows the results of the five numerical models that were applied. There were some differences in specific elevations and ice thicknesses, but in general the solution of each was remarkably consistent with all others. Figure 11-2 shows the results of RIVICE overlain on the results of the other models

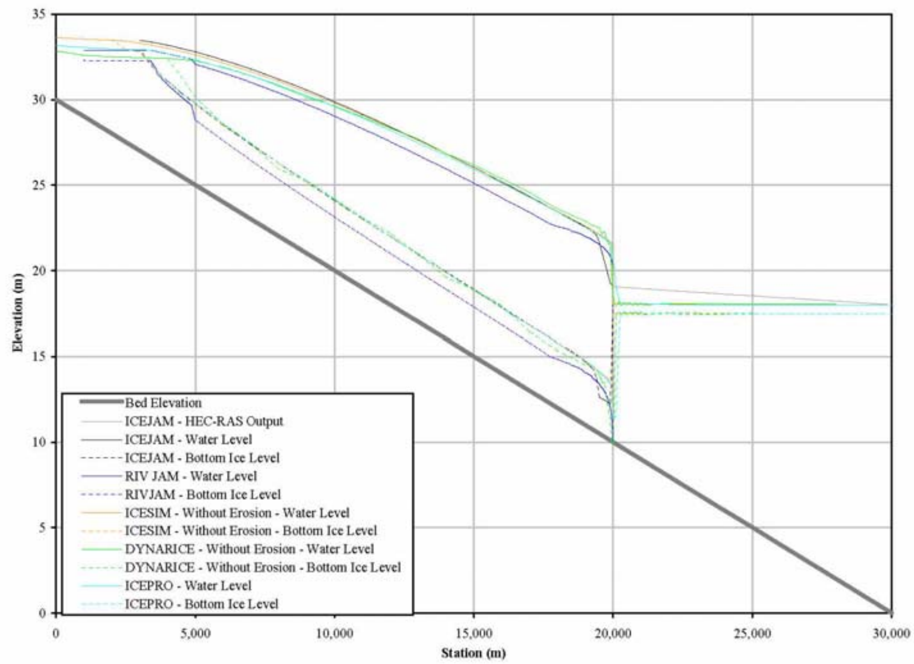


Figure 11-1

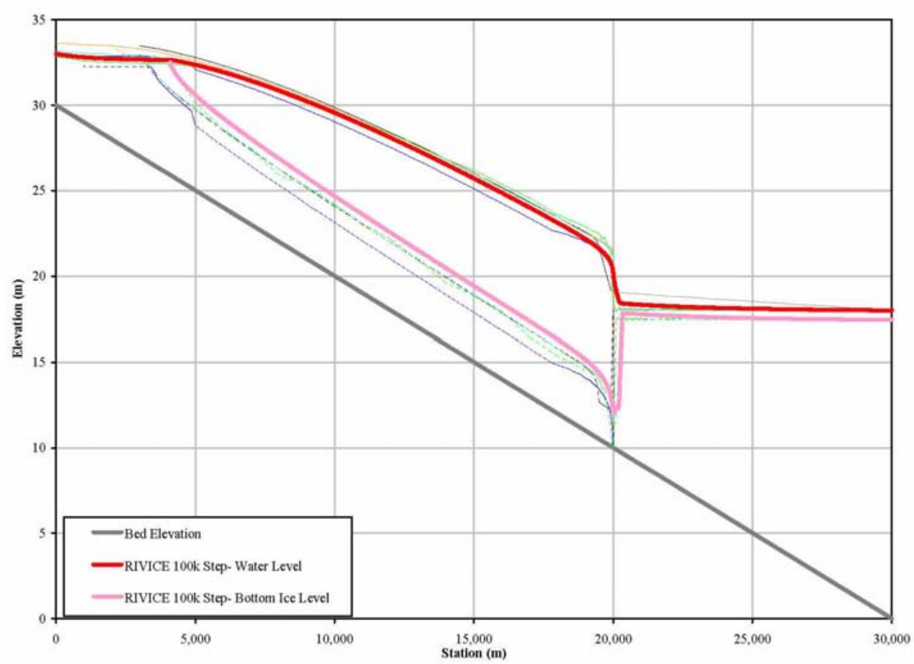


Figure 11-2

11.2 THAMES RIVER ICE JAM

This case study was based on an ice jam that occurred in the Thames River in south-western Ontario in 1986. A team directed by Dr. Spyros Beltaos of Environment Canada made measurements of the peak water surface profile. Post-event observations of the thickness of the ice cover remnants suspended on the riverbanks allowed for estimation of the thickness of the ice jam. The river channel was surveyed and over 20 river cross sections were available for analysis. Figure 11-3 shows the results of the simulations with the best combination of input parameters

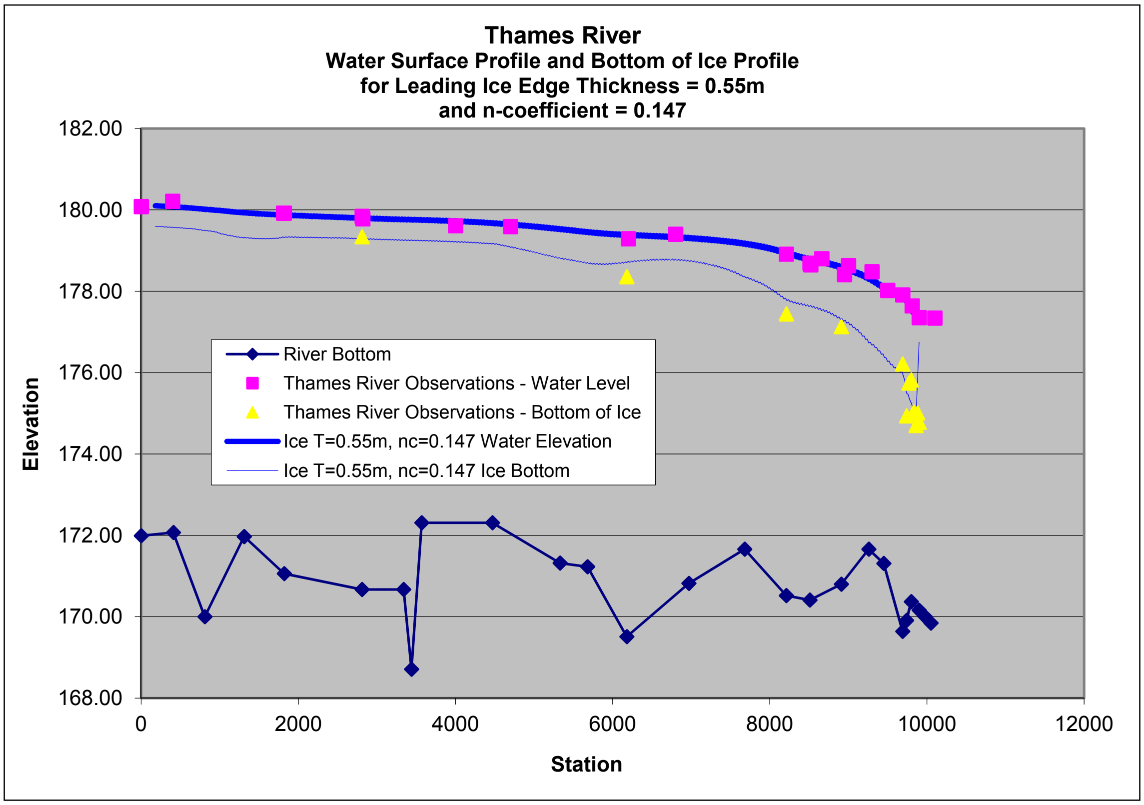


Figure 11-3 – RIVICE Simulated of Dataset for Thames River

The input Datasets for both sample cases is contained on the CD at the back of this document, along with the RIVICE code.

12. MODIFICATIONS BY USER

12.1 GUIDELINES

The development of RIVICE provided a variety of user-selected options, both in terms of methodology as well as selection of parameters and constants. However, as further flexibility, the program is divided into separated subroutines that could, if desired by the user, be replaced by alternative coding using the methodology desired by the user. The user is encouraged to review the list of subroutines and their functions in Section 2.1.

The key point in development of alternative subroutines is that the general information that may be required in the new subroutines is available by specifying the common blocks that contain these parameters. These are in three files called:

- 1DVAR.CMN
- ICEVAR.CMN
- TEMP.CMN.TXT

All variables included in these common blocks are defined in Table 12.1, and listed in the order in which they appear in the common blocks.

The "deliverables" that are expected by RIVICE from each of the major ice subroutines are summarized and described in Table 12.1, along with the locations in the program coding where these subroutines are called. If the user wishes to experiment with alternative subroutines, it is important that these "deliverables" are made available to other parts of the program for each time step.

TABLE 12.1 – Common Variables

1DVAR

THICK – Ice thickness (m)

WB, WBT- Border ice width (total left plus right bank) (m)

WBR, WBRT – right border ice width (m)

WBL,WBLT – left border ice width (m)

CNB – Manning n-value of riverbed

CNC – Manning n-value – composite for riverbed and ice under surface

TLXICE – chainage for each cross section in the reach under study (m)

TLXdT – reduction in temperature (degrees C) at a cross section as computed by the Water Quality subroutines of RIVICE. If TLXdT<0 this indicates the potential for ice generation

TLXVOL – volume of water in the cross section and its characteristic length (m^3)

TLXTW – Effective top width computed by subroutine ICEGENER. It is the open water width at the existing water level, minus the border ice width, minus the width of slush ice that is being carried by the flow at this location (m)

TLXTMP – Air temperature at this cross section for this time step (degrees C)

TLXWT – water temperature at this cross section for this time step (degrees C)

TLXWIND – Wind speed at this cross section for this time step (m/s)

ICEVAR

BORDVOL – volume of border ice broken off during this time step at this location (m3)

COHESN – cohesion of ice cover at the ice/river bank interface (Pa)

CN –

DTT – time step length (seconds)

DEPOPT – Option for simulation of deposition of ice-in-transit

LEOPT – option for simulation of leading edge stability

NSEG, NSEGT – number of ice segments

NFRT1T, NFRT1 – cross section number for leading edge of ice segment

NICES, NICEST – ice segment number to which ice cover at this cross section belongs

NISEG – ice segment number being analysed

NTRL1T, NTRL1 – cross section number for trailing edge of ice segment

XFRZ1T, XFRZ1 – length of ice cover advancing upstream of leading edge number NFRT1T (see Appendix 3)

TLE1, TLE1T – thickness of ice within length of ice cover defined by XFRZ1T (see Appendix 3)

FORCEF – coefficient that is used to "turn off" ice force calculations in thermal ice downstream of a specified ice bridge (if FORCEF=0, no forces are transmitted through the thermal ice cover)

VEL, VELT – velocity at a cross section (m/s)

Z, ZT,ZTT – water levels at a cross section at this time step (m)

VDEP – maximum velocity at which ice will deposit under an ice cover (used if DEPOPT=1)

VERODE – minimum velocity at which an ice cover will erode (used if EROPT=1)

VFACTR – factor applied to flow velocity to determine velocity at which ice-in-transit moves downstream (default=1.0).

VOLSUB – volume of ice submerged at the leading edge of ice cover if juxtaposition is not possible (m3)

VTRN, VTRNT – ice volume in transit (m3)

VOLIN – ice volume reaching the leading edge during this time step (m3)

VOLOUT – ice volume escaping from under the training edge of an ice segment during a time step (m3)

TIMED – time (days)

IEVOL – ice evolution type (only 1 is allowed at this time)

POROSC – porosity of ice cover

ZZK1TAN – coefficient of ice strength (see Section 2 and Section 5)

ZZK2 – coefficient of ice strength (see Section 2 and Section 5)

NSNTOT – total number of cross sections in the reach

THERMD – thermal ice thickness that is user-defined downstream of an ice bridge (optional) (m)

FRONTTHICK – thickness of leading edge of juxtaposed ice cover if LEOPT=3 (m)

ISTOP – User – defined cross section number at which simulation will stop when leading edge reaches this location

THICKMAX – record of the maximum ice thickness within an ice segment (for intermediate output of ice information) (m)

NMAXTHICK – cross-section number at which the maximum ice thickness occurs within this ice segment

HWI – heat transfer coefficient water to ice (if ice cover melting is to be simulated)

RLOCBRG – ice bridge location (as a cross section number)

NBRGSW – number of ice bridge switches that will define the initiation of ice segments

DAYSBR – day at which ice bridge switch takes effect (number of days since start of simulation)

BRIDTH – ice bridge thickness (m)

FTRLIM – maximum tractive force, above which, erosion of the ice under surface will occur, if the user has specified EROPT= 3 (in Pa)

EROPT – option to assess the potential for ice cover erosion

FRMAX – maximum user- defined densimetric Froude number (see Section 2); if exceeded and DEPOPT=3, no deposition of ice-in-transit can occur

DIAICE – average diameter of ice particles to be used in a Meyer Peter analogy to represent under ice transport rates (m); this is used when DEPOPT=2

POROSFS – Porosity of ice pans in the open water where ice generation is occurring

ICEGENMETHOD – method selected by the user to estimate heat loss from open water (either 1 or 2)

USICEVOL – user supplied volume of ice incoming to upstream end of reach for each time step (m3 per time step)

HLC – If ICEMETHOD=2, the user must define a heat loss coefficient from water surface to air (see Section 2) (in watts per square metre per degree Celsius)

HLOSS – heat loss from open water within the internal calculations in Subroutine ICEGENER

USICE – internal variable to represent ice volume from upstream (m3)

IXV – temporary cross section counter

ICEG – ice generated at a cross section during time step (m3)

VELOC – flow velocity (m/s)

OPENW – open width at a cross section (m)

SLUSHT – user-specified thickness of slush ice pans in the river (m)

MELTOPT – user defined option for addressing potential reduction in ice thickness due to heat content of incoming water (see Section 2)

MELTSTART – time step number at which melt calculations are begun

ICEVOLPREV – volume as slush ice being transported at each cross section at the end of the previous time step (m3)

ICEVOLCUR – volume as slush ice being transported at each cross section at the end of the current time step (m3)

VOLNFRT1 – Volume of ice reaching the leading edge of the ice segment during this time step (m3)

ICEnOPT – Option number for estimation of Manning n-value for ice cover (1 of 3 see Section 2)

BELTCOEF – coefficient for Beltaos method of estimating Manning n-value of ice cover

VICEnKGS8M – Manning n-value specified by user for an 8-metre thickness of ice. This is applied in the KGS method (see Section 2) and n-values are prorated in accordance to this value

VICEnUSER – Manning n-value for ice cover specified by user if ICEnOPT=3

TPWDTH – top width of channel at this cross section at current water level (ZTT) (in metres)

IBORD – user selected option number for estimating border ice growth

IBSTART -

DDAYS – degree days of freezing (Degrees C * days)

BORDCOEF1 – coefficient for calculation of border ice

BORDCOEF2 – coefficient for calculation of border ice

BORDSTRT – Starting water level at each cross section for formation of border ice(m)

BORDUPBRK – limit selected by user to cause breakup of border ice if water level rise exceeds this amount since start of border ice growth

BORDOWNBRK – limit selected by user to cause breakup of border ice if water level subsidence exceeds this amount since start of border ice growth

BRDTHK – border ice thickness at each cross section (m)

DAYBORDSTART – starting day for border ice growth (days)

DISTRIGINC – increment in ice front advancement that triggers intermediate results

TIMETRIGINC – increment in elapsed time steps that triggers intermediate results

SECTIONINC – increment between sections for which intermediate output is requested

IPRTYPE – type of intermediate results

DISTRIGGER – specific location (cross section number) when leading edge triggers intermediate output

TIMETRIGGER – specific timestep at which intermediate output is triggered

NTOTHER – number of specific time steps at which intermediate output is requested by the user

NDOTHER – number of specific ice front locations (cross section numbers) at which intermediate output is requested.

NEVER – internal memory of cross sections for which intermediate output has been triggered

IPRTRIGGER -

FTHRUST – for output purposes, calculated hydraulic thrust on the ice cover at the leading edge (kN)

FDRAGT – calculated hydraulic drag force exerted over the length at this cross section (kN)

FWEIGHT – calculated weight component at this cross section (kN)

FBANKT – calculated resistance of ice cover exerted against the river banks at this cross section (includes cohesion, if applicable)

FRESIDUAL – residual force in ice cover after applied forces are resolved and balanced against bank resistance (kN).

THMIN – minimum ice thickness at each cross section that is required to provide adequate strength to resist the residual force that exists (m)

LIMITOUT – number of cross sections for which detailed output is provided downstream of each leading edge (for detailed output option only).

Table 12.1 – "Deliverable" Information that must be Generated by the Primary Subroutines

Note: Only those subroutines that are likely to be candidates for user-defined modifications are listed here.

BORDICE

WBT – total width of border ice (m)

WBLT – width of border ice on left side of channel

WBRT – width of border ice on right side of channel

BORDICEBREAK

BORDVOL – volume of broken border ice at each cross section to be added to ice generated in the open water during this time step (m3)

BORDSTRT – water level at any cross section where the border ice has broken off during this time step; this forms the indicator for future breakup of border ice if water levels again exceed the allowed limits of fluctuation

DEFINEROUGH

CN – Composite Manning n-value for ice covered channel at each cross section

CNC – Manning n-value for ice cover at each cross section

ICECEA

NFRT1T – cross section number for leading edge for each ice segment

XFRZ1T – length of ice cover in front of leading edge (see definition sketch 3-1 in Appendix 3)

TLE1T – Thickness of leading edge of ice length XFRZ1T

NICES, NICEST – ice segment number to which the cross section belongs

VOLSUB – volume of ice submerged at leading edge if leading edge advancement is not possible (m3)

ICECEB

THICKT – Ice thickness at each cross section

VTRNT – Ice volume in transit at each ice-covered cross section (m3)

VOLOUT – Ice volume released under trailing edge of ice segment during this time step (m3)

ICECED

NFRT1T – leading edge location (cross section) for each ice segment

XFRZ1T – length of ice cover in front of leading edge (see definition sketch 3-1 in Appendix 3)

TLE1T – Thickness of leading edge of ice length XFRZ1T

NICES, NICEST – ice segment number to which the cross section belongs

THICKT – ice thickness at each cross section in the ice segment (m)

ICECI

NSEGT, NSEG – number of ice segments

THICKT – thickness of ice for cross sections within new ice segment

NICES, NICEST – ice segment number to which the cross section belongs

NFRT1T – leading edge location (cross section) for each ice segment

XFRZ1T – length of ice cover in front of leading edge (see definition sketch 3-1 in Appendix 3)

TLE1T – thickness of leading edge of ice length XFRZ1T

NTRL1T – cross-section number of trailing edge for ice segment

ICEGENER

ICEG – Volume of ice generated during this time step at each cross section that has no ice cover (m3)

- The primary graph that shows a profile of the interconnected reaches including thalweg, water surface, ice under surface, and top of ice for all ice segments as they evolve. The graph would be an on-screen demonstration of the simulation period and the evolution of the ice cover during that period.
- A supplementary graph that shows a plan view of the interconnected reaches, and shows channel width, border ice growth in open water zones, and ice covered areas where ice segments are in the process of being developed.

13. POTENTIAL FUTURE IMPROVEMENTS

13.1 GRAPHICAL ASSISTANCE FOR USER

The ability to graph the evolution of the ice cover during a simulation would be a great asset for a user to judge how well his selections of controlling parameters are performing. This capability was considered as a candidate for development during this completion phase of RIVICE, but there was not enough budget to permit the full and satisfactory completion (graphical capabilities were not included in the terms of reference for this project). Two graphs are visualized to be the deliverables from this module of the RIVICE programs:

- The primary graph that shows a profile of the interconnected reaches including thalweg, water surface, ice under surface, and top of ice for all ice segments as they evolve. The graph would be an on-screen demonstration of the simulation period and the evolution of the ice cover during that period.
- A supplementary graph that shows a plan view of the interconnected reaches, and shows channel width, border ice growth in open water zones, and ice covered areas where ice segments are in the process of being developed.

13.2 OTHER DESIRABLE IMPROVEMENTS

1. Temporal variance and evolution of Manning n-value of ice cover
2. Variable concentration of frazil ice in the water column, and the rate of rise to the surface.
3. User-defined ice thickness profile for a "hot start" of the simulation of on-going evolution of the ice cover.
4. GIS compatibility.
5. On-going improvements as the technology and understanding of river ice pressure improves.

14. REFERENCES

This list contains the entire list of references for the RIVICE development in the 1990's when the model options and capabilities were being evaluated and considered.

1. Acres Consulting Services Ltd. for Canadian Electrical Association, "Behaviour of the Ice Covers Subject to Large Daily Flow and Level Fluctuations", December, 1980, pp 82-84.
2. Abbott. M.B., "Method of Characteristics, Chap. 3 of Unsteady Flow in Open Channels", Mahmood & Yevjevich, Water Resources Pub., 1975, U.S.A.
3. Acres Int.; "Documentation on "ICESIM" - River Ice Engineering Simulation Model", 1988.
4. American Meteorological Society, "Workshop on Micrometeorology", edited by D.A. Haugen, 1973, 392 pages.
5. Ashton, G., "River and Lake Ice Engineering", Water Resources Publications, 1986, pp 290-295.
6. B.C. Hydro; "Description of B.C. Hydro Ice Simulation Model", May, 1983.
7. Beltaos S. and Dean A.M. (1981) - Field investigations of a hanging dam -Proceedings IAHR International Symposium on Ice - Quebec, July 27-31, 1981 pp. 345-355.
8. Beltaos, S., 1983, "River Ice Jams : Theory, Case Studies, and Applications", ASCE Journal of Hydraulic Engineering, Vol 109, No. 10.
9. Beltaos, S., August 1988, "Configuration and properties of a Breakup Jam", Canadian Journal of Civil Engineering, Vol. 15n, No. 4.
10. Beltaos, S. and Burrel, B.C. (1992), "Ice Breakup and Jamming in the Restigouche River", New Brunswick, IAHR Ice Simposium, Banff, 1992, pp. 436-449.
11. Cowley J.E. and Lavender S.T. (1975) - Convective heat transfer at an ice-water interface - Discussion by Dr. B. Michel - Proceedings of the Research Seminar on Thermal Regime of River Ice - Technical Memorandum No 114 -National Research Council Canada - October 1974 - pp. 60-76.
12. Danish Hydraulic Institute, "System 11 A Short Description" Horsholm, Denmark, Jan., 1986.
13. Danish Hydraulic Institute., "MIKE 11 A short Description" Horsholm, Denmark, July, 1987.
14. Environment Canada, "One-Dimensional Hydrodynamic Model: Data Preparation Programs, COORD1 & COORD2 Computer Manual", Water Modelling Section, Water Planning and Management Branch, (revised), June, 1988.
15. Environment Canada, "Documentation for the BREACH Routine Package to be used with the One-Dimensional Hydrodynamic Model", Water Planning and Management Branch, (revised), Feb., 1986.
16. Ferrick, M. and Mulherin, N.D. (1989), "Framework for Control of Dynamic Ice Breakup by River Regulation, U.S. Army CRREL Report 89-12, Hanover, USA.
17. Fread, D.L., "The NWS DAMBRK Model: Theoretical Background/User Documentation", Office of Hydrology, National Weather Service, NOAA, Silver Spring, Maryland, 1988.
18. Fread. D.L., "National Weather Service Operational Dynamic Wave Model," Office of Hydrology, National Weather Service, NOAA, Silver Spring, Md., April, 1978.
19. Garcia-Navarro, P. and Saviron, J., "McCormack's Method for the Numerical Simulation of

One-Dimensional Discontinuous Unsteady Open Channel Flow", IAHR, Vol. 30, 1992.

20. Gunaratnam, D.J. and Perkins, F.E., "Numerical Solution of Unsteady Flows in Open Channels", Report No. 127, Hydrodynamics Laboratory, Dept. of Civil Engineering, M.I.T., Cambridge, Massachusetts, July, 1970, 260 pages.
21. Harley, B.M., "Linear Routing in Uniform Open Channels", M. Eng. Sc. Thesis presented to the National University of Ireland, Dept. of Civil Engineering, Cork, Ireland, 1967.
22. Hausser R., Drouin M. and Parkinson F.E. (1986) - Thin ice sheet formation on warm water - Proceedings IAHR Symposium on Ice - Iowa City, August 18 - 22, 1986 - pp. 521-532.
23. Hausser, R., Pariset, E., "Formation and Evolution of Ice Covers on Rivers", Transactions of the Engineering Institute of Canada, Vol. 5, No. 1, 1961, pp 41-49.
24. Hausser R (1989) - Heat exchange at the ice / water interface of flowing water - Rivice progress report - Winnipeg, July 13-14, 1989.
25. Hausser, R. and Parkinson, F.E. (1991), "Heat Exchange at the Ice/Water Interface on Flowing Water". Sixth Int. Cold Regions Engineering Conference, Feb. 26-28, 1991, West Labanon, N.H.
26. Hausser, R., Pariset, E., and Gagnon, A., "formation of Ice Covers and Ice Jams in Rivers", ASCE Journal of Hydraulics Division, Vol. 92, No. HY6, November, 1966.
27. Hausser, R., Pariset, E., "Formation and Evolution of Ice Covers on Rivers", Transactions of the Engineering Institute of Canada, Vol. 5, No. 1, 1961, pp 41-49.
28. Huokuna, M.; "The Finnish River Ice Research Project" Proceedings of the 5th Workshop on the Hydraulics of River Ice, Winnipeg, August, 1988.
29. Hydro-Québec, Mathematical model RHIVER. Source program, Service hydraulique, 1984.
30. Krishnappan. B.G., "Users Manual: Unsteady Nonuniform, Mobile Boundary Flow Model," National Water Research Institute, Burlington, Ontario, Feb., 1981.
31. La Salle Hydraulics Laboratory; "RHIVER" Ice Model", Translation of key parts from documentation in French, 1988.
32. Lal. A.M.W., and Shen, H.T., "A Mathematical Model for River Ice Processes (RICE), Report No. 89-4, Clarkson Univ., Potsdam, N.Y., Feb., 1989.
33. Marcotte N. (1984) - Recent developments on mathematical modelling of winter thermal regime of rivers - Proceedings IAHR Ice Symposium - Hamburg, August 27-31, 1984 - pp. 315-327.
34. Marcotte N. (1981) - Régime thermique des glaces en rivière - Étude de cas Proceedings IAHR International Symposium on Ice, Quebec, July 27-31, 1981 pp. 412-422.
35. Marcotte N. (1986) - A simple mathematical model of moving sheet ice - Proceedings IAHR Symposium on Ice - Iowa City, August 18-22, 1986, pp 89-100.
36. Marcotte N. et Duong VL. (1973) - Le calcul de la température de l'eau des rivières - Journal of Hydrology, 18, pp. 273-287.
37. Marcotte, N. et Duong V.L., "Calcul de la température de l'eau. Analyse des observations dans un canal expérimental", Rapport T-73-2, Université de Moncton, Moncton, N.-B., 1973, 118 pages.
38. Marcotte, N. et Duong V.L., "Rayonnement solaire global et rayonnement solaire absorbé par l'eau, Ciel clair moyen, Latitude 42 N à 62 N", Rapport T-74-3, Université de Moncton, Moncton, N.-B., 1974, 59 pages.
39. Marcotte N. (1984) - Anchor Ice in Lachine Rapids - Results of observations and analysis -

Proceedings IAHR Ice Symposium - Hamburg, August 27-31, 1984 pp. 151-158.

40. Marcotte N. and Robert S. (1986) - Elementary mathematical modelling of anchor ice - Proceedings IAHR Symposium on Ice - Iowa City, August 18-22, 1986 - pp. 493-506.

41. Marcotte, N., "Projet Archipel. Analyse des observations de glaces durant l'hiver 1981-1982", Internal report No 08743-RA-83/02, Hydro-Québec, service Hydraulique, division Etudes spéciales et recherches, 1983, 41 pages.

42. Marcotte, N., "Projet Archipel. Etude de la glace de fond", Internal report No 08743-RA-83/74, Hydro-Québec, service Hydraulique, division Etudes spéciales et recherches, 1983, 128 pages.

43. Marcotte, N., "Projet Archipel. Modélisation et analyse des glaces", Internal report No 08743-RA-84/30, Hydro-Québec, service Hydraulique, division Etudes spéciales et recherches, 1984, 106 pages.

44. Marcotte, N., Hausser, R. and Marcoux A., "Projet Archipel. A New Concept of Ice Control in Open-Water Operation of Hydro Power Plant", Transactions, Canadian Electrical Association, Engineering and Operating Division, Volume 24, 1984/85.

45. Matousek, V., "Regularity of the Freezing-Up of the Water Surface and Heat Exchange Between Water Body and Water Body and Water Surface", IAHR Ice Symposium Proceedings, 1984, pp 187-200.

46. Matousek V. (1981) - A mathematical model of the discharge of frazil in rivers - Proceedings IAHR International Symposium on Ice - Quebec, July 28-31, 1981 - pp. 79-102.

47. Matousek V. (1984) - Types of ice run and conditions for their formation - Proceedings IAHR Ice Symposium - Hamburg, August 27-31, 1984 - pp. 315-327.

48. Michel, B., April 1971, "Winter Regime of Rivers and Lakes", U.S. Army Corps of Engineers Publication.

49. Michel, B., "Archipel. Résultats de la campagne de mesures des glaces, hiver 1982-83", Final report for Hydro-Québec, service Hydraulique, 1983.

50. Michel, B., "Ice Mechanics", Les Presses de l'Université Laval, Québec, 1978, pp. 280-281.

51. National Weather Service, "The NWS DAMBRK Model: Theoretical Background/User Documentation," Office of Hydrology, NOAA, Silver Spring, Md., 1988.

52. Newbury, R., "The Nelson River - A Study of Subarctic River Processes" - Phd Dissertation to Johns Hopkins University, 1968.

53. Osterkamp, T.E., Gilfilian, R.E., Gosink, J.P. and Benson, C.S., "Water Temperature Measurements in Turbulent Streams during Periods of Frazil-Ice Formation" - Proceedings of the Second Symposium on applied Glaciology , New Hampshire, USA, 23-27-August 1982.

54. Pariset, Hausser and Gagnon, 1966, "Formation of Ice Covers on Rivers", ASCE Journal of Hydraulics Division, Vol. 92, No. HY6.

55. Parkinson F.E. and Hausser R. (1986) - Thin sheet ice generation on cold water - Proceedings Fourth Workshop on Hydraulics of River Ice - Montreal, June 19-20, 1986.

56. Petryk, S., "Numerical Models", Chapter 5 of "Monograph on River Ice Jams" Draft, 1988.

57. Raphael, J.M., "Prediction of Temperature in Rivers and Reservoirs", Journal of the Power Division, Proceedings of the Amer. Soc. of Civil Engineers, Vol. 88, No. PO 2, July 1962.

58. Roache, J.P., "Computational Fluid Dynamics", Hermosa Publishers, 1982, pp. 46-47.

59. Report T-74-3, University of Moncton - Calculation of solar radiation at latitudes comprised between 42p and 62p north.

60. Roesner. L.A., Aldrich. J.A., and Dickson. R.E., "Storm Water Management Model, Version 4, Part B: EXTRAN Addendum," Camp, Dresser and McKee Inc., June, 1988.

61. Rousseau, Sauvé, Warren, Inc. et Hydro-Québec; "SIMGLACE" - Simulation des conditions des glaces en rivière" - Documentation du programme, 1983.

62. Samuels, P.G., "Modelling Open Channel Flow using Preissmann's Scheme", Proceedings of the 2nd International Conference on the Hydraulics of Floods and Flood Control, Cambridge, England, 24-26 September, 1985, pp. 91-102.

63. Saucet, J.-P. and Holder, G.K.; "Modèle mathématique RHIVER - application au lac St-Louis, 4th Workshop on hydraulics of River Ice - Vol. 1, Montreal, 1986.

64. Shen, T.H., and Chiang, L., "Simulation of Growth and Decay of River Ice Cover", ASCE, Journal of Hydraulic Engineering, Vol. 110, No. 7, July 1984.

65. SNC Inc., and CTA Consultants, "Study of Saint-Lawrence River, Cornwall-Varennes Section: Hydrodynamic Simulation Model", Report for the Environment Canada, Ministry of Transport, Quebec Water Board and Quebec Dept. of Natural Resources, March 31, 1973.

66. Tesaker, E., "Accumulation of Frazil Ice in an Intake Reservoir", Proceedings of IAHR Symposium on Ice Problems, Hanover, 1975.

67. Tesaker, E., "Accumulation of Frazil Ice in an Intake Reservoir", Proceedings of IAHR Symposium on Ice Problems, Hanover, 1975.

68. United States Environmental Protection Agency., "WASP4, A Hydrodynamic and Water Quality Model - Model Theory, User's Manual, and Programmer's Guide," Environmental Research Laboratory, Athens, Georgia, January, 1988.

69. Vreugdenhil, B.C., "Computational Hydraulics", Springer-Verlag, 1989.

70. Wasantha, M.A., and Shen, T.H., "Mathematical Model for River Ice Processes", ASCE, Journal of Hydraulic Engineering, Vol. 117, No. 7, July 1991.

71. Wood, E.F., Harley, B.M. and Perkins, F.E., "Operational Characteristics of a Numerical Solution for the Simulation of Open Channel Flow", Report No. 150, Ralph M. Parsons Laboratory for Water Resources and Hydrodynamics, Dept. of Civil Engineering, M.I.T., Cambridge, Massachusetts, June 1972, 186 pages.

APPENDIX 1
GLOSSARY OF TERMS

aboiteau	see floodbox
border ice	ice that advances laterally from the river bank if conditions permit
boundary	an interface between a modelled system and the world outside the model, usually at a location where known water level or discharge data exists or can be reasonably estimated.
boundary condition	water level or discharge specified by the user at a boundary of a model
breach	a break in a dyke or other embankment - not to be confused with breach structure, a term used in earlier ONE-D manuals that meant a quasi-dynamic flow control structure (see QFCS)
bug	an unintentional discrepancy or error in a program or its Data which causes unexpected results when the program is executed
calibration	the process of adjusting variables in a model to achieve a more reliable simulation of actual behaviour, which is monitored by comparing model results with measured data for the same time and location
composite n-value	combination of riverbed n-value and ice cover n-value in accordance with the Torok-Saboneev equation (see Section 2.2.8)
core area	in a ONE-D model, that portion of a cross sectional area of water which can convey discharge (as opposed to channel storage areas which hold water but do not convey flow)
core width	in a ONE-D model, that portion of the water surface top width which corresponds to the core area
crash	premature termination of a model simulation
floodbox	a structure with a one-way flow control device, such as a flap gate, which operates by water pressure differences, and is commonly configured as a flap gate at the downstream end of a culvert
frazil ice	ice that forms in open water and is carried with the flow
hydrodynamic	a term relating to unsteady flow conditions, or water levels and discharges which vary with time
hydrograph	a graph of continuous water levels or discharges versus time
hydrograph oscillation	an instability of water level and/or discharge which features alternating high and low values with time, usually from one time step to the next

ice segment	ice cover over a portion of the reach under study; has a leading edge at cross-section “NFRTIT” and a trailing edge at cross-section “NTRLIT”
ice-in-transit	ice being carried under the stationary ice cover
initial condition	water level or discharge specified by the user at every Data cross section in the model which represents the hydraulic situation at the beginning of the simulation
junction node	a node which represents the junction of two or more reaches
lateral inflow	an inflow or outflow (defined as a negative inflow) which enters or leaves a reach at a location other than a node, that is, at any mid-mesh point along a reach
leading edge	upstream end of ice segment, where a full ice cover exists at cross-section “NFRTIT”. Ice can be advancing into the next upstream cross-section.
mesh point	one of many evenly spaced points along a reach at which the water levels and discharges are calculated by the model
mesh space	the distance between adjacent mesh points
model	the combination of a program and its Data which represents a part of the real world
node	an endpoint of a single reach (see junction node and terminal node)
numerical instability	a condition in a model generated within the mathematical routines that features unrealistic, oscillating values with time and/or location - these often lead to crashes, especially when they diverge (grow with time)
pre-processing	the preliminary calculations and organization of Data of a model required to prepare it for execution of a simulation
post-processing	the sorting, organization and presentation of the results from a model simulation intended to facilitate communication with the user
profile	a graph of water level or discharge versus distance along a reach, or a series of consecutive reaches
reach	the basic element of a ONE-D model which is capable of conveying water, and represents a segment of a channel, lake, bridge waterway, culvert or floodplain storage cell
sawtooth profile	a profile along a reach that features oscillating water levels or discharges from one mesh point to the next, usually with a very regular pattern, and representing unrealistic results
shove	thickening of the stationary ice cover to resist hydraulic forces exerted
slush ice	agglomerations of frazil ice, usually in pans on the surface of the open water
terminal node	a node which does not occur at a junction of two or more reaches
thermal ice	ice formed by thermodynamic processes only, driven by heat loss from water
time step	the regular time increment for which the ONE-D program computes successive solutions of the entire model
trailing edge	downstream end of a stationary ice cover (defined by cross-section “NTRLIT”)

APPENDIX 2 THEORETICAL BACKGROUND AND COMPUTATIONAL METHODS FOR HYDRAULICS / HYDRODYNAMICS

A2.A THEORETICAL CONSIDERATIONS

A1. GOVERNING EQUATIONS AND ASSUMPTIONS[5]

The process of propagation of long waves in open channels is described by the Equations of Saint Venant. These equations represent the conservation of mass and momentum of flow of fluid in the channel. The fundamental assumptions that are made in the derivation of the Saint Venant Equations are:

1) The flow is assumed to be one dimensional, i.e. the flow in the channel can be well approximated with uniform velocity over each cross-section and the free surface is taken to be a horizontal line across the section. This implies that centrifugal effect due to channel curvature and Coriolis effect are negligible.
2) The pressure is assumed to be hydrostatic, i.e. the vertical acceleration is neglected and the density of the fluid is assumed to be homogeneous.
3) The effects of boundary friction and turbulence can be accounted for through the introduction of a resistance force which is described by the empirical "Manning" or "Darcy Weisbach" Friction Factor Equations.

Having made these assumptions, the conservation equations may be formulated by the "material" method or the "control volume" method. In the "material" method the flow characteristics are obtained by following the motion of a given mass of fluid through a small increment of time in the vicinity of the fixed section. In the "control volume" method the equations are derived by considering the fluxes of mass and momentum through a fixed control volume (see Figures A-1 and A-2).

Derivations of the Saint Venant Equations by the "material" method formulations were made by Harleman[6]. The equations in terms of average velocity v, and water surface elevation z, are given below (see Figure A-1 and Figure A-2 for notation).

[5] Daniel J. Gunaratnam and Frank E. Perkins, "Numerical Solution for Unsteady Flows in Open Channels", Hydrodynamics Laboratory, Report No. 127, July 1970, p.25-32.

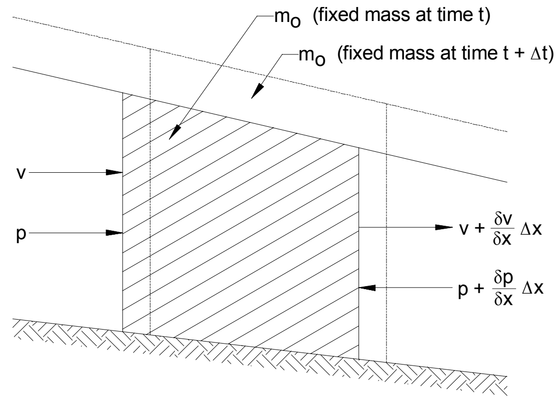


Figure A-1 Definition Sketch of Control Volume

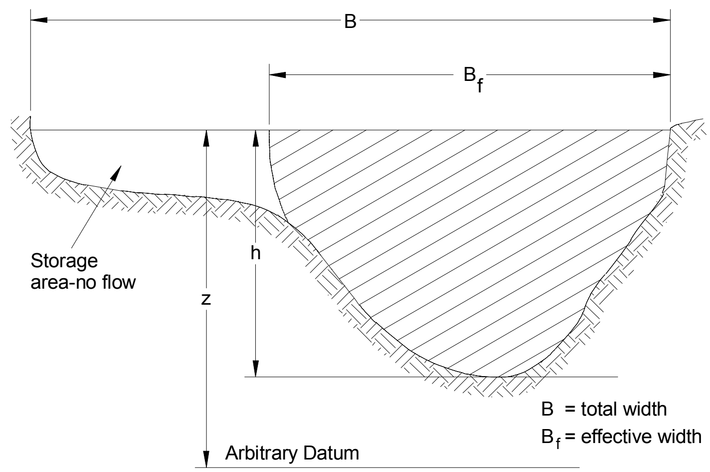


Figure A-2 Definition Sketch of Cross Section of Irregular Channels

CONTINUITY EQUATION

$$B \frac{\partial z}{\partial t} + Bv \frac{\partial z}{\partial x} + A \frac{\partial v}{\partial x} + v \frac{\partial A}{\partial x}\Big|_z = const. = q_l$$

1 2 3 4 5

[6] Harleman, D.R.F. and Lee, C.H., "The Computation of Tides and Currents in Estuaries and Canals", U.S. Army Corps of Engineers Report, Oct. 1967.

MOMENTUM EQUATION

$$\frac{\partial v}{\partial t} + v\frac{\partial v}{\partial x} + \frac{vq_l}{A} = g\,(S_0 - \frac{v}{C_z^2}\frac{|v|}{R}) - g\frac{\partial h}{\partial x}$$

6 7 8 9 10 11

The role of various terms are defined below:

CONTINUITY EQUATION

(1) Rate of rise term which gives the storage changes due to water surface elevation changes with time.
(2) Prism storage term (see Figure A-3) due to variation in velocity with space.
(3) Wedge storage term due to areal variations in velocity with space. (see Figure A-3)
(4) Wedge storage term due to areal variations in velocity with space. (see Figure A-3)
(5) Lateral inflow term which gives the net mass change spatially and temporally beyond the storage terms.

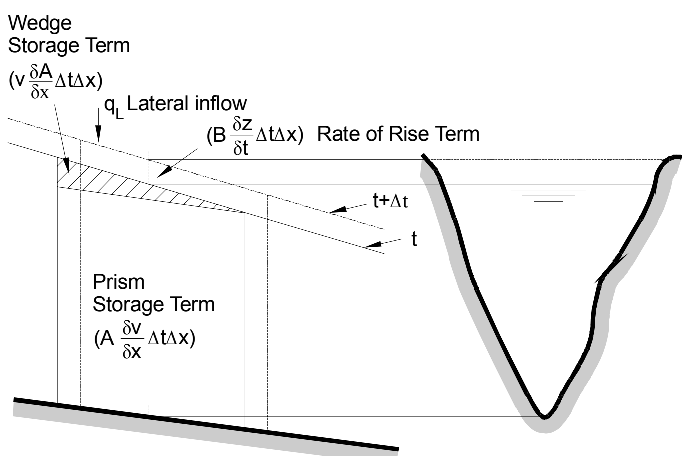


Figure A-3 Physical Representation of Terms in Continuity Equation

MOMENTUM EQUATION

(6) Acceleration due to time variation in flow
(7) Acceleration due to spatial variation in velocity
(8) Acceleration effects due to lateral inflow
(9) Gravity body force due to bed slope
(10) Frictional force effects
(11) Pressure force term

For a wide rectangular channel the equations take the special form:

CONTINUITY EQUATION

$$\frac{\partial h}{\partial t} + v\,\frac{\partial h}{\partial x} + h\,\frac{\partial v}{\partial x} = \frac{q_l}{B}$$

MOMENTUM EQUATION

$$\frac{\partial v}{\partial t} + v\,\frac{\partial v}{\partial x} + \frac{v\,q_l}{B\,h} = g\,(S_o - \frac{v}{C_z^2}\frac{|v|}{h}) - g\,\frac{\partial h}{\partial x}$$

It has been found that since the boundary conditions are generally specified in terms of discharge, Q, or water surface elevation, z, it is more useful to express the St. Venant equations in terms of Q and z as follows:

CONTINUITY EQUATION

$$B\,\frac{\partial z}{\partial t} + \frac{\partial Q}{\partial x} = q_l$$

MOMENTUM EQUATION

$$(\frac{\partial Q}{\partial t} + 2v\,\frac{\partial Q}{\partial x})\,\frac{1}{Ag} = -\,(1 - F^2)\frac{\partial z}{\partial x} + (\frac{1}{B}\,\frac{\partial A^z}{\partial x})\,F^2 - \frac{Q\,|Q|}{K^2}$$

Where: F = Froude Number

$$K = AC_Z\,\sqrt{R}$$

$$\frac{\partial A^z}{\partial A} = \frac{\partial A}{\partial x}\Big|_z = const.$$

The non-prismatic channel effects are shown in the momentum equation by the term, $\partial A^z/\partial x$, which is sometimes written in the alternative form,

$$\frac{\partial A^z}{\partial x} = B\,(S_0 + \frac{1}{B}\,\frac{\partial A^h}{\partial x})$$

A1.1 INCLUSION OF VERTICAL ACCELERATIONS TERMS:

In some special cases, the vertical acceleration terms which were neglected in the St. Venant Equations can be significant. In order to understand the nature of these special cases, a form of the governing equations including acceleration effects is presented. This form which was proposed by Keulegan[7] is restricted to the following simplified cases:

1) wide rectangular channel;
2) the variation with z of the x-component of local velocity, v, is negligible i.e. v = v (x,t);
3) lateral inflow and lateral velocity are neglected.

By virtue of these simplifications it is possible to focus attention solely on the effects of the vertical acceleration associated with rapid changes in the water surface elevation. A derivation of the equation is presented to Keulegan. For the present purposes it is sufficient to note that the resulting form of the governing equations is:

CONTINUITY EQUATION

$$\frac{\partial h}{\partial t} + v\frac{\partial h}{\partial x} + h\frac{\partial v}{\partial x} = 0$$

$$\frac{\partial v}{\partial t} + v\frac{\partial v}{\partial x} + \frac{hv^2}{3} + \left(\frac{\partial^3 h}{\partial x^3} + \frac{2}{v}\frac{\partial^3 h}{\partial x^2 \partial t} + \frac{1}{v^2}\frac{\partial^3 h}{\partial x \partial t^2}\right) =$$

$$= g\left(S_0 - \frac{v\,|v|}{C_z^2}h\right) - g\frac{\partial h}{\partial x}$$

:.......vertical acceleration term.......:

By comparing these results with equations (3) and (4), without the lateral inflow term, it can be seen that the continuity equation is unaffected while three additional vertical acceleration terms appear in the momentum equation. The vertical acceleration terms are only significant when the water surface curvatures are large. It can be shown through an order of magnitude analysis that for normal transient flow conditions (i.e. h/L much less than 1 where h is the flow depth and L the characteristic wave length of the transient) they are second order terms (see Keulegan). The effects of these acceleration terms under strong water surface curvatures of a steep front due to rapid transients is to disperse the front (i.e. to create a series of undulations behind the front)[8,9].

However, even if steep fronts do form, it is unnecessary from an engineering point of view to represent them in detail since they are small and damp out rapidly. Furthermore, by writing the St. Venant equations in conservation form, or by using some accurate, numerical procedures, the mean shape of these steep fronts can be predicted. Hence, it had been chosen to ignore the effects of the vertical acceleration terms.

A2.B THEORETICAL CONSIDERATIONS

B1. METHOD OF SUPERPOSITION

A basic technique in formulation of the network solution procedure is the method of superposition, which was proposed for channel network computations by Gunaratnam and Perkins[1]. It resembles the method of influence functions which is commonly used in structural analysis. The technique makes use of the fact that, at a given time step, the solutions for water surface eleva-

[7] Keulegan, G.H., and Patterson, G.W., "Effects of Turbulence and Channel Slope in Translation Waves", Journal of Research, National Bureau of Standards, Vol. 30, June 1943.

[8] Shonfield, J.C., "Distortion of Long Waves Equilibrium and Stability", U.G.G.I., A.I.H.S., 1951, Vol. 4, p.140-157.

[9] Peregine, D.H., "Long Waves on a Beach", Journal of Fluid Mechanics, Vol. 27, Part 4, p. 815-827, 1967.

tions, discharges, and concentrations as mesh points are given by sets of simultaneous linear equations. Because these equations are linear the principle of superposition will hold. The actual mechanics of executing the method of superposition are discussed in the following section; however, the basic notions are simple. In a given reach, the solution may be decoupled from the boundary mesh points by solving a set of equations for the interior mesh points under the assumption that solution values at the boundary mesh points are zero. This will be called a null solution. Using the same set of equations, influence factors due to unit solution values at the boundary mesh points are computed at the interior mesh points. There remains a set of equations for the boundary mesh points, which may be solved after the boundary conditions are included. Solution values at the interior mesh points are obtained best by multiplying the boundary solution values with the influence factors and then adding the products to the null solution

B2. SOLUTION PROCEDURE

In line with the operational features of the method of superposition, discussion of the solution procedure will be divided into two sections: one on interior mesh points and the other on boundary mesh points. Section B.1.3 - Interior mesh points, is taken from the work of Gunaratnam and Perkins. The rudiments of Section B.1.4 on Boundary mesh points are from the work of Wood, Harley and Perkin10s; however, the actual procedure discussed is a result of the study by Dailey and Harlema11n

B3. INTERIOR MESH POINTS

The notation in this section is taken directly from Gunaratnam and Perkins[1]. It differs from the notation used previously in that it is adjusted to conform more closely with subscripted variable notations in FORTRAN programming. The time step superscript n+1 has been dropped from water surface elevations and discharges.

At mesh point j, the water surface elevation z(j) and the discharge Q(j) may be written in terms of their respective null solutions and products of the appropriate influence factors and boundary values:

$$z(j) = z^0(j) + dx_1(j)\bullet z(1) + dx_3(j)\bullet Q(1) + dx_{11}(j)\bullet z(N) + dx_{33}(j)\bullet Q(N)$$
$$Q(j) = Q^0(j) + dx_2(j)\bullet z(1) + dx_4(j)\bullet Q(1) + dx_{22}(j)\bullet z(N) + dx_{44}(j)\bullet Q(N)$$

where:

$z^0(j)$ = null solution for z(j)

$dx_1(j)$ = influence on z(j) of unit z(1)

$dx_3(j)$ = influence on z(j) of unit Q(1)

$dx_{11}(j)$ = influence on z(j) of unit z(N)

$dx_{33}(j)$ = influence on z(j) of unit Q(N)

$Q^0(j)$ = null solution for Q(j)

$dx_2(j)$ = influence on Q(j) of unit z(1)

$dx_4(j)$ = influence on Q(j) of unit Q(1)

[10] Eric F. Wood, Brendal M. Harley and Frank R. Perkins, "Operational Characteristics of a Numerical Solution for the Simulation of Open Channel Flow", Report No. 150, M.I.T. R72-30, June 1972.

[11] James E. Dailey and Donald R.F. Harleman, "Numerical Model for the Prediction of Transient Water Quality in Estuary Networks", Report No. 158, M.I.T. R72-72, Oct. 1972.

$dx_{22}(j)$ = influence on Q(j) of unit z(N)

$dx_{44}(j)$ = influence on Q(j) of unit Q(N)

The set of simultaneous linear equations resulting from the finite difference scheme and consisting of Equation (3.70) of Dailey and Harleman7 and the corresponding boundary equations can be written in a matrix form such as

$$[A] \bullet [x] = [d]$$

where:

$$[A]= \begin{array}{cccccccccc}
b_3^1 & b_4^1 & c_3^1 & c_4^1 & & & & & & \\
a_1^2 & a_2^2 & b_1^2 & b_2^2 & c_1^2 & c_2^2 & & & & \\
a_3^2 & a_4^2 & b_3^2 & b_4^2 & c_3^2 & c_4^2 & & & & \\
 & & & & & & & & & \\
 & & & & a_1^{N-1} & a_2^{N-1} & b_1^{N-1} & b_2^{N-1} & c_1^{N-1} & c_2^{N-1} \\
 & & & & a_3^{N-1} & a_4^{N-1} & b_3^{N-1} & b_4^{N-1} & c_3^{N-1} & c_4^{N-1} \\
 & & & & & & a_1^N & a_2^N & b_1^N & b_2^N
\end{array}$$

and:

$$[x] = \begin{vmatrix} z_1 \\ Q_1 \\ z_2 \\ Q_2 \\ \\ z_{N-1} \\ Q_{N-1} \\ z_N \\ Q_N \end{vmatrix} \qquad [d] = \begin{vmatrix} \\ d_2^1 \\ d_1^2 \\ d_2^2 \\ \\ d_1^{N-1} \\ d_2^{N-1} \\ d_1^N \\ \\ \end{vmatrix}$$

The coefficients a, b, c and d relate to the terms of the finite difference equation and are defined precisely by Dailey and Harleman.

The null solution and the influence coefficients may be obtained in a single operation by solving the following matrix equation in the computer.

$$[A'] \bullet [x'] = [d']$$

Where:

[A'] = the (2N-4) x (2N-4) bitridiagonal submatrix which is defined by the dashed lines in the coefficient matrix [A] of Equation (11).

$$[x'] = \begin{vmatrix} z^0(2) & dx_1(2) & dx_3(2) & dx_{11}(2) & dx_{33}(2) \\ Q^0(2) & dx_2(2) & dx_4(2) & dx_{22}(2) & dx_{44}(2) \\ : & : & : & : & : \\ : & : & : & : & : \\ z^0(N\text{-}1) & dx_1(N\text{-}1) & dx_3(N\text{-}1) & dx_{11}(N\text{-}1) & dx_{33}(N\text{-}1) \\ Q^0(N\text{-}1) & dx_2(N\text{-}1) & dx_4(N\text{-}1) & dx_{22}(N\text{-}1) & dx_{44}(N\text{-}1) \end{vmatrix}$$

$$[d'] = \begin{vmatrix} d_1^{\,2} & -a_1^{\,2} & -a_2^{\,2} & 0 & 0 \\ d_2^{\,2} & -a_2^{\,2} & -a_4^{\,2} & " & " \\ : & 0 & 0 & " & " \\ : & " & " & 0 & 0 \\ d_1^{\,N-1} & " & " & -c_1^{\,N-1} & -c_2^{\,N-1} \\ d_2^{\,N-1} & 0 & 0 & -c_3^{\,N-1} & -c_4^{\,N-1} \end{vmatrix}$$

The first column of [d'] supplies the null solution when water surface elevation and discharge are pre-scribed as zero at the upstream and downstream boundaries. The remaining four columns in [d'] are the carry overs from the appropriate columns outside the dashed lines in matrix [A] of Equation (11). When unit water surface elevations and discharges are prescribed at the boundaries, the appropriate columns in [A] are multiplied by 1 and moved to victors in [d'].

A double sweep algorithm for solving a set of simultaneous linear equations whose coefficient matrix is bitri-diagonal is given by Gunaratnam and Perkins. The five solution vectors in [x'] are obtained by generalizing this algorithm to handle five right hand sides. Since the solution procedure is essentially Gauss reduction, the coefficient matrix must be decomposed only once rather than five times.

Once the boundary values are computed, as shown in the following section, z(j) and Q(j) may be deter-mined by back substitution.

B4. BOUNDARY MESH POINTS

From equation (11) the first and last equations remain to be used in solving for the boundary values. These equations may be written in terms of only the boundary values by replacing z(2), Q(2), z(N 1) and Q(N 1) with equations (9) and (10).

$$\alpha_1 z(1) + \alpha_2 Q(1) + \alpha_3 z(N) + \alpha_4 Q(N) = D_2^1$$
$$\beta_1 z(1) + \beta_2 Q(1) + \beta_3 z(N) + \beta_4 Q(N) = D_1^N$$

where:

$\alpha_1 = b_3(1) + dx_1(2) \cdot c_3(1) + dx_2(2) \cdot c_4(1)$

$\alpha_2 = b_4(1) + dx_3(2) \cdot c_3(1) + dx_4(2) \cdot c_4(1)$

$\alpha_3 = dx_{22}(2) \cdot c_3(1) + dx_{22}(2) \cdot c_4(1)$

$\alpha_4 = dx_{33}(2) \cdot c_3(1) + dx_{44}(2) \cdot c_4(1)$

$ß_1 = dx_1(N-1) \cdot a_1(N) + dx_2(N-1) \cdot a_2(N)$

$ß_2 = dx_3(N-1) \cdot c_1(N) + dx_4(N-1) \cdot a_2(N)$

$ß_3 = b_2(N) + dx_{11}(N-1) \cdot a_1(N) + dx_{22}(N-1) \cdot a_2(N)$

$ß_4 = b_2(N) + dx_{33}(N-1) \cdot a_1(N) + dx_{44}(N-1) \cdot a_2(N)$

$D_2^1 = d_2^1 - c_3(1) \cdot z^0(2) - c_4(1) \cdot Q^0(2)$

$D_1^N = d_1^N - a_1(N) \cdot z^0(N-1) - a_2(N) \cdot Q^0(N-1)$

Equations (13) and (14) and two boundary conditions are sufficient to solve for the water surface eleva¬tions and discharges at the boundary mesh points. In formulating the network solution procedure it is useful to consider the example network in Fig. B-1 once again. For each reach that enters a node, the boundary water surface elevation and the boundary discharge are unknown. As an example, there are 6 unknowns at node 2. To solve for these unknowns there is one boundary equation for each reach which enters the node. The remaining equations must be specified by boundary conditions. Thus, at exterior nodes 1 and 4, a single boundary condition specifying water surface elevation or discharge is required. At interior nodes 2 and 3, the required boundary conditions are supplied by compatibility of water sur-face elevations and continuity of discharge. Returning to node 2, the compatibility condition is:

$$z_N^I = z_1^{II} = z_1^{III}$$

where:

z_N^I = water surface elevation at mesh point N of reach I

z_1^{II} = water surface elevation at mesh point 1 of reach II

z_1^{III} = water surface elevation at mesh point 1 of reach III

and the continuity condition is:

$$Q_N^I - Q_1^{II} - Q_1^{III} = 0$$

where:

Q_N^I = discharge at mesh point N of reach I

Q_1^{II} = discharge at mesh point 1 of reach II

Q_1^{III} = discharge at mesh point 1 of reach III

The equations are grouped for solution by looping over the nodes. At each node, the compatibility condition is enforced by solving for a single water surface elevation. Then the boundary equations off-set the unknown discharges which enter the node, and either a boundary condition or a continuity condition offsets the water surface elevation. In matrix form, the set of equations for the boundary values of the example network are grouped as follows:

$$
\begin{bmatrix}
\alpha_1^I & \alpha_2^I & \alpha_3^I & \alpha_4^I & & & & & & & & \\
1 & & & & & & & & & & & \\
\beta_1^I & \beta_2^I & \beta_3^I & \beta_4^I & & & & & & & & \\
 & & \alpha_1^{II} & & \alpha_2^{II} & & \alpha_3^{II} & \alpha_4^{II} & & & & \\
 & & \alpha_1^{III} & & & \alpha_2^{III} & \alpha_3^{III} & & \alpha_4^{III} & & & \\
 & & & 1 & -1 & -1 & & & & & & \\
 & & \beta_1^{II} & & \beta_2^{II} & & \beta_3^{II} & \beta_4^{II} & & & & \\
 & & \beta_1^{III} & & & \beta_2^{III} & \beta_3^{III} & & \beta_4^{III} & & & \\
 & & & & & & \alpha_1^{IV} & & & \alpha_2^{IV} & \alpha_3^{IV} & \alpha_4^{IV} \\
 & & & & & & & 1 & 1 & -1 & & \\
 & & & & & & \beta_1^{IV} & & & \beta_2^{IV} & \beta_3^{IV} & \beta_4^{IV} \\
 & & & & & & & & & & & 1
\end{bmatrix}
\begin{bmatrix}
z_1 \\ Q_1^I \\ z_2 \\ Q_N^I \\ Q_1^{II} \\ Q_1^{III} \\ Z_3 \\ Q_N^{II} \\ Q_N^{III} \\ Q_1^{IV} \\ z_4 \\ Q_N^{IV}
\end{bmatrix}
=
\begin{bmatrix}
D_2^{1,I} \\ z_{mean} \\ D_1^{N,I} \\ D_2^{1,II} \\ D_2^{1,III} \\ 0 \\ D_1^{N,II} \\ D_1^{N,III} \\ D_2^{1,IV} \\ 0 \\ D_1^{N,IV} \\ Q_{mean}
\end{bmatrix}
\qquad (17)
$$

In equation (17) water surface elevation z is specified at node 1 and discharge Q is specified at node 4.

A very fortunate result of grouping the equations in this way is that the coefficient matrix of equation (17) is banded. The bandwidth can be determined by checking each reach and looking for the maxi-mum throws of the nodal submatrices from the main diagonal. In large networks, the banded structure of the coefficient matrix becomes very important computationally. The number of unknowns being evaluated in equation (17) is twice the number of reaches plus the number of nodes. For a network of 30 reaches and 30 nodes, the coefficient matrix will be 90 x 90, requiring more than 32K bytes of com-puter storage (IBM 360 or 370 System) in standard precision. Depending on the network topology and efficient node numbering, banding may significantly reduce the core storage requirements and improve execution times.

B5. INITIAL CONDITIONS

It is presumed that one knows the initial conditions, i.e., z and Q at all computational stations at time t=0. In the absence of such detailed knowledge approximate values should be given.

In the steady state studies a convergence to steady state values will proceed. The better the initial ap-proximation the sooner a convergence is obtained with a corresponding saving in computer time. In making a transient study it is advisable to "lead in" to the initial time of the transient study by a previ-ous steady state run or by a previous transient run of shorter duration. The objective of such a "lead in" is to start the run with the best possible set of initial conditions.

B6. BOUNDARY CONDITIONS

Both initial and boundary conditions must be specified for the existence of a unique solution to the governing equations. For subcritical flow, three possibilities exist. They are:

1) The specification of the discharge Q

2) The specification of the surface elevation Z

3) The specification of a relationship between Z and Q.

As the M.I.T. Open Channel Network Model is applicable only to subcritical flow conditions (practical considerations), only one time history Z(t), Q(t) or Z vs. Q. is required at each boundary. Typical boundary specifications would be a water surface elevation at the downstream boundary of a tidal estuary, a discharge boundary condition for upstream flood flows or releases from a dam, and a Z vs. Q. rating curve for control structures such as weirs, gates and spillways.

The concept of a control structure can be extended to the downstream boundary in long rivers in terms of a stage routing condition. Henderson shows that for flood routing a loop rating curve applies, as shown in Fig. B-2.

The curve in Figure B-2 is defined by:

$$Q = C_z A \left(R(S_o - \frac{\partial h}{\partial x} - \frac{v}{g}\frac{\partial v}{\partial x} - \frac{1}{g}\frac{\partial v}{\partial t}) \right)^{‰}$$

where C_z, the Chezy coefficient, can be expressed in terms of Mannings' 'n' as

$$C_z = \frac{1.486}{n} R^{1/6}$$

For flood routing and for uniform flow in straight channels the terms:

$$\frac{v}{g}\frac{\partial v}{\partial x} \, , \; \frac{1}{g}\,\frac{\partial v}{\partial t} \;<<\; 1$$

$$Q \;=\; C_z A \left(R(S_0 \;-\; \frac{\partial h}{\partial x}) \right)^{‰}$$

Equation 19 can be used to define the relationship of Figure B-2. Gunaratnam and Perkins have used this as a boundary condition inasmuch as the relationship yields a rating curve. They expand Q in a Taylor series in time using the dashed line of Figure B-2 as a basis for the expansion. The Z vs. Q. relationship derived in this manner has given good results in many cases.

B7. CONTROL STRUCTURES AND RAPIDS IN THE NETWORK

The M.I.T. Open Channel Network Model has been modified to consider control structures within the network itself instead of only at the boundaries. In general it is advisable to divide up a large network into smaller ones, using control structures as the natural points of subdivision. This results in large sav-ings in computation costs as well as organizational convenience. Such subdivisions would place control structures at boundaries, but this is not always possible, nor desirable.

The modification to the Model permits the user to specify a boundary condition at the upstream side of the control structure. The upstream side of the control structure becomes a node in the Network Topology a boundary node. The downstream side of the control structure is also a boundary node, distinct from the upstream node. The boundary condition applied at this downstream node will be the discharge calculated at the upstream node at the previous time step or, if discharge were the specified boundary condition, the specified discharge. Figure B-3 shows a typical control structure network where the flow splits a node 2 into two branches. One branch (or reach) goes directly to node 5, whereas the other passes through a control structure. Node 3 is the upstream node of the control structure, node 4 the downstream node. Confluence occurs at node 5.

One special feature, added to the Peace Athabasca Model version, is the addition of a special site specific weir equation.

Rapids

Rapids represent a section of river where flow becomes critical. Although the mathematical model of Gunaratnam and Perkins is valid for such cases it is not practical as the discretization increment, Δx for critical flow would be very small in order to satisfy convergence criteria. Rapids are similar to control structures in that they can be studied in terms of a rating curve. If such a curve can be established by field measurements then the rapids can be treated as a control structure. If obtaining such a curve is impractical, two other possibilities exist. One is to assume a rating curve, such as:

$$q \;=\; 3.33\;\, H^{3/2}$$

where q is the discharge per unit width and H is the depth at the head of the rapids. The other possibility is to treat the upstream boundary of the equivalent control structure as a stage routing type boundary.

Finally, it should be mentioned that a modification, or patch, to the M.I.T. Open Channel Network Model could be made, permitting a "critical flow" reach to be treated as such.

B8. SEA DAMS

The opening and closing of sea dams can now be effectively simulated with the model. A maximum of two structures can be modelled through the addition of Data Group H and TAPE51-TAPE54 Data files as required.

The opening and closing of the sea dams has been successfully implemented for the "Serpentine-Nicomekl Floodplain Mapping Study" in British Columbia where the structures open and close as the tide levels fall and rise.

B8. PUMPING

The pumping subroutine simulates the operation of one or more pumps in a pump station by determining whether each pump should be on or off at each time step and for those pumps which are required to be on by calculating total pump discharge. These switches are located as follows:

1. Local switches at the pump intake which operates the pump during normal operation and information regarding their location and elevations is required by the program.
2. High level emergency switches on the river side of the pump-station are intended to shut off the pumps if exceedingly high water levels occur. Data on these switches is also required for the simulation.
3. A set of remote switches at a distant site which would be used to shift the operating range of the local switches to lower elevations, or into "storm mode". These switches are optional in the model.

The user may specify the location of each set of switches for each pump to be anywhere in the system being modelled, using reach number and distance along the reach. Two elevations are required for each set of switches, one to set the "on" condition and the other to set the "off" state.

The user may specify any number of pumps for each pump station, provided the model has been dimensioned to accept that number. Each pump can have its own switch settings and head-discharge relationship (pump curve). Calculation of discharge uses the pump curves and data on inlet and discharge pipe configurations. Using a method of successive approximations, the program iterates to a solution of head and discharge that satisfies the dynamic head loss calculations and falls on the pump curve

APPENDIX 3 ICE PROCESSES - THEORETICAL BACKGROUND AND COMPUTATIONAL METHODS

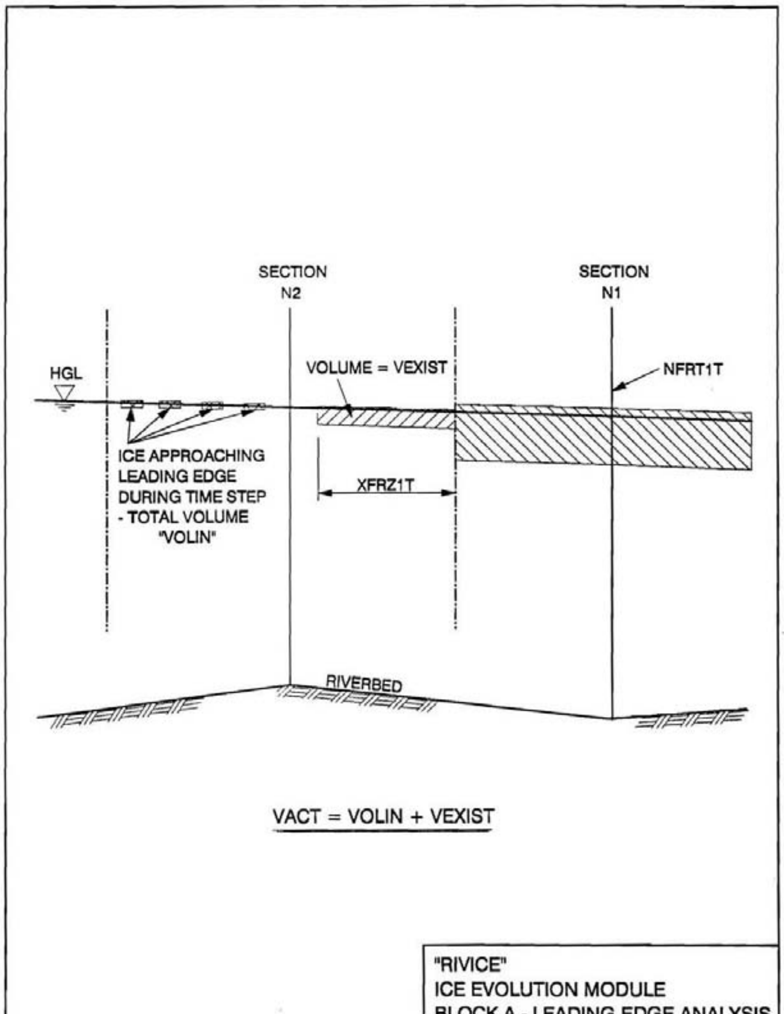
A3.A COMPUTATIONAL METHODS USED TO REPRESENT ICE PROCESSES

The theoretical background and numerical bases for representing the dominant ice processes that are addressed by RIVICE are summarized in Section 2. There are three key strategies of carrying out the numerical simulations that are described below. For each process, it should be noted that the general strategy is for cross section numbers to proceed from upstream (cross section #1) to downstream (cross section "NSNTOT" is the last cross section in the reach).

1. Numerical Representation of the Leading Edge Processes

Sketch 3-1 shows schematically the elements that are involved. The definition of the leading edge location is "NFRT1T", and is the upstream-most cross section where the ice cover extends fully across and over its full length. The incoming ice volume in one time step is represented by the variable "VOLIN", and has been calculated prior to this point by Subroutines ICEGENER and ICEMOVEMENT. The amount of ice that already exists upstream of the leading edge is located in the domain of cross section "N2" in Sketch 3-1. It has a length of "XFRZ1T" and a thickness of "TLE1T" and displaces a volume of "VEXIST".

In this time step the program uses the criterion that is selected by the user (one of three options as described in Section 2.2.5) to define the thickness of the ice front and whether ice can indeed accumulate or whether it must be drawn under the leading edge and form "ice-in-transit". The important point is that the model combines the volumes "VOLIN+VEXIST" and either allows it to accumulate with the thickness prescribed, or be submerged ("VOLSUB") where it is transported downstream.



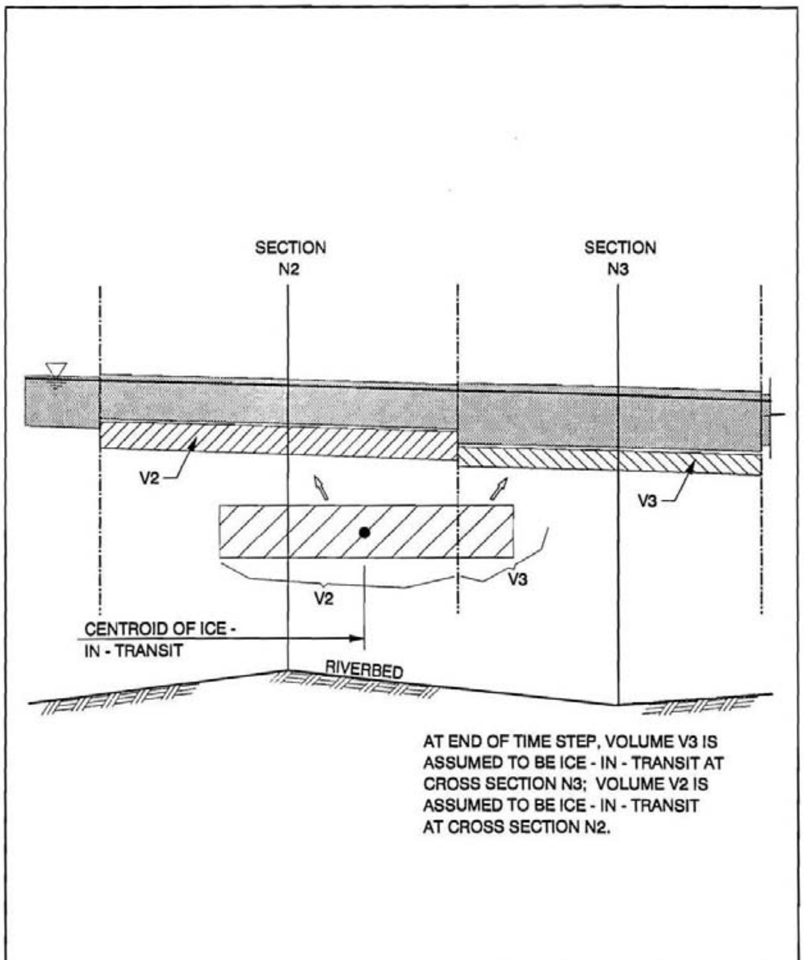
Sketch 3-1

2. Numerical Representation of "Ice-In-Transit"

"Ice-in-transit" can deposit under the ice cover if the velocity at this location and at this time step is sufficiently low to permit that. The user has three possible options to characterize and quantify this, as described in Section 2.2.6. If there is no deposition, then the ice is transported as far as is computed based on the prevailing velocities at each cross section. At that point, the centroid of the ice-in-transit is located relative to the divisions between cross sections, as shown schematically in Sketch 3-2. rather than a complicated form of tracking the ice at partial distances between cross sections, the program is formulated to split the ice between the cross sections that are straddled by the ice-in-transit. In the example shown in Sketch 3-2, at the end of the time step, two portions of the total volume are split. The downstream portion is assigned to the downstream cross section "N2", and the upstream portion is assigned to the upstream cross section. The split is decided based on the location of the centroid as shown in Figure 3-2. The new volumes, V2 and V3, then form the starting point for the next time step.

"Ice-in-transit" can deposit under the ice cover if the velocity at this location and at this time step is sufficiently low to permit that. The user has three possible options to characterize and quantify this, as described in Section 2.2.6. If there is no deposition, then the ice is transported as far as is computed based on the prevailing velocities at each cross section. At that point, the centroid of the ice-in-transit is located relative to the divisions between cross sections, as shown schemati-

cally in Sketch 3-2 rather than a complicated form of tracking the ice at partial distances between cross sections, the program is formulated to split the ice between the cross sections that are straddled by the ice-in-transit. In the example shown in Sketch 3-2, at the end of the time step, two portions of the total volume are split. The downstream portion is assigned to the downstream cross section "N2", and the upstream portion is assigned to the upstream cross section. The split is decided based on the location of the centroid as shown in Figure 3-2. The new volumes, V2 and V3, then form the starting point for the next time step.



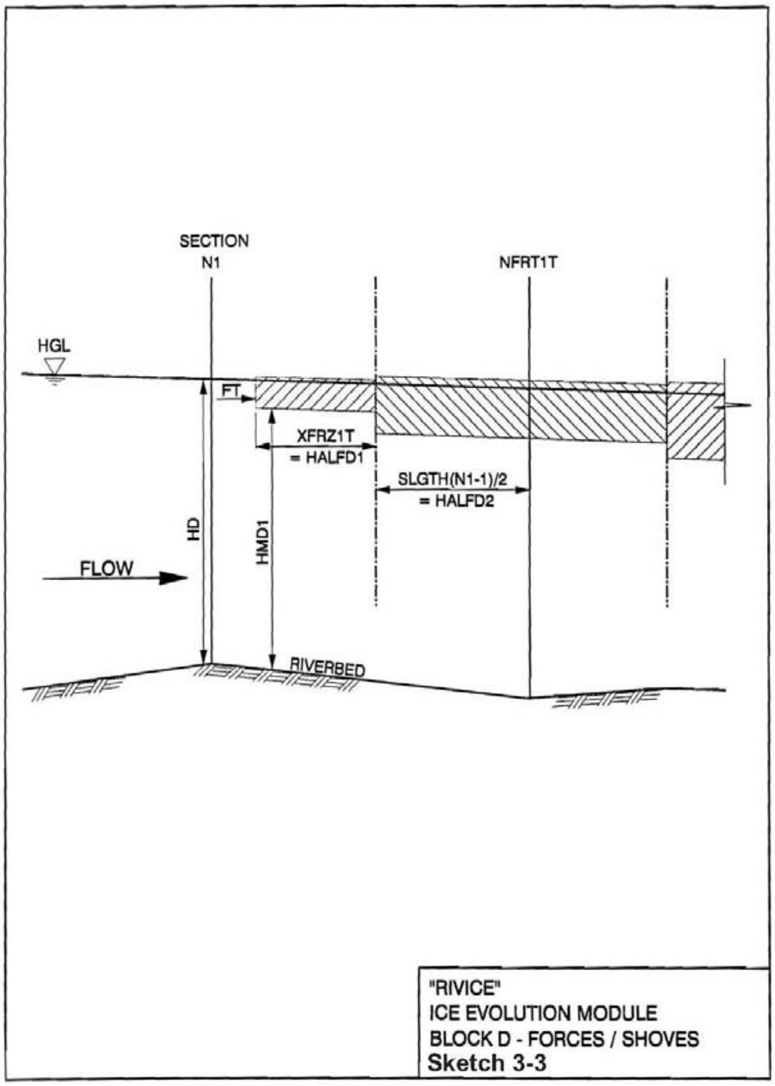
"RIVICE"
ICE EVOLUTION MODULE
BLOCK C - ICE TRANSPORT
Sketch 3-2

3. Numerical Strategy to Compute Forces and Shoves

Figure 3-3 shows a schematic representation of the ice cover at the leading edge of an ice segment. The starting point for force calculations is at the upstream end of the length denoted as "XFRZ1T" in Sketch 3-3. The thrust on the leading edge of that segment is computed as FT, using the algorithm described in Section 2.2.7. Other similar forces are also computed as described in Section 2.2.7. The forces are computed in increments of 1/10th of the length "HALFD1" shown in Sketch 3-3, and then continued for increments of 1/10th the length of "HALFD2". Once the residual force is obtained at the location of NFRT1T, it is then compared to the resistance that can be provided by the existing ice thickness at that location. If there is sufficient resistance, the calculations proceed in two parts to the next downstream cross section. The first is in 1/10th increments to the dividing line between the cross sections, and then in 1/10th increments top the next cross section number (in the example of Sketch 3-3, NFRT1T + 1).

If the resistance is less than the load, then a shove is simulated to thicken the ice adequately to resist the load. However, this is done only at a rate that is realistic, given the water velocity and the speed at which the upstream ice cover could move to supply the ice volume required.

The following is the information supplied by Dr. S. Beltaos to assist the development of a method to estimate the manning n-value for an ice cover dependent on its thickness.



A3.B HYDRAULIC RESISTANCE OF RIVER ICE COVERS

Prepared by S. Beltaos, Dec. 14, 2008, to assist in formulating relevant algorithms in the RIVICE numerical model. This material reflects the writer's understanding of this complex issue and is presented very briefly, without rigorous proofs and thorough literature review.

It is not written to publication standard – merely contains a synthesis of published material.

1. Background Information

The well-known Darcy-Weisbach friction factor, f, is a dimensionless quantity describing the hydraulic resistance of a flow boundary, defined as:

$$f \equiv 8\tau / \rho V^2 \qquad (1)$$

in which τ = boundary shear stress; ρ = water density; and V = mean velocity. In fully rough turbulent flow, such as prevails in most natural streams, the friction factor is a logarithmic function of the relative roughness, defined as the ratio of the roughness height of the boundary to the hydraulic radius (R) associated with this boundary. The roughness height is often quantified using Nikuradse's equivalent sand-roughness (k_s), with the understanding that the roughness of a particular boundary is equal to some constant times k_s. For the beds of gravel/cobble streams, Limerinos' findings (1970; USGS Water Supply Paper 1898-B), suggest that k_s = 3.2d, with d being the 84th percentile of the bed material size (usually denoted as d_{84}, but I am dropping the suffix for convenience). The Nikuradse logarithmic equation then transforms to:

$$f = [1.16 + 2\log(R/d)]^{-2} \quad (2)$$

This relationship is shown graphically in the following figure. [A plot of f vs. d/R would have been more natural, but the graph of Fig. 1 is the customary version]. Though the logarithmic structure of the friction factor – relative roughness relationship has been known for many years, it is cumbersome in practical applications, even with open-water conditions, and engineers have sought simplification via approximate, power-type expressions. When it comes to two-boundary flows, such as occurs in ice-covered streams, the problem is not simply a matter of convenience: it also not possible to obtain explicit expressions of the composite-resistance parameters f (or n) when we use the logarithmic equation.

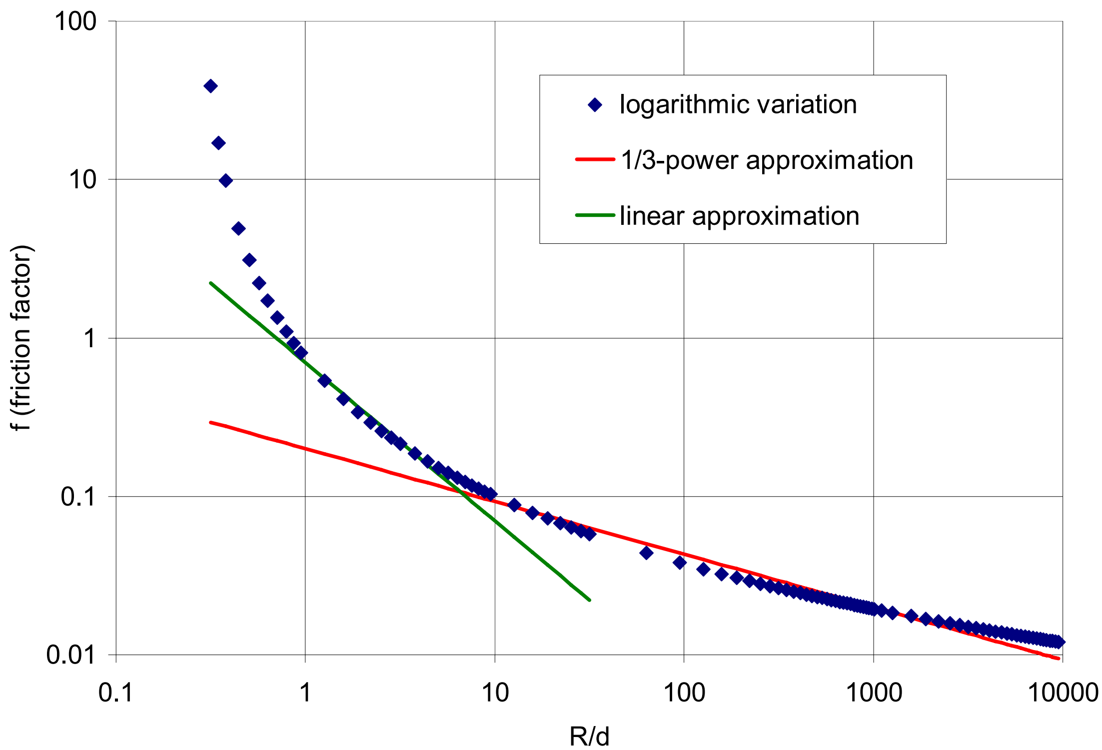


Figure 1: Variation of f with R/d in fully rough turbulent flows

We can see that when R/d is between 10 and 2000 (common range for rivers), f varies approximately in proportion to $(R/d)^{-1/3}$, i.e.:

$$f \approx 0.2(R/d)^{-1/3} = 0.2(d/R)^{1/3} \quad \text{(for } 10 < R/d < 2000\text{; } d/R = 0.0005 \text{ to } 0.1) \quad (3)$$

CAVEAT: The upper limit of the R/d range for Eq. 3 derives from the mathematical calculations and is subject to the fully-rough flow requirement. In reality, flows with very large values of R/d may not be fully rough, and the friction factor would then also depend on the Reynolds number. This limitation is not expected to influence the present discussion, but the readers should be aware of its existence because it can come into play when we deal with open water conditions in sluggish rivers with very fine bed materials.

The Manning coefficient n and the friction factor f are related by the identity (since $V = R^{2/3} S_f^{1/2} / n = [(8/f)gRS_f]^{1/2}$, with S_f = friction slope):

$$n = 0.113R^{1/6}\sqrt{f} \qquad \text{metric units)} \qquad (4)$$

where the numerical coefficient is based on the value 9.81 m/s^2 for the gravitational acceleration (g).

Using Eq. 3, we can re-arrange Eq. 4 to read:

$$n \approx 0.05d^{1/6} \text{ (for 10 <R/d < 2000; d/R = 0.0005 to 0.1)} \qquad (5)$$

This equation shows that, within the specified range of R/d, n is solely a property of the boundary and explains its robustness and popularity among hydraulic engineers. Note that the numerical coefficient in Eq. 5 (which is essentially the Strickler equation) will change if a different statistic of the bed material size (e.g. median value) is used. Example: for d = 1 cm (0.01 m), n = 0.023. Equation 5 does not apply when R/d < 10, or when the relative roughness d/R exceeds 0.1.

As the relative roughness increases above 0.1 (and R/d decreases below 10), the Manning coefficient will be increasingly dependent on the hydraulic radius as well as on the boundary roughness. In the range R/d ~ 0.8 to 7 (d/R ~ 0.14 to 1.25), Fig. 1 indicates that

$$f \approx 0.7(R/d)^{-1} = 0.7(d/R) \text{ (for 0.8 <R/d < 7; d/R ~ 0.14 to 1.25)} \qquad (6)$$

This equation shows f to increase linearly with relative roughness (d/R) and leads to:

$$n \approx 0.095d^{1/2}R^{-1/3} \text{ (for 0.8 <R/d < 7; d/R ~ 0.14 to 1.25)} \qquad (7)$$

The roughness is still the predominant factor in determining the Manning coefficient, but the influence of the hydraulic radius is also significant.

CAVEAT: Even though Fig. 1 shows Eq. 2 plotted for R/d down to ~ 0.3, there are few data to justify such extrapolation, and I would be very sceptical about using f-values much above 1.

As noted earlier, most natural streams under open water conditions have fully rough flow with a constant Manning coefficient. However, it is known from experience that at very low stages, n is higher than what is indicated by Eq. 5. This has been attributed to the increasing prominence of channel irregularities as the river stage drops.

2. Ice Jams Composed of Ice Blocks (Freeze up/Breakup)

This is a situation of extreme roughness, where conventional assumptions of the two-layer, log-law formulation do not hold. Two problems have been identified:

(a) The hydraulic resistance coefficient (Manning n) does not depend solely on the absolute roughness of the boundary but also on the hydraulic radius controlled by that boundary. This effect has been known from gravel/cobble - bed rivers and also from laboratory experiments with very large roughness elements.

(b) In the presence of a very rough cover, like an ice jam, the riverbed appears to be rougher than what might be expected from the size of the bed material, which "works" fairly well in estimating resistance coefficients under open-water conditions. Consequently, the apparent n_b (Manning coefficient of the riverbed) is greater than the open-water value. This effect has been noticed by the writer, based on analysis of own and others' ice jam data. It could be related to increasing predominance of bed irregularities as the bed-controlled hydraulic radius decreases (no solid proof at hand as yet).

The formulation of hydraulic resistance in RIVJAM is designed to account for these effects, and to allow for the now fully established relationship between jam thickness and absolute roughness. For accumulation covers, the roughness is extreme and d/R is of the order of 1. In this range, f varies in proportion to d/R (see Eq. 6 above), so that n will now depend on both d and R (Eq. 7

above). Nezhikhovskiy's (1964) data represent a limited range of R (~ 1 to 1.5 m) and this could explain why he did not discuss the effect of the hydraulic radius.

RIVJAM computes the composite friction factor (f_o) under the jam as:

$$f_o = c t_s^{m_1} h^{-m_2} \quad (8)$$

in which t_s = "keel" of the jam, taken as 0.92 x jam thickness; h = average depth of flow under the jam (~ 2 composite hydraulic radius); c = dimensionless coefficient; and m_1, m_2 = user-specified empirical exponents. The reader may note that the pair $m_1 = m_2 = 0$ results in a constant friction factor, whereas putting $m_1 = 0$ and $m_2 = 1/3$ results in a fixed Manning coefficient, equal to $0.10\sqrt{c}$.

Practical experience and ice jam thickness/roughness measurements indicate that the absolute roughness of the jam, d, (84^{th} percentile) increases linearly with the thickness while the d/R ratio typically falls in the extreme-roughness range (Eq. 6). After some algebra, it can be shown that the most likely combination for m_1 and m_2 is $m_1 = m_2 = 1$. This is confirmed by experience with calibrations of the RIVJAM model. In most applications, therefore, Eq. 8 simplifies to:

$$f_o = c(t_s / h) \quad (9)$$

The data and analysis leading to Eq. 9 derive from ice-jam-profiler measurements on breakup jams and are detailed in Beltaos (2001; ASCE JHE Vol. 127, No. 8, pp. 650-656). Experience with Eq. 9 indicates that the coefficient c is typically in the range 0.4 to 0.6.

The data sets that form the basis of Equations. 8 and 9 do not extend beyond jam thicknesses of 5 m, therefore it is not known whether extrapolations beyond this value are credible. To ensure that RIVJAM does not generate implausible values, the composite Manning coefficient, n_o, is also calculated and not allowed to exceed 0.10, or a user-specified value. If the program computes a value exceeding 0.1, then f_o is adjusted so that $n_o = 0.1$. The value $n_o = 0.10$ represents an upper limit to what is known from experience. This limitation is typically "activated" in grounded or nearly grounded jams where the toe is very thick and the flow depth very small.

At the low end of the spectrum, RIVJAM will calculate thicknesses approaching zero near the head of the jam, even though it is not physically possible for the jam thickness to be less than the thickness of the individual ice blocks that make up the jam. This is not a problem as far the length of the jam goes, because the calculated length will be very slightly greater than the real length of the jam. However, Equations 8 and 9 would generate implausibly small values for the hydraulic resistance. To avoid this, RIVJAM places a user-specified lower limit on n_o. The default value is 0.03 and reflects the writer's judgment for a surface accumulation of ice blocks. This could also apply to a thin jam over a very deep channel

Because natural streams are much wider than they are deep, the wetted perimeter of the bed is assumed to be equal to the width of the channel at the elevation of the average bottom surface of the jam. Consequently, $h \approx 2R_o$ (with R_o = hydraulic radius of the composite flow). The following, well-known, relationships are used to determine the individual friction factors and hydraulic radii associated with the bed and the ice cover:

$$f_i / f_o = R_i / R_o \qquad f_b / f_o = R_b / R_o \qquad f_i / f_o + f_b / f_o = 2 \quad (10)$$

Note that the last relationship in Eq. 4 can also be written as $f_o = (f_i+f_b)/2$, which is equivalent to the Sabaneev formula that is expressed in terms of the Manning coefficients.

In RIVJAM the ratio f_i/f_o is user-specified input. Modelling experience and available measurements indicate that f_i/f_o = 1.0 to 1.2 (and thence f_b/f_o = 1 to 0.8). With this information, the shear stresses associated with the bed and the ice cover can also be determined. If τ_o is the composite flow shear stress, equal to $(f_o/8)\rho V^2$ (with ρ = density of water, V = mean flow velocity), then:

$$\tau_i / \tau_o = f_i / f_o \qquad \tau_b / \tau_o = f_b / f_o \tag{11}$$

The shear stresses can also be expressed in terms of the hydraulic radius and friction slope S_f:

$$\tau_o = \rho g R_o S_f \qquad \tau_i = \rho g R_i S_f \qquad \tau_b = \rho g R_b S_f \tag{12}$$

Suggested approach for RIVICE

To use the above relationships in RIVICE, which is geared to work with Manning coefficients rather than friction factors, one can utilize Eq. 4. Then, Eq. 9 results in (remembering to substitute h/2 for R_o):

$$n_o = 0.10\sqrt{c}\, t_s^{1/2} h^{-1/3} \tag{13}$$

with the constraints: $0.03 < n_o < 0.10$ (or other, user-specified, limits). Recall that the default range for c is 0.4 to 0.6.

In RIVICE, it may be required to also calculate the values of the ice- and bed- coefficients n_i and n_b. This can be accomplished by noting that

$$n_i / n_o = (R_i / R_o)^{2/3} = (f_i / f_o)^{2/3} \qquad n_b / n_o = (R_b / R_o)^{2/3} = (f_b / f_o)^{2/3} \tag{14}$$

3. Freeze up Accumulations of Slush

Nezhikhovskiy (1964) also presented approximate relationships between n_i and thickness for freeze up ice covers consisting of loose slush and dense (frozen) slush. I am not sure how he defined these terms, but I suppose they refer to the amount of solid ice that is present in representative ice pans that make up the accumulation. Ice pans typically comprise a thin layer of solid ice underlain by a porous layer of frazil particles and flocs. The longer they travel before being arrested to form an ice cover, the thicker will be the solid-ice layer. Available measurements (Jasek et al., papers in CRIPE Workshops 2003-2007) indicate that the aggregate thickness of an ice pan is ~ 0.5 m, though thinner and thicker ones may also be encountered. In any event, I doubt whether the Manning coefficient of a slush ice cover, even if it is a mere surface accumulation, can drop below ~ 0.02 or 0.03 (Nezhikhovskiy shows values down to 0.01).

For RIVICE, I would recommend using Eq. 13 with a reduced value of the coefficient c when the cover is thicker than about 1 m, and with reduced limiting values for n_o (e.g. 0.03/0.09 for dense slush and 0.02/0.06 for loose slush). Of course these numbers will be better defined when the model is applied to actual case studies. Ice covers thinner than about 1 m are very likely to have ordinary roughness, so they could be handled with the conventional Sabaneev approach (n_o is calculated from the values of n_i and n_b via the well-known averaging of the 3/2-powers).

4. Winter Ice Cover

The preceding considerations apply specifically to the initial condition of the ice cover that is, when it is newly formed. With the passage of time, the ice cover becomes smoother, as already discussed by Nezhikhovskiy. In my view there are two ways in which the cover can smoothen.

a) The freeze up ice cover is a thick and porous accumulation of slush, which does not completely freeze solid during the winter. Irregularities on the bottom surface are gradually obliterated by means of thermal erosion, and possibly by preferen tial transport of material from protruding areas and deposition into cavities (there is a 1991 conference paper by Zufelt and Ashton where theoretical aspects of these mechanisms are discussed).

b) The freeze up ice cover is not very thick; thermal ice growth, which proceeds downwards, eventually results in an underside that consists of solid ice. In mid-winter this surface is very smooth, with n_i values being ~ 0.010-0.012. The same would apply in cases where the slush layer is thick, but the combined effects of thermal attrition from below, and of solid-ice growth from above, result in a solid-ice flow boundary.

For modelling purposes, one could use Nezhikhovskiy's negative exponential form, and adapt it to the composite Manning coefficient, i.e.:

$$n_o(T) = (n_o)_{init} + [(n_o)_{init} - (n_o)_{end}]\exp(-kT) \quad (15)$$

in which the suffixes "init" and "end" denote the freeze up and end-of winter values; T = time since freezeup in days; and k is a user-specified coefficient (in $days^{-1}$). The end-of-winter value is by definition the lowest that can occur during the season. Both k and $(n_o)_{end}$ should be based on local experience. If no actual measurements are available, the user would have to exercise some judgment, after taking into account the likely condition of the bottom of the ice cover at the end of winter (case (a) or (b) above). For case (b), the low value of n_i (~ 0.01-0.012) means that the Strickler formulation applies, and therefore $(n_o)_{end}$ can be calculated from n_i and n_b via the Sabaneev relationship. The same may be true for case (a) if the winter smoothing brings n_i down to non-extreme values (~ 0.04 or less).

With the approach of spring, the smooth bottoms of solid-ice sheets begin to develop two dimensional ripples, at first, and random roughness patterns later on. The Manning coefficient, n_i, increases from ~ 0.01 to nearly 0.03 (Carey, 1966, 1967; USGS Professional Papers 550-B and 575-C). If the bottom of the cover still consists of porous slush during the pre-breakup period, I do not know whether the thermal effect will make the cover smoother or rougher.

CAVEAT: Nezhikhovskiy (1964) does not present detailed evidence supporting the negative exponential form of the resistance decline during the winter, and his linking the decay coefficient to the severity of the winter. Where there are some local data, one should use whatever form is suggested by the measurements.

APPENDIX 4 USER SPECIFIED PARAMETERS WITH DEFAULT VALUES

A4.A THE FOLLOWING IS A LIST OF ALL THE DEFAULT VALUES CURRENTLY USED IN RIVICE

Default Values

NOTATION	DESCRIPTION	DEFAULT
A1COEF	Coefficient specified by user, see Data BB-b-4	0.054
B1COEF	Coefficient specified by user	
COHESN	Cohesion per unit area of ice/bank interface	0.0 pa
DEPOPT(1)	Option number for ice deposition methodology for ice cover evolution during freeze-up	1
DIAICE(1)	Diameter of broken ice fragments for a Meyer-Peter sediment transport analogy for ice depositions	0.3 m
EROPT(1)	Option number for ice cover erosion 1 methodology	1
FRMAX(1)	Maximum densimetric froude number 0.2 above which deposition of ice is not possible	0.2
FTRLIM(1)	Limiting tractive force above which 10 pa erosion of the ice cover will occur, in an ice cover evolution during freeze-up mode	10 pa
LEOPT(1)	Indicator of which algorithm to use to simulate leading edge stability	3
POROSC	User specified value of porosity of ice cover	0.7
POROSFS	User specified value of porosity of ice pans approaching leading edge of ice cover	0.5
VDEP(1)	Maximum velocity for ice deposition under ice cover	1.2 m/s
VERODE(1)	Maximum velocity for ice erosion, in an ice cover evolution during freeze-up mode	1.8 m/s
VERODE(2)	Maximum velocity for ice erosion, in an ice cover evolution during break-up mode	2.0 m/s
VFACTR(1)	Factor to be applied to flow velocity to determine velocity of movement of ice-in-transit, in an ice cover evolution during freeze-up mode	0.9

ZZK1TAN	Coefficient relating transfer of stress to the river bank	0.15
ZZK2	Coefficient of ice strength analogous to passive conditions in soil mechanics	8.5