On the issues associated with 2-D modeling for flood mapping purposes

Devinder S. Dhillon, Fabián A. Bombardelli, Wm. E. Fleenor and Kaveh Zamani

Source: http://www.eonfusionflood.com/applications
Table of Contents

ACKNOWLEDGMENTS .................................................................................................................. 5

CHAPTER 1: INTRODUCTION ................................................................................................. 6

CHAPTER 2: LITERATURE REVIEW ......................................................................................... 9
1. 1-D AND 2-D EQUATIONS .................................................................................................. 9
2. 1-D OR 2-D ....................................................................................................................... 11
3. 1-D/2-D COUPLED ........................................................................................................... 12
4. DATA .................................................................................................................................. 12
5. 1-D AND 2-D MODELS ..................................................................................................... 14
6. MODELER .......................................................................................................................... 23
7. ROUGHNESS COEFFICIENTS ......................................................................................... 23
8. NUMERICAL STABILITY .................................................................................................... 26
9. TIME STEPS ...................................................................................................................... 27
10. CALIBRATION AND VALIDATION ................................................................................ 27
11. BENCHMARK TESTS ......................................................................................................... 29

CHAPTER 3: 1-D AND 2-D FLOODPLAIN MODELING ISSUES ............................................ 31
1. HOW DOES ONE DECIDE WHEN TO USE A 1-D STEADY STATE MODEL VERSUS A 1-D
   UNSTEADY STATE MODEL VERSUS A 2-D MODEL (UNSTEADY STATE)? .......................... 31
   • What are the technical considerations in this decision? .................................................. 31
   • What are the financial considerations in this decision? ................................................... 33
   • What are the issues associated with performing Floodway Analysis in Unsteady
     State? ................................................................................................................................. 34
2. HOW DOES ONE DECIDE WHICH MODEL SOFTWARE TO CHOOSE? ............................. 34
   • 1-D: Prismatic Section Evaluation versus Variable Cross Section Geometry? .............. 35
   • 2-D: Grid or Mesh based Terrain? .................................................................................. 35
   • 2-D: If Grid, 4 directional Flow or 8 directional flow, and what size grid? ...................... 35
   • 2-D: Is it necessary to embed 1-D elements and what are the software capabilities to
     do so? ................................................................................................................................. 37
   2-D: Is cost a factor? For current model needs and future derivative works? ..................... 37
   • What Equations are Solved? ............................................................................................ 37
3. HOW DOES ONE EVALUATE THE QUALIFICATIONS OF THE CONSULTANT PREPARING
   THE MODEL? ....................................................................................................................... 39
4. HOW DO LOCAL AGENCIES ENSURE THEY ARE GETTING A GOOD MODEL FOR THEIR
   MONEY? .................................................................................................................................. 39
5. HOW DOES FEMA OR LOCAL AGENCIES COST EFFECTIVELY REVIEW THE MODEL? .... 40
   • What are the technical elements that go into the review of the model? ......................... 41
   • Are there standard submittal templates that can be developed to standardize the model
     submissions? ....................................................................................................................... 42
   • How does FEMA ensure they have trained reviewers for the different models? ............... 42
   • How does FEMA ensure it is reviewing the elements that matter? .................................. 43
6. MANY TIMES THE MODELS HAVE LARGE DATA SETS. WHAT ARE THE DATA STORAGE AND
   HANDLING CONSIDERATIONS? WHAT NEEDS TO BE SUBMITTED WITH A MODEL FOR REVIEW? 43
7. FEMA HAS THE REQUIREMENT TO MAKE THE DATA FREELY AVAILABLE TO REVIEW.
   HOW DOES FEMA HANDLE THE ISSUE OF PROPRIETARY SOFTWARE? ........................ 44
8. SOFTWARE VENDORS WORK TO IMPROVE THEIR MODELING SOFTWARE WITH NEW
   VERSIONS. WHAT ARE THE ISSUES THAT ARISE FROM VERSION CHANGES? .................. 44
   • How does FEMA track software version changes? ......................................................... 44
   • How does FEMA ensure that the developed models remain usable through software
     version changes? .................................................................................................................. 45
9. **WHAT ARE THE ISSUES ASSOCIATED WITH SOCIAL JUSTICE AND MODEL COMPLEXITY?**

What happens when communities do not have the technical resources to review, use, or modify the 2-D model? .................................................................................................................. 46

10. **HOW ARE FLOODWAYS ADDRESSED IN 1-D UNSTEADY OR 2-D MODEL? WHAT HAPPENS WHEN NUMERICAL INSTABILITIES PREVENT 1-D UNSTEADY MODEL FROM BEING VALID?** ............................................................................................................................. 46

11. **HOW ARE MODEL PARAMETERS ESTABLISHED? HOW DOES A MODELER/REVIEWER KNOW IF THE CORRECT PARAMETERS HAVE BEEN USED?** .............................................................. 47

   - In 1-D models n-values are a lumped parameter for uncertainty. How is the n-value different for 2-D models and what are the appropriate n-values? .................................................. 47
   - How should infiltration be handled? .................................................................................. 49
   - How should the hydrology be handled? .......................................................................... 49
   - Should there be guidance regarding when to consider sheet flow and when to consider point discharge? In urban areas, sheet flow may be more prevalent. ........................................ 50
   - How is grid/mesh element size determined? What makes that size appropriate for certain applications? .......................................................................................................................... 50
   - How should time steps be established and reviewed? ...................................................... 50
   - How are the grids established? Can/should mesh convergence be developed to ensure grid is appropriate for physics used? Can/should grid size be tied to the level of accuracy of the topography? .................................................................................................................. 51
   - How should islands and ponds be handled? .................................................................... 52

12. **HOW ARE LEVEE SYSTEMS MODELED?** ........................................................................ 52

   - How are levees modeled? .............................................................................................. 52
   - How are non-levee embankments and roads modeled? ................................................... 53

13. **HOW SHOULD LOCAL DRAINAGE AND IN-GROUND PIPING SYSTEMS BE MODELED?**

When you put them in, when do you leave them out? ......................................................... 53

14. **HOW ARE THE RESULTS MAPPED?** ............................................................................ 53

   - What needs to be done to correctly convey the results of the analysis on a FEMA map? ................................................................................................................................. 53
   - Are the current AE/AH/AO zones appropriate? Should there be a special zone that includes the velocity? ......................................................................................................... 54

15. **IF DIFFERENT MODELS GIVE DIFFERENT RESULTS HOW DOES FEMA HANDLE “MODEL SHOPPING” OR “TWEAK IT UNTIL IT GOES AWAY” ISSUES? IS THIS A JUDGMENT CALL? IF SO, CAN IT BE APPLIED IN THE MAPPING TRANSITION OF ALL MODELS? WHAT WOULD BE THE GUIDELINES?** ................................................................................................................................. 54

16. **HOW ARE ISSUES RELATED TO MODEL CALIBRATION AND GENERAL LACK OF CALIBRATION DATA TO BE ADDRESSED?** ......................................................................................... 54

17. **HOW ARE MODELERS TRAINED? SHOULD THERE BE SOME SORT OF CERTIFICATION PROGRAM?** ................................................................................................................................. 55

18. **HOW SHOULD ISSUES RELATED TO RUN TIMES BE ADDRESSED?** .......................... 55

19. **WHAT KINDS OF HARDWARE AND SOFTWARE INVESTMENTS MUST FEMA MAKE TO VIEW AND USE THE MODELS?** ......................................................................................... 55

20. **HOW CAN MODEL RESULTS BE MOST EFFECTIVELY COMMUNICATED TO THE PUBLIC?** .......................... 55

21. **HOW ARE LOMA DETERMINATIONS MADE FOR HOMEOWNERS REQUESTING A LOMA? HOW ARE BFES ESTABLISHED FOR A SPECIFIC LOCATION?** ........................................... 56

22. **SHOULD THE FEMA MAPS MAKE SOME EFFORT TO DISPLAY THE UNCERTAINTIES? IF SO HOW?** ................................................................................................................................. 57
23. Can FEMA address some of the uncertainties with extra freeboard requirements that are specified in local ordinances?
24. FEMA requires that new models tie into existing models to within 0.5 feet. What challenges does this pose for 2-D models?
25. How are profiles and profile base lines established when 2-D models are used?
26. How does FEMA cost effectively integrate the regulatory 2-D mapping results with the non-regulatory 2-D mapping results? (Edges are smoothed for regulatory, not smoothed for non-regulatory.)
27. Can alluvial fans be adequately/safely modeled in 2-D software? If so, what guidelines are necessary when doing so?
28. Can information like velocity grids be used to assist OES in emergency evacuation planning? If so what is the best way to use this information?
29. What are some of the ways this information can be displayed electronically besides through the FEMA maps?
30. How do models match the effective modeling? Or should they? 2-D results are different from 1-D but has to match the effective model;
31. Is it possible to develop a test that compares level of detail with needed accuracy?
32. If you run steady state and unsteady model for same reach, is there residual risk in the area that is NOT in the SFHA for the unsteady model?
33. Since 2-D calculates flood depth more accurately than 1-D, should 1% boundary be mapped just to the <1 ft depth threshold? Or all the way to 0 ft depth?
34. It is not possible to get a duplicate effective (DE) model when revising a 2-D model. Sometimes this is due to a versioning issue. Is a DE model needed if you are revising a 1-D study using a 2-D model?
35. If flooding is contained in the street, can it be mapped as Shaded X?
36. If the depth is averaged, can detailed depth data from a 2-D study be used to evaluate the application?
37. Should there be a certificate program for modelers? If there is, what should it include? Who should administer the program?
38. Collecting field data. Field data collection for a 2-D study would be different than for a 1-D study. In addition to collecting data linearly downstream, data should be collected across the horizontal extent of the floodplain.
39. How do you ensure that the modeler’s assumptions are fully documented?
40. Should FEMA give more latitude to communities that are actively advancing or innovating in the area of 2-D studies?
41. Should there be an on-line forum?
42. Can FEMA establish an open source 2-D software system that anyone can adapt and improve upon? Something in line with Linux system?
43. Can FEMA collaborate with other agencies to develop common set of guidelines, or is there a way for the agencies to provide input?
44. How can FEMA provide opportunities for communities to provide input and comment on new 2-D policies or procedures?
45. “If it’s not too deep, it’s 2D.” – Canadian colleague
46. All equations and models are approximations: 1-D, 2-D or 3-D. Must understand how Manning’s values vary from 1-D to 2-D. In 2-D, Manning’s is a
FUNCTION OF THE CELL/GRAID RATHER THAN (AVERAGE FOR) THE ENTIRE CROSS-SECTION (AS IN 1-D). 1-D – WILL GET SIMILAR ANSWERS FROM DIFFERENT FOLKS; 2-D – WILL LIKELY GET DIFFERENT RESULTS BECAUSE WE DIFFER ON REALLY SMALL INPUTS (LIKE MANNING’S) WHICH COULD DRASTICALLY CHANGE THE RESULTS (HAVE TO KNOW WHAT’S GOING ON IN FIELD FOR 2-D BECAUSE IT IS CALCULATED AT THE CELL LEVEL). ..................................................... 64

CHAPTER 4: NUMERICAL MODEL VERIFICATION .......................................................... 65
1. INTRODUCTION ........................................................................................................... 65
2. VERIFICATION CRITERIA FOR NUMERICAL SOLVERS .......................................... 65
3. MESH-CONVERGENCE AS A VERIFICATION METHOD IN COMPUTATIONAL HYDRAULICS .............................................................. 66
4. AMBIGUITY IN THE MESH CONVERGENCE TEST .................................................. 68
5. ERROR MEASURES AND THEIR QUALIFICATION ..................................................... 70
6. OBSERVED ORDER OF ACCURACY .............................................................................. 70
7. VAVUQ: A TOOLK FOR VERIFICATION AND VALIDATION, AND UNCERTAINTY
   QUANTIFICATION IN ENVIRONMENTAL-FLUID-MECHANICS SIMULATIONS ............... 71
8. EXAMPLE ONE: RICHARDSON EXTRAPOLATION ..................................................... 72
9. EXAMPLE TWO: MESH CONVERGENCE TEST VERSUS A KNOWN BENCHMARK (MES) .................................................................................. 73
10. EXAMPLE THREE: QUANTITATIVE MODEL SKILL ASSESSMENT (VALIDATION) ....... 74
11. EXAMPLE FOUR: VISUALIZATION OF THE NUMERICAL RESULTS’ DIFFERENCE IN
    VARIOUS MESHES ........................................................................................................ 76
12. EXAMPLE FIVE: NUMERICAL UNCERTAINTY QUANTIFICATION ................................ 76

CHAPTER 5: QUESTION FORM FOR 2-D SOFTWARE VENDORS ..................................... 78

CHAPTER 6: 2-D FLOOD MODELING RECOMMENDATIONS ........................................ 79
1. 2-D MODELS .................................................................................................................. 79
2. QUALIFICATIONS FOR CONSULTANT PREPARING THE MODEL ................................ 79
3. FEMA MODEL REVIEW AND MODEL DOCUMENTATION .......................................... 80
4. SOFTWARE VERSION CHANGES AND MODEL RESULTS ........................................ 80
5. FLOOD MAPS ................................................................................................................. 81

CHAPTER 7: REFERENCES ................................................................................................. 82

CHAPTER 8: APPENDIX ..................................................................................................... 87
1. GARY BRUNNER’S WHITE PAPER FOR NATIONAL WEATHER SERVICE .................... 87
2. ENGINEERS AUSTRALIA, 2012 .................................................................................... 91
3. OPEN CHANNEL FLOW RESISTANCE, BEN YEN, 2002 ........................................... 94
4. SCALING AND SIMILARITY IN ROUGH CHANNEL FLOWS, G. GIOIA AND F.A.
   BOMBARDELLI ............................................................................................................ 95
5. SNAPSHOT FROM VAVUQ GUI FOR VERIFICATION, VALIDATION AND UNCERTAINTY
   QUANTIFICATION ....................................................................................................... 99
Acknowledgments

We are greatly thankful to the Federal Emergency Management Agency (FEMA) and California Department of Water Resources (DWR) for providing the funding and making the completion of this report possible. Also, we would like to thank the Blue Ribbon Panel (BRP) for taking the time out of their busy schedules to provide their contributions and share their expertise. The BRP includes the following members: Gary Brunner, Hydrological Engineering Center (HEC), US Army Corps of Engineers; Shyamal Chowdhury, Wood Rodgers; Brad Hall, Northwest Hydraulic Consultants (nhc); M. D. Haque, Department of Water Resources of California; Pal Hegedus, RBF Consultants; Thomas Plummer, Civil Solutions; Kathleen Schaefer, FEMA; Peter Smith, U.S. Geological Survey (retired); and Zhida Song-James, Baker Corp.
Chapter 1: Introduction

On September 4, 2012, FEMA, the Federal Emergency Management Agency, in collaboration with the Floodplain Management Association (FMA), hosted a 1-day Symposium on the use of 2-D modeling for flood mapping purposes. Symposium participants identified a number of issues related to FEMA’s use of 2-D models in the administration of the National Flood Insurance Program. After the FMA Symposium, FEMA assembled a team of participants that consolidated the discussion topics into 46 specific issues associated with the use of 2-D models in flood studies. Simultaneously, FEMA contracted with the University of California, Davis (UCD), Department of Civil and Environmental Engineering, to: a) develop a literature review on 2-D modeling used in floodplain mapping; b) constitute a Blue Ribbon Panel (BRP) of experts to conduct an in-depth technical review of the 46 issues; c) produce a white paper to contribute to the general understanding of the assumptions, limitations, and capabilities of 2-D models for the determination of floodplain areas, thereby improving the quality of the application of 2-D models; and d) produce written guidelines that address limitations and capabilities of 2-D models to help practitioners, decision and policy makers use 2-D models more effectively.

The assembled BRP for the project consisted of members from academia, federal and state agencies, and practicing engineers. The BRP included the following members: Gary Brunner, Hydrological Engineering Center (HEC), US Army Corps of Engineers; Shyamal Chowdhury, Wood Rodgers; Brad Hall, Northwest Hydraulic Consultants (nhc); M. D. Haque, Department of Water Resources of California; Pal Hegedus, RBF Consultants; Thomas Plummer, Civil Solutions; Kathleen Schaefer, FEMA; Peter Smith, U.S. Geological Survey (retired); Zhida Song-James, Baker Corp; Fabián Bombardelli, University of California, Davis; and William Fleenor, University of California, Davis.

The BRP met on two separate occasions at the Department of Civil and Environmental Engineering of the UCD, on July 22, and September 18, 2013. In addition, UCD participated in the organization of a session during the 2013 FMA conference (September 3), in Anaheim, where Prof. Bombardelli and Dr. Fleenor presented the progress with the project and several members of the BRP provided their comments on the 46 issues to the audience.

In general, there were some topics which surfaced and re-surfaced during the multiple discussions. One of those topics involved the decision on model complexity for a given case, which should be based on two main ideas: a) the goal of the analysis, and b) the modeler. It was stressed during the UCD meetings and during the FMA conference that the modeler cannot be taken out of the model result, emphasizing the need for having a good modeler to implement and use the modeling tools. Thus, the steps to follow when facing a modeling task should be: a) define the goal of the study; b) select the adequate modeler for the task; c) chose the right tool for the goal of the study and the modeler; and d) perform the study considering all conditions.
This work reports the activities developed as part of the project described. The words are mainly those of the authors of the report, and the members of the BRP are NOT responsible for any error(s) or mistake(s) of this manuscript.

It is relevant to mention at this point that in 1968, the U.S. Congress created the National Flood Insurance Program (NFIP), to help property owners protect their property from floods. Communities can participate in the NFIP after establishing an agreement with FEMA. The agreement states that “if a community will adopt and enforce a floodplain management ordinance to reduce future flood risks to new construction in high-risk areas known as Special Flood Hazard Areas, SFHAs, the Federal Government will make flood insurance available within the community as a financial protection against flood losses.” Participating communities agree to adopt and enforce ordinances that meet or exceed FEMA requirements to reduce the risk of flooding (http://www.fema.gov/national-flood-insurance-program).

Under the Emergency Phase of the NFIP, FEMA issued Flood Hazard Boundary Maps, or FBHMs, to 19,000 flood-prone communities. The FBHMs provided areas subject to inundation by the base flood, the 100-year flood. As flood hazard information and mapping was updated and modernized, FEMA helped and worked with communities to become Regular Phase participants. For the Regular Phase participants, FEMA conducted Flood Insurance Studies, FISs, which were based off a more detailed engineering analysis. The milestones that the community and FEMA had to meet to participate in the Regular Phase of the NFIP are shown in the figure below:

![Figure 1: Milestones for Participation in Regular Phase of NFIP. Source: http://www.fema.gov/media-library-data/20130726-1727-25045-6497/mitdiv12chpt1_dec09.pdf](http://www.fema.gov/media-library-data/20130726-1727-25045-6497/mitdiv12chpt1_dec09.pdf)

FEMA redefined the Special Flood Hazard Area boundaries with the updated study and presented the results on a Flood Insurance Rate Map, FIRM. The FIRM displayed the
100-year and 500-year flood zone boundaries, flood insurance risk zones, and base flood elevations (BFEs).

New FIRMs are created when a new community plans to participate in the NFIP and existing FIRMs are updated to better represent the community’s flooded areas (either due to better data or better engineering analysis). To develop a new FIRM or update an existing FIRM, the following four phases are needed: 1) Mapping Needs Assessment, 2) Project Scoping, 3) Topographic and Flood Hazard Data Development/Report and 4) Map Production, and Preliminary/Post-Processing. The Mapping Needs Assessment consists of evaluating that the flood hazard data on the FIRM are adequate and, if it is not, the community will identify the specific data elements that need to be updated to FEMA. This serves as the building block for the Project Scoping phase, because it determines what flood map projects need to be initiated. The Project Scoping phase consists of conducting background research and community outreach, determining the flood hazard data that will be used, establishing priority levels for flooding sources to be analyzed and mapped, developing schedules and cost estimates, assigning project tasks to Mapping Partners, and developing contracts for completion of assigned work. The Topographic and Flood Hazard Data Development/Report and Map Production phase includes developing and obtaining topographic data and cross-section data for engineering analysis, performing such engineering analysis, preparing base map for FIRM, digitizing from the previous FIRM those boundaries that are not being updated, merging new/updated FIRM with previous FIRM, and producing/revising the FIS report. Upon completion of the new/updated FIRM and FIS, the Preliminary/Post-Processing phase begins. During this phase, FEMA sends the new/updated FIRM and FIS report to officials of the affected communities for preliminary review. Here, FEMA provides the community the opportunity to comment on the FIRM and FIS report through formal public meetings. The comments are considered in the final FIRM and FIS report that FEMA publishes. When required, FEMA also initiates a 90-day appeal period to provide community officials and citizens an opportunity to appeal any new/modified FIRM and FIS report. FEMA considers and evaluates all comments in cooperation with the community. FEMA then does a final quality assurance/control (QA/QC) to ensure all information in the FIRM and the FIS report are adequate and in compliance with FEMA guidelines. Upon successful completion of the QA/QC process, the U.S. Government Printing Office (GPO) and the FEMA Map Service Center (MSC) print and distribute the FIRM and FIS report (http://www.fema.gov/media-library-data/20130726-1543-20490-1120/frm_gsv1.pdf).

In July 2012, the U.S. Congress passed the Biggert-Waters Flood Insurance Reform Act (BW-12). BW-12 proposes a number of changes to the way the NFIP is run. Some changes will require the NFIP to raise rates to reflect true flood risk, make the program more financially stable, and change how Flood Insurance Rate Map (FIRM) updates impact policyholders (http://www.fema.gov/national-flood-insurance-program).
Chapter 2: Literature Review

Numerical modeling is the standard tool used in the determination of floodplain areas for engineering and insurance purposes. For many years one-dimensional (1-D) models have been primarily used for this purpose, with HEC-RAS arguably the most widely used 1-D model. Relatively robust guidelines and specifications, as well as an industry standard of practice, currently exist for 1-D models. While 1-D models are relatively simple to apply and rather inexpensive to develop, they are based on several assumptions that do not always hold true in overbank flooded conditions. Those limitations constitute particularly significant shortcomings for models intended to simulate flow over levees or in areas that are very flat, such as those found in the Central Valley of California.

In recent years 2-D models have been incorporated as an additional tool in floodplain mapping in order to allow for more detailed analyses. Unfortunately, the assumptions behind these 2-D models are not fully understood by all users. 2-D models are even less understood by decision and policy makers. The Federal Emergency Management Agency (FEMA) has recognized the need for greater understanding and improved guidelines for applying 2-D models in flood studies and intends to widely disseminate the advantages and limitations of 2-D models.

The following will review available scientific and engineering literature on the application of 1-D and 2-D models in flood studies and literature regarding existing 2-D model guides and specifications as applied to flood studies.

1. 1-D and 2-D Equations

One of the most important aspects of a flood model is the equations used in the software model. 1-D flood models typically use finite-differences, finite-volumes, or finite-elements solutions of the full Saint-Venant Equations (Bates and De Roo, 2000). The Saint-Venant Equations express both continuity and momentum which are obtained by cross sectionally-averaging the Navier-Stokes Equations after turbulence averaging (Wu, 2007):

\[ \frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0 \tag{1} \]

\[ \frac{\partial Q}{\partial t} + \frac{\partial}{\partial x} (uQ) + gA \left( \frac{\partial h}{\partial x} - S_0 \right) + gAS_f = 0 \tag{2} \]

where \( Q \) is the discharge, \( A \) is the cross-sectional area, \( S_0 \) is the bed slope, \( h \) is flow depth, \( x \) is the longitudinal distance, \( t \) is time, and \( S_f \) is the friction slope. The terms included in the momentum equation (Equation 2) above are: unsteadiness or local time variance of flow; spatial flow variation; pressure water level gradient; component of weight and friction forces, respectively. The first two terms are often called inertia terms. Assumptions embedded in the 1-D Saint-Venant Equations are that the pressure distribution is hydrostatic, the resistance relationship for unsteady flow is the same for steady flow, and the bed slope is sufficiently mild such that the cosine of the slope can be
replaced by unity (Stelling and Verwey, 2005). Some models use a reduced form of the momentum equation shown above. One of the reduced forms is the Kinematic Wave equation, which can be used when the energy grade line is quasi parallel to the channel bottom as shown below:

$$S_o = S_f$$  \hspace{1cm} (3)

Another reduced form of the momentum equation is the non-inertia wave equation, in which the local and convective inertia terms are ignored and is shown below:

$$gA \left( \frac{\partial h}{\partial x} - S_o \right) + gAS_f = 0$$  \hspace{1cm} (4)

MIKE11 uses a more general form of the cross sectionally-averaged Navier-Stokes Equations, and uses either the Chezy coefficient, $C$, or Manning’s $n$ to parameterize the bottom resistance (DHI, 2009).

The same principles used to derive the 1-D Saint-Venant Equations by an integration over the whole the cross section are used to derive the 2-D equations by integration over the depth, whose final forms are expressed below:

$$\frac{\partial h}{\partial t} + \frac{\partial (hU)}{\partial x} + \frac{\partial (hV)}{\partial y} = 0$$  \hspace{1cm} (5)

$$\frac{\partial (hU)}{\partial t} + \frac{\partial (hUU)}{\partial x} + \frac{\partial (hUV)}{\partial y} = \frac{\partial (h T_{xx})}{\partial x} + \frac{\partial (h T_{xy})}{\partial y} - gh \frac{\partial z}{\partial x} - \frac{\tau_{hx}}{\rho}$$  \hspace{1cm} (6)

$$\frac{\partial (hV)}{\partial t} + \frac{\partial (hUV)}{\partial x} + \frac{\partial (hVV)}{\partial y} = \frac{\partial (h T_{xy})}{\partial x} + \frac{\partial (h T_{yy})}{\partial y} - gh \frac{\partial z}{\partial x} - \frac{\tau_{hy}}{\rho}$$  \hspace{1cm} (7)

where $h$ is again the water depth; $U$ and $V$ are the depth-averaged velocity components in the $x$ and $y$ directions; respectively; $T_{xx}$, $T_{xy}$, and $T_{yy}$, are depth-averaged turbulent stresses; $z$ is the water surface elevation; and $\tau_{hx}$, $\tau_{hy}$ are the bed shear stresses due to friction (Moore, 2011). The terms above are: unsteadiness or local time variation of flow; spatial flow variation; shear stresses; pressure gradients and component of weight and friction. Shear stresses have an important role in the numerical solution of the equations.

2-D approaches assume that the lateral boundary layers are negligible. Whereas this is a good approximation in most studies, the modeler needs to check that this is the case for the application under analysis.
2. 1-D or 2-D

One of the most important aspects of flood studies is determining whether to use a 1-D steady-state, 1-D unsteady-state, 2-D steady-state, or 2-D unsteady-state model. Decision between unsteady-state vs. steady-state model solutions is more of an issue when it comes to smaller streams/rivers with presence of hydraulic structures. The issue would occur, for example, at a culvert or bridge opening where the culvert could not handle the 100-year flood. The water would flow over the embankment and into the road. One would have to use an unsteady-state model in this case, as the steady-state model will obviously produce erroneous descriptions of the flow. The modeler should have a clear physical insight on the way the river channels, culverts, floodplain areas, levees, and roads respond during floods. Also, the modeler needs to know the size, length, and complexity of the system that will be modeled. Lastly, the modeler needs to determine the purpose of the study and the expected level of accuracy before selecting any model (Brunner, 2010).

When no constrictions are present in the flow, 1-D steady-state modeling is by nature more conservative because the peak of the flood remains constant throughout the simulation; 1-D steady-state modeling is simpler to conduct, and requires lower level hydrology data, although it is often less accurate when applied to complex flooding areas (Hosseingipour et al., 2012). In addition, most communities have engineers who likely know the use of 1-D models such as HEC-RAS from the U.S. Army Corps of Engineers. (We stress herein that knowledge of a particular model is not enough justification for its use when clearly it should not be applied). In 1-D models, floodplains are described by series of cross sections perpendicular to the flow direction and water levels in the main channel and floodplain are the same in steady state (Frank et al., 2001) (This is clearly not true when the flood is rising or receding). Steady state models should not be used when flood events are very dynamic and levees overtop or breach.

In some cases 1-D models can produce the equivalent results as 2-D models when the dominant flow directions of a river or floodplain follow the general river flow path. It is also feasible to use a 1-D model when a levee is overtopped or breached and the protected area is small enough that it is known to fill to a leveled surface elevation (Brunner, 2010). On the other hand, 1-D models should not be used when a levee will be overtopped or breached and the water can go in many directions (larger protected areas). In addition, 1-D models are not good for bays and estuaries where the flow will go in multiple directions due to tidal fluctuations. 1-D models are also not appropriate to use when significant lateral elevation differences occur due to the flow around sudden bends (Brunner, 2010).

1-D models consist of series of linked channels with discrete cross-sections at regular intervals and output contains water level, depth and velocity. Advantages of 1-D models include being relatively fast to run, easy to modify, and result files are relatively small. Disadvantages of 1-D models are that they require of cross-sections to be input by field survey or Digital Elevation Model (DEM) and thus it can be prone of smoothing out of DEM data. 2-D models feature detailed grid or mesh-based topography with element
resolutions for an urban environment typically ranging from 1 m to 10 m. For more extensive floodplain environments, element resolution can typically range from 10 m to 100 m. Outputs include water level, depth and velocity. Advantages of 2-D models are less interpolation of results required and are more readily linked to GIS; the modeler is not required to identify flowpaths in advance, modeler can model complex flowpaths; inputs and outputs defined spatially in GIS type environments, results in better data continuity and more readily accessible/understandable results for the community/client (Engineers Australia, 2012). Disadvantages of 2-D models are they require detailed grid/mesh to be interpolated from aerial and/or field survey based DEM (plus roughness mapping over catchment); they can be time consuming to build; modifications often not as easy as for 1-D; they are relatively slower to run; files are large and can in some cases provide overconfidence in the results that may not be justified if the underlying data are inadequate (Engineers Australia, 2012).

3. 1-D/2-D Coupled

It is sometimes convenient to combine 1-D within 2-D approaches to save computational time in those streams where a 2-D description is not necessary. The capabilities for doing so are obviously software dependent.

To take advantage of both 1-D and 2-D models, it is common to simulate the stream channel, where flow is typically longitudinal, using a 1-D model, and model floodplain flow in 2-D (Moore, 2011). In this way, the best features of both approaches are combined and used where they are mostly efficient and accurate.

Transfer of flow between the channel and main bank is an important factor when dealing with 1-D/2-D coupled models. Many software models link the 1-D river to the 2-D floodplain in very different ways. This can result in significantly different predictions of the volume of water exchanged between the river and the floodplain. Further research needs to be done to better understand models that use 1-D/2-D model linking (Environment Agency, UK, 2013).

4. Data

Another aspect that needs to be considered when choosing a software model is the available data for the flood study. Data requirements for flood inundation models fall into four groups: 1) topographic data of the channel and floodplain (model bathymetry), 2) time series of flow rates and stage data for model input and output boundary conditions, 3) roughness coefficients for channel and floodplain, and 4) data for model calibration and validation (Mason et al., 2010). In what follows, we discuss those data requirements in detail, with the exception of roughness, and to each we devote a separate section.

The data requirement for topography is a high quality Digital Terrain Model (DTM) to represent the ground surface. For rural floodplain modeling, a DTM should have a vertical accuracy of about 0.5 m and a spatial resolution of at least 10 m (Ramsbottom
and Wicks, 2003). Although the level of resolution needed varies from case to case, Horritt and Bates state that: “Whilst this level of accuracy and spatial scale is insufficient to represent the micro-topography of relict channels and drainage ditches existing on the floodplain that control its initial wetting, at higher flood depths inundation is controlled mainly by the larger scale valley morphology, and detailed knowledge of the micro-topography becomes less critical.” Embankments and levees controlling overbank flows require approximately 10 cm vertical accuracy and 2 m spatial resolution is needed (Smith et al., 2006). Urban floodplains require a vertical accuracy of 5 cm with a spatial resolution of 0.5m is needed to resolve gaps between buildings (Smith et al., 2006).

Discharge and stage data are another flood model data requirement which help provide the model boundary conditions. The data are acquired from gauging stations 10-60 kilometers apart on the river network. Modelers ideally require gauged flow rates to be 95% accurate (Mason et al., 2010). They state that: “However, problems with the rating curve extrapolation to high flows and gauge bypassing may mean discharge measurement errors may be much higher than this acceptable value during floods. At such times gauged flow rates are likely only to be accurate to 10% at best, and at many sites errors of 20% will be much more common. At a few sites where the gauge installation is significantly bypassed at high flow errors may even be as large as 50%.”

The final data requirement for flood models is measured data for calibration and validation. If a model can be successfully validated using independent data, it gives confidence in its predictions for future events. Validation data for hydraulic models consist most of the time of bulk flow measurements taken at small number of points in the model domain. Since flow measurements were taken at a small number of points in the model domain, proper validation requires more dedicated efforts. Mason et al. (2010) state: “The 2D nature of modern distributed models requires spatially distributed observational data at a scale commensurate with model predictions for successful validation. The observations may be synoptic maps of inundation extent, water depth or flow velocity. If sequences of such observations can be acquired over the course of a flood event, this allows the possibility to further improve model predictions.”

The available data used in a flood study can be a major deciding factor in whether to use a 1-D or 2-D model. Naturally, unsteady–state, 1-D and 2-D modeling approaches require both a higher level of input hydrologic data (time series of discharges or hydrographs, possibly at several locations), higher levels of output analysis, and more rigorous model calibration, as opposed to the steady-state counterparts (Hosseinpour et al., 2012). Data availability required to calibrate and validate a model is variable depending on the community where the study is being developed; the data may be absent in many cases and unfortunately most flooding models applied recently lack calibration. It is easy to understand that coverage of data is never “enough,” but that models need to be supported by data. When data are available, it may be that the data are old, the river system has changed, or maybe the available data are insufficient or with inadequate level of detail because the purpose of the study has changed. All modelers need good channel data to conduct a satisfactory study.
Not having detailed enough terrain data to undertake the modeling effort is a major concern for 2-D models. If the data do not exist, it is not recommended to use a 2-D model because the model effort will not provide improved results over a 1-D representation.

5. 1-D and 2-D Models

<table>
<thead>
<tr>
<th>Software Model Name</th>
<th>Description (retrieved directly from the software model website)</th>
<th>Vendor</th>
<th>Cost ($ unless otherwise specified)</th>
<th>Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>RMA2</td>
<td>Two-dimensional depth-averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields. RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning’s or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady state (dynamic) problems can be analyzed.</td>
<td>USACE</td>
<td>Public Domain</td>
<td><a href="http://chl.erdc.usace.army.mil/rma2">http://chl.erdc.usace.army.mil/rma2</a></td>
</tr>
<tr>
<td>Software</td>
<td>Description</td>
<td>Author</td>
<td>License</td>
<td>Website</td>
</tr>
<tr>
<td>----------</td>
<td>-------------</td>
<td>--------</td>
<td>---------</td>
<td>---------</td>
</tr>
<tr>
<td>HIVEL2D</td>
<td>A free-surface, depth-averaged, two-dimensional finite element model designed specifically to simulate flow in typical high-velocity channels. HIVEL2D is applicable for flow fields that contain supercritical and subcritical regimes as well as the transitions between the regimes. HIVEL2D provides numerically-stable solutions of advection-dominated flow fields containing shocks such as oblique standing waves and hydraulic jumps. The system of nonlinear equations is solved using the Newton-Raphson iterative method. The Newton-Raphson method was selected for this model because the nonlinear terms in high-velocity channel flow fields are quite significant. Stresses are modeled using the Manning’s formulation for boundary drag and the Boussinesq relation for Reynolds stresses. Eddy viscosities are approximated using an empirical relation based on Manning’s coefficient and local flow variables.</td>
<td>Scientific Software Group</td>
<td>?</td>
<td><a href="http://www.scisoft-gms.com/sms_details/sms_hivel2d/sms_hivel2d.html">http://www.scisoft-gms.com/sms_details/sms_hivel2d/sms_hivel2d.html</a></td>
</tr>
<tr>
<td>FESWMS-2DH</td>
<td>A modular set of computer programs that simulates two-dimensional, depth-integrated, surface-water flows. FESWMS-2DH consists of an input data preparation program (DINMOD), flow model (FLOMOD), simulation output analysis program (ANOMOD), and graphics conversion program (HPLOT).</td>
<td>USGS</td>
<td>Public Domain</td>
<td><a href="http://water.usgs.gov/software/FESWMS-2DH/">http://water.usgs.gov/software/FESWMS-2DH/</a></td>
</tr>
<tr>
<td>TUFLOW</td>
<td>TUFLOW (finite difference) is a computational engine that provides one-dimensional (1D) and two-dimensional (2D) solutions of the free-surface flow equations to simulate flood and tidal wave propagation. TUFLOW also has abilities for 1D/2D linking, flexibility, robustness and a range of other features.</td>
<td>BMT Group Ltd.</td>
<td>6000</td>
<td><a href="http://www.tuflow.com/Tuflow.aspx">http://www.tuflow.com/Tuflow.aspx</a></td>
</tr>
<tr>
<td><strong>TUFW FV</strong></td>
<td>A flexible mesh finite volume numerical model that simulates hydrodynamic, sediment transport and water quality processes in oceans, coastal waters, estuaries and rivers. Modules available include: 2D HD: the standard hydrodynamic engine AD: an advection dispersion module ST: a mud and sand transport module Advanced structures for floodplain application WQ: water quality processes 3D: all of the above in 3D, with option for automatic 3D/2D transition for floodplain applications.</td>
<td>BMT Group Ltd.</td>
<td>6000</td>
<td><a href="http://www.tuflow.com/Tuflow%20FV.aspx">http://www.tuflow.com/Tuflow%20FV.aspx</a></td>
</tr>
<tr>
<td><strong>FLO-2D Basic</strong></td>
<td>FLO-2D Basic is essentially the v2009 model updated and simplified for free distribution. Applications have been approved for FEMA FIS studies. It is a good tool for demonstrations, students, training and FIS studies. A basic license is free but does not include training, webinars or technical support. FLO-2D Basic can tackle many diverse flooding problems including: River overbank flooding Unconfined alluvial fan flows Urban flooding with street flow, flow obstruction and storage loss Overland progression of tsunami and hurricane storm surges. Watershed rainfall and runoff Flood insurance studies Flood mitigation design</td>
<td>FLO-2D Software</td>
<td>FREE</td>
<td><a href="http://www.flo-2d.com/flo-2d-pro/">http://www.flo-2d.com/flo-2d-pro/</a></td>
</tr>
</tbody>
</table>

FLO-2D is a combined hydrologic and hydraulic model so there is no need to separate rainfall/runoff and flood routing. FLO-2D is a FEMA approved hydraulic model for riverine studies and unconfined flood analyses.
<table>
<thead>
<tr>
<th>Software</th>
<th>Description</th>
<th>Price</th>
<th>Website</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLO-2D Pro</td>
<td>FLO-2D is a flood routing model that simulates river, alluvial fan, urban and coastal flooding. FLO-2D can tackle any diverse flooding problems including: River overbank flooding Unconfined alluvial fan flows Urban flooding with street flow, flow obstruction and storage loss Overland progression of tsunami and hurricane storm surges Mud and debris flows Watershed rainfall and runoff Surface and groundwater interaction Flood insurance studies Flood mitigation design NWS Breach embedded EPA-SWMM Interface FLO-2D is a combined hydrologic and hydraulic model so there is no need to separate rainfall/runoff and flood routing. FLO-2D is a FEMA approved hydraulic model for riverine studies and unconfined flood analyses.</td>
<td>995</td>
<td><a href="http://www.flo-2d.com/flo-2d-pro/">http://www.flo-2d.com/flo-2d-pro/</a></td>
</tr>
<tr>
<td>RIVER FLO-2D Plus</td>
<td>RiverFLO-2D Plus is a breakthrough in 2D hydraulic modeling, offering a high performance finite-volume engine for faster, accurate and volume conservative computations in all flow situations. RiverFLO-2D Plus results achieve zero volume conservation errors (&lt; 10^-10 % Error) thanks to the new model engine based Fourth-Generation Finite-Volume algorithm which offers unmatchable stability and ease of use, and a new wetting-drying method handles the most demanding topography.</td>
<td>6970</td>
<td><a href="http://www.hydronia.net/two-dimensional-models/">http://www.hydronia.net/two-dimensional-models/</a></td>
</tr>
<tr>
<td>ANUGA</td>
<td>ANUGA is a Free &amp; Open Source Software (FOSS) package capable of modeling the impact of hydrological disasters such as dam breaks, riverine flooding, storm-surge or tsunamis. ANUGA is based on the Shallow Water Wave Equation discretized to unstructured triangular meshes using a finite-volumes numerical scheme. A major capability of ANUGA is that it can model the process of wetting and drying as water enters and leaves an area. This means that it is suitable for simulating water flow onto a beach or dry land and around structures such as buildings. ANUGA is also capable of modeling difficult flows involving shock waves and rapidly changing flow speed regimes (transitions from sub critical to super critical flows)</td>
<td>Australia National University (ANU) and Geoscience Australia (GA) SourceForge <a href="http://anuga.anu.edu.au/">http://anuga.anu.edu.au/</a></td>
<td>Public Domain</td>
</tr>
<tr>
<td>ISIS 2D</td>
<td>ISIS 2D has a fully hydrodynamic computational engine designed to work alone or with ISIS 1D, enabling dynamic interaction between 1D and 2D models. 1D and 2D models are linked through shapefiles specifying the model cells in the 2D domain to be linked to 1D model nodes. These shapefiles can be produced in standard software packages, or by specialist tools within ISIS MAPPER. Models can be linked by water level (levels computed by the 1D model are sent to the 2D model) or by flow (flows computed by the 1D model are sent to the 2D model). These linking methods allow ISIS 1D and ISIS 2D to represent lateral floodplains, a 1D channel running into a 2D estuary, spill over defences, and other representations of river, coastal or floodplain systems.</td>
<td>Halcrow</td>
<td>?</td>
</tr>
<tr>
<td>MIKE 11</td>
<td>MIKE FLOOD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------------------</td>
<td>-----------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MIKE 11 covers more application areas than any other river modelling package.</td>
<td>MIKE FLOOD is a toolbox for flood modelling. It includes a wide selection of 1D and 2D flood simulation engines, enabling you to model virtually any flood problem whether it involves rivers, floodplains, floods in streets, drainage networks, coastal areas, dam and levee breaches or any combination of the above.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Typical MIKE 11 applications:</td>
<td>Typical MIKE FLOOD applications:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flood analysis and flood alleviation design studies.</td>
<td>Rapid flood assessments.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Real time flood forecasting.</td>
<td>Flood hazard mapping.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dam break analysis.</td>
<td>Flood risk analysis for industrial, residential or cultural heritage areas.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Optimization of reservoir and canal gate / structure operations.</td>
<td>Flood contingency planning, for example planning of evacuation routes and priorities.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ecological and water quality assessments in rivers and wetlands.</td>
<td>Impact assessments of climate change issues.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sediment transport and river morphology studies.</td>
<td>Flood defense failure studies.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Salinity intrusion in rivers and estuaries.</td>
<td>Integrated urban drainage, river and coastal flood modelling.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wetland restoration studies.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DHI</td>
<td>9,500 Euros</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DHI</td>
<td>5,700 Euros (combination of size limited versions)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DHI</td>
<td>42,500 (comb. of unlimited versions, including MIKE 11, MIKE URBAN and 2D engines)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

http://mikebydhi.com/Products/WaterResources/MIKE11.aspx

http://mikebydhi.com/Products/WaterResources/MIKEFLOOD.aspx

<table>
<thead>
<tr>
<th>MIKE 21C</th>
<th>SOBEK</th>
</tr>
</thead>
<tbody>
<tr>
<td>MIKE 21C is a tool for simulating the development in the river bed and channel plan form caused by changes in the hydraulic regime. Simulated processes include bank erosion, scouring and shoaling brought about by activities such as construction and dredging or seasonal fluctuations in flows. Typical MIKE 21C applications: Design protection schemes against bank erosion. Evaluate measures to reduce or manage shoaling. Analyze alignments and dimensions of navigation channels for minimizing maintenance dredging. Predict sedimentation of water intakes, outlets, locks, harbors and reservoirs. Forecast the impact of bridge, tunnel and pipeline crossings on river channel hydraulics and morphology. Optimize restoration plans for habitat environment in channel floodplain systems.</td>
<td>SOBEK is a powerful modeling suite for flood forecasting, optimization of drainage systems, control of irrigation systems, sewer overflow design, river morphology, salt intrusion and surface water quality. The programs within the SOBEK modeling suite simulate the complex flows and the water related processes in almost any system. The programs represent phenomena and physical processes in an accurate way in one-dimensional (1D) network systems and on two-dimensional (2D) horizontal grids. It has been developed - and is being further developed - jointly with Dutch public institutes and governmental organizations, research institutes, universities and private consultants all over the world. Integrated approach SOBEK offers one software environment for the simulation of all management problems in the areas of river and estuarine systems, drainage and irrigation systems and wastewater and storm water systems. This allows for combinations of flow in closed conduits, open channels, rivers overland flows, as well as a variety of hydraulic, hydrological and morphological processes.</td>
</tr>
</tbody>
</table>
environmental processes.

A powerful hydrodynamic 1D/2D simulation engine
The hydrodynamic 1D/2D simulation engine is the computational core of SOBEK. This engine is used in all D-Flow programs within SOBEK modeling suite. Thus allowing the combined simulation of pipe, river-, channel- and overland flow through an implicit coupling of 1D and 2D flow equations. SOBEK is a tool for studying the effects of dam breaks, river floods, dike breaches, urban flooding etc.

Robustness of numerical operations
The hydrodynamic 1D/2D simulation engine is equipped with a very robust scheme for numerical computation. It also guarantees mass conservation, even in case of transitions through suddenly varying cross section shapes. The engine combines computations of subcritical and supercritical flow, at scales selected by the user. It handles flooding and drying of channels without the use of artificial methods such as the Preissmann slot.

Numerical efficiency
The hydrodynamic 1D/2D simulation engine has a very efficient numerical solution algorithm; this is based upon the optimum combination of a minimum connection search direct solver and the conjugate gradient method. It also applies a variable time step selector, which suppresses the waste of computational time wherever this is feasible.

Size of models
The size of the model is only limited by the size of the internal memory of the computer used.

<p>| InfoWorks ICM | InfoWorks ICM (Integrated Catchment Modeling) is the first truly integrated modeling platform to incorporate both urban and river catchments. With full integration of 1D and 2D hydrodynamic simulation techniques, both the above- and below-ground elements of catchments can be modeled to accurately represent all flow paths. InfoWorks ICM enables the hydraulics and hydrology of natural and man-made environments to be incorporated into a single model. | Innovyze | ? | <a href="http://www.innovyze.com/products/infoworks_icm/">http://www.innovyze.com/products/infoworks_icm/</a> |</p>
<table>
<thead>
<tr>
<th>Software</th>
<th>Description</th>
<th>Developer/Source</th>
<th>Website</th>
</tr>
</thead>
<tbody>
<tr>
<td>XPStorm</td>
<td><strong>xpstorm</strong> is a comprehensive software package for dynamic modeling of urban stormwater systems and river systems. It can develop link-node (1D) and spatially distributed (2D) hydraulic models for analysis and design. <strong>xpstorm.</strong> xpstorm simulates natural rainfall-runoff processes and the hydraulic performance of drainage systems and floodplains. It allows integrated analysis of flow in engineered and natural systems including ponds, rivers, lakes, overland floodplains and interaction with groundwater.</td>
<td>XP Solutions</td>
<td><a href="http://www.xpsolutions.com/software/xpstorm/">http://www.xpsolutions.com/software/xpstorm/</a></td>
</tr>
<tr>
<td>LISFLOOD</td>
<td>LISFLOOD is a GIS-based hydrological rainfall-runoff-routing model that is capable of simulating the hydrological processes that occur in a catchment. It can be used in large and transnational catchments for a variety of applications, including flood forecasting, and assessing the effects of river regulation measures, land-use change and climate change.</td>
<td>European Commission Joint Research Centre, Institute for Environment and Sustainability</td>
<td><a href="http://floods.jrc.ec.europa.eu/lisflood-model.html">http://floods.jrc.ec.europa.eu/lisflood-model.html</a></td>
</tr>
</tbody>
</table>
6. Modeler

Another equally important aspect in flood modeling is the modeler him/herself. The modeler needs to have deep knowledge of hydraulics, hydrology, as well as an in-depth knowledge of the local system. Engineers with greater knowledge, more experience and better training would likely be more successful in producing better results, regardless of the software that is being used. Also, the modeler should have experience with the software model they are using for the flood study, and understand the mathematics and disadvantages of the solution scheme.

7. Roughness Coefficients

The modeler also needs to have a full understanding of the roughness coefficients used in the model. The most widely used roughness coefficient is Manning’s $n$. Manning’s $n$ in Manning’s equation is based on empirical data of rivers around the world, but Manning’s formula has been obtained through theoretical means based on the phenomenological theory of turbulence (see Appendix 4: Gioia and Bombardelli, 2002). Manning's $n$ is less sensitive to changes in water levels than other roughness coefficients (Yen, 2002), and it does not vary too dramatically as the hydraulic radius changes, in comparison with other roughness coefficients. Manning's $n$ also changes in time as vegetation does in addition to the variability in space. Manning's $n$ values range from 0.02-0.15 for rivers (Chow, 1959).

In 1-D, $n$ is used for cross sections (the original assumption), whereas $n$ is used in 2-D for each mesh/grid element (cell). In most cases, Manning’s $n$ values are relatively close for 1-D and 2-D approaches. A few exceptions are where there are rapid changes in flow direction and magnitude (e.g., at a structure, sharp bend or embankment opening). Software that solves the full 2-D equations will model the loss of energy due to the water changing direction and magnitude through a change in roughness. Therefore, the $n$ value in a 1-D model at, for example, a sharp river bend may need to be higher than for the 2-D model to incorporate the losses (Engineers Australia, 2012).

Values of Manning’s $n$ (roughness) can be determined with the help of aerial or satellite imagery, textbooks, guidelines found in manuals, and site inspections (surveying).

Roughness appears in simplified or discrete numerical models to account for momentum and energy dissipation at wall boundaries, and it is a measure of flow resistance. Thus, it is needed for models of all dimensions. The representation of the flow resistance categories is, however, different for 1-D, 2-D, and 3-D approaches. In 1-D and 2-D models, roughness is more uncertain and less accurate in terms of the definition and sizing than in 3-D models (Knight et al., 2010). Flow resistance can be classified as skin drag, form drag, or shape drag. Skin drag is produced by “roughness” due to surface texture; form drag is “roughness” due to surface geometry, and shape drag is “roughness” due to overall channel shape (Knight et al., 2010).
In order to approximate the notion of roughness, simple experiments such as flow of liquids in circular pipes were used to gain knowledge on roughness in early times of fluid mechanics. The 1-D Darcy-Weisbach equation for flow in circular pipes relates channel geometry, flow, and pipe characteristics (relative roughness, $k_s/d$, where $k_s$ is the Nikuradse equivalent sand roughness size). The extension of this simple concept to compound channels is not simple. For instance, in open channels, the surface can be mobile, composed of loose boundary material such as sand or gravel, and/or flexible, composed of deformable material such as vegetation or instream weeds (Knight et al., 2010). In addition, channels can be compound where the cross-section is developed from multiple shaped elements. The Moody chart should not be used when specifying head losses in compound channels or complex shaped ducts because the hydraulic radius, $R$, is not a suitable characteristic geometric parameter, as is in the case of simple 1-D pipe flow. The wetted parameter, $P$, will have an unexpected change at the bank-full stage and the cross-sectional area, $A$, will not, which gives rise to discontinuous relationships in $f$ versus the $Re$ domain (Knight et al., 2010).

Unsteadiness in the flow affects the turbulence structure and as a result alters the bed shear and the resistance. Vegetation alongside the river channel also affects the resistance to flow. This is why seasonal growth/decay or managed weed cutting mean that vegetation will vary spatially. In alluvial fans, sediment has an effect on flow resistance. Alluvial resistance relationships are much more complex because geometry bed forms change with velocity and depth of flow (Knight et al., 2010). Also, some models include turbulence but some other models do not, so it is not correct to use the same friction factor for both 1-D and 2-D models (Knight et al., 2010).

As stated earlier, the 2-D shallow water equations are derived from vertical integration of the Navier-Stokes Equations, whereas 1-D models are obtained by integrating those equations over the 2-D area of the cross-section. Friction is characterized by using Manning’s $n$ or Chezy’s $C$. 1-D Manning’s $n$ and Chezy’s $C$ friction coefficients represent the shear stress exerted by the entire bed and banks bounding the flow while 2-D models use the friction factor to represent shear stress exerted at the base of a vertical column of water. In 3-D, a no-slip condition applies at the walls meaning the tangential and normal velocities to the wall are zero, whereas turbulence is modeled explicitly by the code. For 3-D models, the roughness appears in the boundary condition rather than as a term in the equations. Therefore, the impact on the solution is not as significant as in 1-D or 2-D models (Knight et al., 2010).

It is difficult to understand roughness without considering the numerical discretization of the model, the physical model, and the flow characteristics. The main difference for roughness is the amount of physics that each level of flow model includes (lateral/vertical velocity, density, and turbulence). Each approach for defining roughness coefficients has its own definition. Roughness implicitly means it is a function of the flow it represents and depends on the water depth as well. Modelers need to keep in mind that increasing the size of the roughness parameters has significant impacts on the algebraic equations of the solution scheme (Knight et al., 2010).
Below is a summary table of the Manning’s roughness coefficients used in the journal paper: “Transient Two-Dimensional Simulation of Real Flood Events in a Mediterranean Floodplain” (Gonzalez-Sanchis et al., 2012). The paper modeled a reach in the middle Ebro River, which is 12 kilometers downstream of Saragoza city (Northeast of Spain). The reach is 2 km long with an island and an oxbow lake.

<table>
<thead>
<tr>
<th>Description</th>
<th>n</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main Channel</td>
<td>0.035</td>
<td>Acrement and Schenides 1990</td>
</tr>
<tr>
<td>Urban</td>
<td>0.05</td>
<td>Sande va Der et al. 2003</td>
</tr>
<tr>
<td>Crop</td>
<td>0.035</td>
<td>Palmeri et al. 2002</td>
</tr>
<tr>
<td>Permanent water</td>
<td>0.024</td>
<td>Palmeri et al. 2002</td>
</tr>
<tr>
<td>Pine forest</td>
<td>0.124</td>
<td>Poole et al. 2004</td>
</tr>
<tr>
<td>Unsurfaced road</td>
<td>0.027</td>
<td>Chow 1959</td>
</tr>
<tr>
<td>Grassland</td>
<td>0.033</td>
<td>Palmeri et al. 2002</td>
</tr>
<tr>
<td>Popular crop</td>
<td>0.05</td>
<td>Martin Vide 2002</td>
</tr>
<tr>
<td>Gravel</td>
<td>0.028</td>
<td>Acrement and Schenides 1990</td>
</tr>
<tr>
<td>Cottonwood or willow dominant with gravel soil</td>
<td>0.1</td>
<td>Acrement and Schenides 1990</td>
</tr>
<tr>
<td>Cottonwood or willow dominant with sand soil</td>
<td>0.04</td>
<td>Acrement and Schenides 1990</td>
</tr>
<tr>
<td>Cottonwood or willow dominant &lt;2 m tall and &lt;5 cm diameter</td>
<td>0.13</td>
<td>Poole et al. 2004</td>
</tr>
<tr>
<td>Reed</td>
<td>0.13</td>
<td>Rhee et al. 2008</td>
</tr>
<tr>
<td>High grassland and disperse willow</td>
<td>0.124</td>
<td>Acrement and Schenides 1990</td>
</tr>
<tr>
<td>Mature forest with blackberry undergrowth</td>
<td>0.12</td>
<td>Poole et al. 2004</td>
</tr>
<tr>
<td>Old secondary channel</td>
<td>0.13</td>
<td>Poole et al. 2004</td>
</tr>
<tr>
<td>Dispersed cottonwood and Tamarix with gravel soil</td>
<td>0.036</td>
<td>Bediment and Huber 1988</td>
</tr>
<tr>
<td>Scrap</td>
<td>0.023</td>
<td>Chow 1959</td>
</tr>
</tbody>
</table>

Below is a summary table of the Manning’s roughness coefficients used and retrieved directly from Department of Water Resource’s Central Valley Floodplain Evaluation and Delineation Program (CVFED). The study area was the upper and lower Sacramento and San Joaquin River, which is composed of about 6,850 square miles. The models used were 1-D Model HEC-RAS for about 1800 miles of stream, 2-D Model FLO2D for about 6,850 square miles of overland, and 2-D Model TUFLOW for the Yuba River (DWR CVFED).
8. Numerical Stability

The modeler also needs to ensure that the developed flood model is stable. The stability of a model can be described by examining the Courant number \((Cr)\), through the apply-named Courant Criteria. The Courant number is defined as follows (Abbott, 1979; Engineers Australia, 2012):

\[
Cr = \frac{(v + \sqrt{g d}) \cdot dt}{dx}
\]  

(8)

“where \(v\) is velocity (m/s), \(g\) is gravity, \(d\) is depth, \(dt\) is the time step and \(dx\) is the grid/mesh dimension. Theoretically, it is only models with explicit solutions that must satisfy the criteria that \(Cr\) shall be no greater than one (1) in order to avoid “unstable” or inaccurate results. A fully implicit solution scheme should be able to utilize Courant numbers that are infinitely high provided that the time step is sufficiently small to depict the shape of the boundary time series. Semi-implicit schemes, such as those used for fixed grid finite difference solutions are able to run at \(Cr\) conditions greater than one but, are unlikely to produce stable and reliable results for \(Cr\) values greater than 10. Note that \(Cr\) varies throughout the model domain, so it is the areas that experience the highest \(Cr\) value that limit the time step. These areas are usually the deepest flow areas where the hydraulic friction from the bed of the model is at a minimum within the model domain. However, for implicit solutions, the actual Courant number can be much smaller in practice and may approach one (1), depending on the application. The introduction of 1D elements can introduce into an implicit model a \(Cr\) limitation much smaller than for the rest of the domain. In general, model stability and accuracy is compromised where the
Cr value is too high for the numerical scheme being used. On investigation of the Courant equation, this typically occurs when the velocity, depth or time step is too large in combination with an element size, which is relatively small. A key issue in the model development process is to gain an understanding of the time step that is likely to be required (and hence overall run time) for a specific mesh/grid resolution. This analysis will require an estimate of the likely modeled flow depth and velocities” (Engineers Australia, 2012).

Instabilities can occur at or near structures, sharp edges, or boundaries especially if the discretization is poor. The process for resolving the instability will vary between software and techniques. For example, a finer grid/mesh size might be needed in some cases (Engineers Australia, 2012).

9. Time Steps

Another important aspect of a flood model is the time step. There are generally two types of time steps, the fixed regular time step and the adaptive time step. The fixed regular time step allows the modeler to pre-determine the model run time and to set the saving step (times at which model results are saved) as a regular multiple of the simulation time step. To achieve stability of the model, the time step will need to be set at the shortest time interval during the most energetic or deepest flows during the simulation. This typically occurs for only a very short period of time during the peak of the flood hydrograph. The modeler needs to ensure that this simulation will complete during the run time (Engineers Australia, 2012).

The adaptive time step uses the Courant condition to determine the appropriate time step. The modeler sets a maximum and minimum time step allowable, which allows the model to use larger time steps, when flow is shallow and smaller time steps during peak of flow. This allows for the shortest time while maintaining model stability but can lead to very long run times. “This is due to the impact of a few minor locations in the model where short-lived energetic fluctuations in the flow can lead to the minimum time step being selected for excessively long periods of time. Run times can also become excessive if the period that it takes for the flood wave to propagate through the model is very long. For example, simulations of large river systems or of flat terrain where the critical rainfall duration is lengthy, will have propagation times in the order of days, if not weeks (Engineers Australia, 2012).”

The advantages of using an adaptive time step is that a longer time step can be used when flow is uniform and a smaller time step can used with rapid changes in flow. The disadvantages of using an adaptive time step is the actual simulation time is unknown a priori and when comparing different parameters the simulation time of each model will be unknown (Engineers Australia, 2012).

10. Calibration and Validation

Calibration is a crucial process in the development of a flood model. Calibration is the adjusting of model parameters in providing the best comparison between model outputs
and historical data. Validation is the simulation of the calibrated model with one of more independent data sets to demonstrate that the calibrated model is able to reproduce flood behavior (or past flood events). Calibration and validation ensures that the hydraulic model can represent the physical system with reliable results. It is as important for 2-D models as it is for 1-D models. Calibration and validation runs can be very time-consuming and costly but they cannot be avoided to produce a model with confident results (Engineers Australia, 2012).

The modeler should always consider the amount, type and quality of data from a given event and the magnitude of the flood event to properly compare it to the design event. For example, a small event confined to a certain location will behave differently than a larger event that has broken banks and flowing overland. The historical flood event should have sufficient flood observations, reliable topographic data, and boundary data meaning that recent flood events will be more complete (Engineers Australia, 2012).

Types of calibration and validation data are described as follows (Engineers Australia, 2012):

- Historical changes to topography, land-use, structures and drainage infrastructure.
- Records (photographs) of bed, bank and floodplain vegetation levels to assist with.
- Interpretation of roughness and provide record of prevailing conditions.
- Rainfall records (daily and pluviograph records), including in adjacent catchments.
- Gauged water level hydrographs, rating curves and derived flow hydrographs at stream.
- Gauge sites.
- Stream flow gauging at gauge sites and over the side of bridge structures (rare, but useful).
- Tidal level records if in a tidal area.
- Flood mark levels, location and measure of reliability. For example, debris marks.
- Watermarks on/in buildings.
- Descriptive anecdotal information and past reports of flood behavior in general.
- Observations of the rate of rise of flood waters and the time of peak.
- Photographs or videos of historical floods.
- Records or observations on water speeds and/or flow patterns.
- Records of blockage at hydraulic structures such as culverts and gully traps.
- Records and photography of the extent of inundation, noting time of the photos.
- Information on road/rail closures.

Anecdotal data can also be helpful for calibration and validation. It is important to calibrate and validate with the cooperation of local residents and stakeholders to see if they observed anything during the flood event. Also, if they noted any features such as flow directions, water speeds, or high water marks (Engineers Australia, 2012).
“It is far more important to understand why a model may not be calibrating a particular location than to use unrealistic parameter values to “force” the model to calibrate (Engineers Australia, 2012).” This is why sensitivity testing of inputs and parameter values is a good way to understand the input/parameter on model results (Engineers Australia, 2012).

11. Benchmark Tests

Benchmarking software models is a very important process in determining if the mathematical formulation of the physical processes controlling flood movement across a floodplain is accurate; and whether the numerical method used to solve the mathematical formulation and the configuration of the numerical grid upon which the numerical solution is applied is suitable (S. Neelz et al., 2012). Over time, scientific and technological progress has let modeling algorithms and tools develop and improve. The United Kingdom Environmental Agency conducted a series of benchmark tests on the latest 2-D models. The title of the report is *Benchmarking the latest generation of 2D hydraulic modeling packages* (S. Neelz and G. Pender, 2012). What follows is a summary of the benchmark tests and conclusions drawn from the benchmark exercises.

The UK Environmental Agency used 19 software model packages. The software models were ANUGA, Flowroute-iTM, InfoWorks ICM, ISIS 2D, ISIS 2D GPU, JFLOW+, MIKE FLOOD, SOBEK, TUFLOW, TUFLOW GPU, TUFLOW FV and XPSTORM which solve the full Shallow Water Equations (SWE). LISFLOOD-FP and RFSM EDA solves the SWE’s neglecting the advective acceleration term. ISIS Fast Dynamic solves the SWE without the acceleration term. ISIS Fast and RFSM Direct, which are based mainly on continuity and topographic connectivity, predict only final inundation with no variation in time (Environment Agency, UK, 2013).

There were a total of 10 benchmark tests used by the UK. A summary of each test is provided in the table below and taken directly from *Benchmarking the latest generation of 2D hydraulic modeling packages* (S. Neelz and G. Pender, 2012).

<p>| Table 4: Summary of Benchmark Tests by UK Environmental Agency |
|---------------|--------------------|------------------|
| <strong>Benchmark Test Number</strong> | <strong>Description</strong> | <strong>Purpose</strong> |
| 1 | Flooding a disconnected water body | Assess basic capability to simulate flooding of disconnected water bodies on floodplains or coastal areas |
| 2 | Filling of floodplain depressions | Tests capability to predict inundation extent and final flood depth for low momentum flow over complex topographies |
| 3 | Momentum conservation over a small (0.25 m) obstruction | Tests capability to simulate flow at relatively low depths over an obstruction with an adverse slope |</p>
<table>
<thead>
<tr>
<th></th>
<th>Description</th>
<th>Tests</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>Speed of flood propagation over an extended floodplain</td>
<td>Tests simulation of speed of propagation of flood wave and the prediction of velocities at the leading edge of the advancing flood</td>
</tr>
<tr>
<td>5</td>
<td>Valley Flooding</td>
<td>Tests simulation of major flood inundation at valley scale</td>
</tr>
<tr>
<td>6a and 6b</td>
<td>Dambreak</td>
<td>Tests Simulation of shocks and wake zone close to failing dam</td>
</tr>
<tr>
<td>7</td>
<td>River to floodplain linking</td>
<td>Evaluate capability to simulate flood transfer between rivers and floodplains using 1D to 2D coupled models</td>
</tr>
<tr>
<td>8a and 8b</td>
<td>Rainfall and sewer surcharge flood in urban areas</td>
<td>Tests capability to simulate shallow flows in urban areas within inputs from the rainfall (8a) and sewer surcharge (8b)</td>
</tr>
</tbody>
</table>

Areas where there is rapidly varying flow such as areas downstream of a dam failure, software that does not include all terms in the momentum conservation equation (LISFLOOD-FP, RFSM-EDA, and UIM) are not effective in modeling predictions. These models are also less robust in high velocity urban areas where there are supercritical flows (Environment Agency, UK, 2013).
Chapter 3: 1-D and 2-D Floodplain Modeling Issues

In what follows, are the results of the discussions at the BRP meetings are presented.

1. How does one decide when to use a 1-D steady state model versus a 1-D unsteady state model versus a 2-D model (Unsteady State)?

- What are the technical considerations in this decision?

Steady-state models represent the flood with a constant discharge throughout time, whereas unsteady-state models have the flood wave represented as a function of time in every detail. In 1-D models, the inherent assumption is that the water flow occurs mainly in one direction, whereas in 2-D models two horizontal coordinates are used to represent the flow velocity vectors; the vertical velocity is assumed to be negligible (Cook, 2008). Therefore, from a theoretical point of view, and given that all other conditions are met, an unsteady-state, 2-D model would produce a more realistic prediction of the flooding extent, because more details of the flood wave and flooding patterns can be represented with horizontal resolution. Cook and Merwade (2009) found that it is reasonable to assume that inundation extent could be more accurately predicted using 2-D simulations, as influences of topographic and geometric features are more accurately represented. However, this straightforward approach for modeling floods does not work well in all cases. In general, the simplest model that can solve the problem with accuracy should be the one selected. In what follows, some important considerations are provided, based on the discussions within the BRP, and based on a white paper developed by BRP member Gary Brunner for the National Weather Service, which is attached as Appendix 1.

First, models need to be compared against field data for both calibration and validation purposes and “ground truthing.” Naturally, unsteady–state, 1-D and 2-D modeling approaches require both a higher level of input hydrologic data (time series of discharges or hydrographs, possibly at several locations), higher levels of output analysis, and more rigorous model calibration, as opposed to the steady-state counterparts (Hosseingipour et al., 2012). Data availability required to calibrate and validate a model is variable depending on the domain where the study is being developed; the data may be absent in many cases and unfortunately most flooding models applied recently lack calibration and validation. It is easy to understand that coverage of data is never “enough,” but that models need to be supported by data. When data are available, it may be that the data are old, the river system has changed, or maybe the available data are insufficient or with inadequate level of detail because the purpose of the study has changed. All modelers need good channel cross-section data to conduct a satisfactory study.

Not having detailed enough terrain data to undertake the modeling effort is a major concern for 2-D models. If the data do not exist, it is not recommended to use a 2-D model because the model effort will not provide improved results over a 1-D representation. The 1-D model is a reasonable choice if the data do not support the 2-D model.
Decision between unsteady-state vs. steady-state model solutions is more of an issue when it comes to smaller streams/rivers with presence of hydraulic structures. The issue would occur, for example, at a culvert or bridge opening where the culvert could not handle the 100-year flood. The water would flow over the embankment and onto the road. One would have to use an unsteady-state model in this case, as the steady-state model will obviously produce erroneous descriptions of the flow. The modeler should have a clear physical insight on the way the river channels, culverts, floodplain areas, levees, and roads respond during floods. Also, the modeler needs to know the size, length, and complexity of the system that will be modeled. Lastly, the modeler needs to determine the purpose of the study and the expected level of accuracy before selecting any model (Brunner, 2010).

When no constrictions are present in the flow, 1-D steady-state modeling is by nature more conservative because the peak of the flood remains constant throughout the simulation; 1-D steady-state modeling is simpler to conduct, and requires lower level hydrology data, although it is often less accurate when applied to complex flooding areas (Hosseingipour et al., 2012). In addition, most communities have engineers who likely know the use of 1-D models such as HEC-RAS from the U.S. Army Corps of Engineers. (We stress herein that knowledge of a particular model is not enough justification for its use when clearly it should not be applied). In 1-D models, floodplains are described by series of cross sections perpendicular to the flow direction and water levels in the main channel and floodplain are the same in steady state (Frank et al., 2001) (This is clearly not true when the flood is rising or receding). Steady state models should not be used when flood events are very dynamic and levees overtop or breach.

In some cases 1-D models can produce the equivalent results to 2-D models when the dominant flow directions of a river or floodplain follow the general river flow path. It is also feasible to use a 1-D model when a levee is overtopped or breached and the protected area is small enough that it is known to fill to a leveled surface elevation (Brunner, 2010). On the other hand, 1-D models should not be used when a levee will be overtopped or breached and the water can go in many directions (larger protected areas). In addition, 1-D models are not good for bays and estuaries where the flood flow will go in multiple directions due to tidal fluctuations. 1-D models are also not appropriate for use when significant lateral elevation differences occur due to the flow around sudden bends (Brunner, 2010).

One important aspect that also differentiates the 1-D and 2-D approaches is the use of resistance coefficients. While in 1-D models the roughness coefficient is attributed to cross sections (typically Manning’s n), the resistance coefficient in 2-D needs to be assigned to each volume/element of the mesh. Resistance coefficients have been shown to be problematic parameters in unsteady modeling of bays during hurricanes, because model results obtained with customary values of Manning’s n, taken from steady-state conditions, do not match observations (A. Kennedy, Civil and Environmental Engineering and Earth Sciences Department, University of Notre Dame, personal communication, 2010). (This is a subject for which there is scarce information.)
To take advantage of the advantages of both 1-D and 2-D models, it is common to simulate the stream channel, where flow is typically longitudinal, using a 1-D model, and model floodplain flow in 2-D (Moore, 2011). In this way, the best features of both approaches are combined and used where they are mostly efficient and accurate.

The Department of Water Resources Central Valley Floodplain Evaluation and Delineation Program (CVFED) is a program designed to protect against loss of life, flood damages, and emergency costs caused by future floods. It is intended to protect California’s people and the infrastructure. The main task is floodplain studies for the Central Valley river systems which include the Upper and Lower Sacramento River System and the Upper and Lower San Joaquin River System (see http://bondaccountability.resources.ca.gov/plevel1.aspx?id=7&pid=4). CVFED has used unsteady models for the floodplain studies because they were deemed necessary due to the complex hydrology; also, it was argued that steady-state analysis would miss interactions between the mainstream and the tributaries.

In a case where a hydrologic model is used to perform both the rainfall-runoff over the watershed and all the routing within the system, then the flow rates in the steady-state model are only as accurate as the hydrologic model. This is why the question that needs to be asked of the hydrologic model is: Is it possible for the hydrologic flow routing to produce accurate flow rates that can be used in the steady state hydraulics model (Brunner, 2010)?

What are the financial considerations in this decision?

The total cost of the doing the flood study is composed of the following sub-costs:

- the cost of purchasing the software;
- the cost associated with training staff being able to learn and use the model;
- and the cost of the time used to create, calibrate and validate, and implement the model, to translate the available data into a usable form, perform the required simulations, conduct mesh convergence test, conduct a sensitivity analysis, and to interpret the model results.

The latter two are the most important factors in determining the cost. Higher-level products will require higher-level data, which will also require more time for someone to process the higher-level data, which will lead to an increase in costs. In general, 2-D models are more costly than 1-D models. Below are quotes of HEC-RAS and FLO-2D regarding costs:

1-D = HEC-RAS, free distribution
2-D = FLO-2D Pro #1 dynamic flood routing model-$ 995 (http://www.flo-2d.com/flo-2d-pro/).

Most communities will use the cheapest available model to meet FEMA’s standards even if that means being more conservative because of budgetary issues. However, caution
should be applied in general because, as stated above, in some cases a 1-D flood simulation will lead to an erroneous flow description.

- What are the issues associated with performing Floodway Analysis in Unsteady State?

A “Regulatory Floodway” is defined as the channel of a river or other watercourse and the adjacent land area that must be reserved in order to discharge the base flood without cumulatively increasing the water-surface elevation by more than a designated height. Communities must regulate development in these floodways to ensure that there are no increases in upstream flood elevations. For streams and other watercourses where FEMA has provided Base Flood Elevations (BFEs), but no floodway has been designated, the community must review floodplain development on a case-by-case basis to ensure that increases in water surface elevations do not occur, or identify the need to adopt a floodway if adequate information is available (FEMA).

In unsteady-state simulations, time discretization is needed and, thus, all potential problems associated with such discretization may occur if not all “rules” for an adequate numerical simulation are followed. (These rules pertain to any modeling effort in any branch of science.) For example,

a) issues of numerical instability can appear if inadequate discretization is used in models with explicit numerical schemes;
b) some aspects of the numerical result in terms of the flooding wave may become smoothed if the numerical scheme possesses excessive numerical diffusion, providing an inaccurate description of the flooding area;
c) the solution may show instabilities in some cases of presence of culverts, lateral structures if the model does not handle this appropriately.

2. How does one decide which model software to choose?

First we would like to differentiate between model software and model applications. Model software, in general, refers to the commercially or freely available software package that contains a graphical interface and the code that solves the equations; the model application refers to the implementation of the model software (1-D or 2-D; steady or unsteady; etc.) to solve the problem of interest.

The decision on particular software needs to be based on the modeling requirements (for the practical case to solve) versus the software capabilities. Software packages provide solutions which differ in many respects: equations and the method of solving those equations as well as boundary conditions. Some of the differences are discussed below. The selected model software will depend on the information available. Also, the model software selection depends on the level of data available, as discussed in Issue 1.
• **1-D: Prismatic Section Evaluation versus Variable Cross Section Geometry?**

When a great level of detail regarding topographic information is not available, “prismatic cross section models” can be useful, but the modeler needs to interpret the model results based on those topographic levels coming from “prismatic cross sections.” Variable cross section models will likely provide more accurate results, because they better represent the topography. An important aspect to consider is how to determine the cross section when there are abundant data. Some modelers make the mistake of generating contours with relatively large height differences between contours, and then they obtain the values of the cross section from the contours. This procedure is likely to produce large errors by linear interpolating between the contours, and does not take into consideration the abundance of data. If data are available, a detailed description of the cross section should be produced.

• **2-D: Grid or Mesh based Terrain?**

The distinction between grid and mesh was raised multiple times and the meaning can vary from education and experience. In the Computational Fluid Dynamics (CFD) texts and journal articles, there is no difference between grid and mesh. This seems to refer to specific terminology of a particular software package or modeling line of thought. We do not particularly share this description because we believe it is confusing. We prefer to talk about “structured” or “unstructured” grids (or meshes) and “rectangular” and “boundary-fitted” coordinates.

• **2-D: If Grid, 4 directional Flow or 8 directional flow, and what size grid?**

This is another terminology associated with a particular software package. The flow direction is usually specified by the commercially available FLO-2D code as shown below:

![Diagram showing 8 directional flow](image)

Initially, FLO-2D was a four-directional flow model (Demonstration video from FLO-2D). Now, it uses eight directions as shown above.
Below is a summary of the solution algorithm incorporated in FLO-2D, adapted from the FLO-2D Reference Manual from 2009 (we have followed the description in the manual as quotations, and tried to provide our understanding of the methodologies):

a) The average flow geometry, roughness and slope at the face between two grid “elements” are incorporated to the model and computed.
b) The flow depth $dx$ for calculating the velocity across a grid element boundary for the next time-step $(i+1)$ is estimated from the previous time-step $i$ by using a linear approach (the average depth between two elements):

$$d_{x}^{i+1} = \frac{(d_{x}^{i} + d_{x+1}^{i})}{2} \quad (9)$$
c) The first estimate of the velocity is computed using the “diffusive wave equation.”
d) The above predicted “diffusive wave velocity” for the current time-step is used as a seed in the Newton-Raphson solution of the “full dynamic wave equation,” to evaluate a corrected flow velocity.
e) The discharge $Q$ across the boundary is computed by multiplying the velocity by the cross sectional flow area.
f) The “incremental discharge” for the time-step, using discharges across the eight boundaries (or upstream and downstream channel elements) is computed as follows:

$$\Delta Q_{x}^{i+1} = Q_n + Q_e + +Q_w + Q_s + Q_{nw} + Q_{sw} + Q_{se} + Q_{ne} \quad (10)$$

and the change in volume (net discharge times time-step) is distributed over the available storage area within the grid or channel element to determine an incremental change in the flow depth, where $\Delta Q_x$ is the net change in discharge in the following manner:

$$d_{x}^{i+1} = \frac{\Delta Q_{x}^{i+1} \Delta t}{A_{surf}} \quad (11)$$

where the time-step $\Delta t$ goes from time $i$ to $i + 1$.
g) Numerical stability criteria are then checked for the new “element” flow depth. If any of the stability criteria are exceeded, the simulation time is reset to the previous simulation time, the time-step is reduced, and the velocity computations are re-started for that simulation time.
h) The simulation progresses in the same manner as above.

We would like to manifest at this point that the explanation provided in the FLO-2D Manual is unclear to the authors of this report as is the terminology employed.
2-D: Is it necessary to embed 1-D elements and what are the software capabilities to do so?

We are not clear about the real meaning of the word “embed” in this context. We interpret it refers to the combination of 1-D and 2-D approaches.

It is sometimes recommended to combine 1-D within 2-D approaches to save computational time in those streams where a 2-D description is not necessary. The capabilities for doing so are obviously software dependent.

Coupling 1- and 2-D models takes advantage of both methods. Modeling some stream channels in 1-D can allow for reducing grid cell resolution in the 2-D model when the stream channel is very narrow (Dhondia and Stelling, 2002).

Another option for coupling 1- and 2-D models is to use a general 1-D model for the main channels and to model the floodplains in 2-D. MIKE FLOOD couples 1- and 2-D approaches using MIKE11 and MIKE21, respectively (all commercially available from DHI Water & Environment). A number of options are available for linking the two models. Lateral linkage explicitly couples MIKE11 to MIKE21 by modeling water entering the floodplain from the stream channel. The flow from the stream channel to the floodplain can be modeled using a simple weir equation. Momentum is not conserved using lateral links, due to the inability of 1-D models to simulate cross-channel flow (DHI, 2009).

Transfer of flow between the channel and main bank is an important factor when dealing with 1-D/2-D coupled models. There should be validation tests for 1-D/2-D-coupled models. One way to test this is to have a test scenario where there is overbank flow and the distributions of momentum and mass are known. The results for the 1-D/2-D-coupled should be compared to the 2-D model to see if 1-D/2-D model produces valid solutions in conserving momentum/mass going from the channel to the floodplain.

2-D: Is cost a factor? For current model needs and future derivative works?

Cost is an important factor for communities. As discussed earlier, the total cost of performing the flood study is the cost of the software, the cost associated with training staff to learn and using the model, and the cost of the time required to model the project (including calibration and validation). The latter two are the most important factors in determining the cost. As models become more advanced, they will require more time for someone to process the higher-level data (or more time for the software developer to include automated processes), which will lead to an increase in costs.

What Equations are Solved?

One-Dimensional Flow Models

1-D flood models typically use finite-difference, finite-volume, or finite-element
solutions of the full Saint-Venant Equations (Bates and De Roo, 2000). The Saint-Venant Equations express both continuity and momentum which are obtained by cross sectionally-averaging the Navier-Stokes Equations after turbulence averaging (Wu, 2007):

\[
\frac{\partial A}{\partial t} + \frac{\partial q}{\partial x} = 0 \tag{12}
\]

\[
\frac{\partial q}{\partial t} + \frac{\partial}{\partial x}(uQ) + gA \left( \frac{\partial h}{\partial x} - S_o \right) + gAS_f = 0 \tag{13}
\]

where \( Q \) is the discharge, \( A \) is the cross-sectional area, \( S_o \) is the bed slope, \( h \) is flow depth, \( x \) is the longitudinal distance, \( t \) is time, and \( S_f \) is the friction slope. The terms included in the momentum equation (Equation 13) above are: unsteadiness or local time variance of flow; spatial flow variation; pressure water level gradient; component of weight and friction forces, respectively. The first two terms are often called inertia terms. Assumptions embedded in the 1-D Saint-Venant Equations are that the pressure distribution is hydrostatic, the resistance relationship for unsteady flow is the same for steady flow, and the bed slope is sufficiently mild such that the cosine of the slope can be replaced by unity (Stelling and Verwey, 2005). Some models use a reduced form of the momentum equation shown above. One of the reduced forms is the Kinematic Wave equation, which can be used when the energy grade line is quasi parallel to the channel bottom as shown below:

\[
\frac{\partial A}{\partial t} + \frac{\partial q}{\partial x} = 0 \tag{14}
\]

\[
S_o = S_f \tag{15}
\]

Another reduced form of the momentum equation is the non-inertia wave equation, in which the local and convective inertia terms are ignored and is shown below:

\[
gA \left( \frac{\partial h}{\partial x} - S_o \right) + gAS_f = 0 \tag{16}
\]

MIKE11 uses a more general form of the cross sectionally-averaged Navier-Stokes Equations, and uses either the \( C \), the Chezy coefficient, or Manning’s \( n \) to parameterize the bottom resistance (DHI, 2009).

Two-Dimensional Flow Models

The same principles used to derive the 1-D Saint-Venant Equations by integration over the whole the cross section are used to derive the 2-D equations by integration only over depth\(^1\), whose final forms are expressed below:

\(^1\) For purposes where vertical resolution is needed and not transverse resolution, the equations can also be integrated over the transverse direction instead.
where \( h \) is again the water depth; \( U \) and \( V \) are the depth-averaged velocity components in the \( x \) and \( y \) directions; respectively; \( T_{xx}, T_{xy}, \) and \( T_{yy} \), are depth-averaged turbulent stresses; \( z \) is the water surface elevation; and \( \tau_{hx}, \tau_{hy} \) are the bed shear stresses due to friction (Moore, 2011). The terms above are: unsteadiness or local time variation of flow; spatial flow variation; shear stresses; pressure gradients and component of weight and friction. Shear stresses have an important role in the numerical solution of the equations.

2-D approaches assume that the lateral boundary layers are negligible. Whereas this is a good approximation in most studies, the modeler needs to check that this is the case for the application under analysis.

3. **How does one evaluate the qualifications of the consultant preparing the model?**

Qualifications that we believe should be required for the consultant preparing the model are: 1) a M.S. degree in modeling from a Department of Civil and Environmental Engineering (or equivalent experience); 2) work experience of no less than 2 years on modeling floods and developing flooding maps; and 3) experience with the selected software.

Also, the consultant needs to have deep knowledge of hydraulics and hydrology, as well as an in-depth knowledge of the local system. Engineers with greater knowledge, more experience and better training would likely be more successful in producing better results, regardless of the software that is being used.

We have learned that in recent years some consultants have incorporated graduates from the Computer Science field to perform flood studies. Whereas this is something nice and encouraging to observe (because it means a multi-disciplinary approach to the problem), we would like to stress that a basic knowledge of hydraulics, hydrology and modeling is necessary to undertake these studies in first place. Therefore, professionals from the Computer Science field are most welcome in hydraulic modeling studies, but we stress that they should be trained in hydraulics and hydrology to adequately perform their duty in floodplain modeling.

4. **How do local agencies ensure they are getting a good model for their money?**

Currently, there is a general submittal package for FEMA maps. The digital Technical Support Data Notebook (TSDN) contains all the support data to be requested of a given
Community. The requirements for the TSDN are documented in Appendix M of FEMA Guidelines and Specifications for Flood Hazard Mapping Partners.

FEMA can ask for the draft TSDN up front and ask for decision-making processes before the study starts. During the BRP meetings, it was determined that documents can include the discovery process, a detailed work plan (more detailed than the scope of work), and have a component in the report for the modeler experience and decision for choosing the selected software.

We believe that the consultant/community must provide FEMA with enough quality assurance documentation to check that both the model has been adequately implemented and validated, and that results can be trusted. Consultant must provide FEMA with detailed information based on GIS layers, written reports, and other clear pieces of documentation, as detailed below.

FEMA needs to ensure that the modeler(s) developing the model properly document/s: a) the data used in the implementation of the model, b) the decision-making process in the selection of modeling approach and parameters, c) the boundary conditions used, and d) the considerations during the calibration process. The uncertainty in the hydrology data needs to be reported (if applicable) and, within the parameters, the uncertainty in the determination of Manning’s \( n \) (for example) should be properly assessed and documented.

We believe that by requesting these pieces of information, FEMA is justifying the investment done, because it will be evident to the reviewer if the model has flaws.

The United States Army Corps of Engineers (USACE) process for new engineering studies is based on obtaining input from a variety of engineers and on discussing the problem before even starting the study. The group includes people from all sections of engineering such as geotechnical, hydraulic, hydrology, flood damage, etc. This allows for the problem to be examined from all aspects and for the determination of any potential problems that can occur during the study. This process is time consuming but the end result is beneficial; USACE has more confidence in the results.

5. **How does FEMA or local agencies cost effectively review the model?**

FEMA and local agencies can cost effectively review a developed model by checking that the flood study follows FEMA guidelines and has been adequately undertaken. Proper and concise documentation should be provided so the study can be easily understood and duplicated by the local agency or FEMA. The more detailed the documentation, the easier a model can be reviewed and verified. Also, FEMA can require the agency to submit their QA/QC review document as part of the standard submittal package.

In addition, FEMA could also reach out to active members of FMA and create a Reviewer Working Group. This group would help define what it takes to be a good
reviewer and this group would be responsible for peer reviewing expedited flood studies and Letter of Map Revisions (LOMRs). A concern that many agencies have is that it takes a lot of time to approve a flood study or LOMR. If the flood study is approved by the Reviewer Workgroup then all FEMA would have to do is validate the documentation process since it has the seal of approval from the Reviewer Working Group. This could help expedite the process to approve an agency’s flood study. Agencies can pay a fee to fund the efforts of the Reviewer Working Group since the review of their study will be expedited.

- **What are the technical elements that go into the review of the model?**

In our opinion, this issue involves two aspects: First, to check that the software used is adequate and, second, that the flooding analysis has been properly developed.

According to FEMA Policy for Accepting Numerical Models for Use in the NFIP and the Clarification of National Flood Insurance Program Criteria for Certification of Coastal, Hydrologic, and Hydraulic Models, the requirements for flood codes are:

- The model software (code) must have been reviewed and accepted by a governmental agency responsible for implementation of programs for flood control and/or regulation of floodplain lands.
- The model software (code) must be well documented, including source codes and user’s manuals.
- The model software (code) must be available to FEMA and all present and future parties affected by the flood insurance mapping.

If the certifying agency, an agency certified by a governmental agency responsible for implementation of programs for flood control and/or regulation of floodplain lands, is not a federal agency they must review, test, and accept the model. They also need to follow the following requirements for certification of the model:

- The certifying agency must review the model in sufficient detail to conclude the model is scientifically correct and technically sound. The model must be based on hydrodynamic, and hydrologic principles. The certifying agency must rely on published technical papers by authors other than the model developers. FEMA may request a list of reviewed technical references.
- The certifying agency must test the model with detailed data or compare the model to similar models on the FEMA list of “Numerical Models Accepted for Use in NFIP,” so the model can produce results similar to a model on the FEMA list.
- FEMA may request certifying agency to provide answers to technical questions.

Technical elements to consider when reviewing the model results (once the study has been developed) are:
• The approach to solve the flow problem has been adequately selected (i.e., it has been correctly determined whether the flow is 1-D or 2-D).
• The domain is properly discretized, which considers both size and location of cells and include mesh convergence testing.
• Boundary conditions are well selected and implemented.
• Solution does not present numerical instabilities.
• Calibration (including adequate validation and verification) of the implemented model has been correctly done and shown.
• A sensitivity analysis has been made of the results to demonstrate the range and uncertainty of the results.

• **Are there standard submittal templates that can be developed to standardize the model submissions?**

The general submittal package for FEMA maps is the digital Technical Support Data Notebook (TSDN), as noted before. TSDN contains all of the support data for a community. The requirements for the TSDN are documented in Appendix M of FEMA Guidelines and Specifications for Flood Hazard Mapping Partners (http://www.fema.gov/media-library/assets/documents/6998?id=2206).

FEMA can ask for the draft TSDN up front and ask for the decision-making process before the study starts. Also, documents can include the discovery process, a detailed work plan, more detailed than the scope of work, and have a component in the report for the modeler experience and decision for choosing the selected software.

Standard submittal templates should include the Digital Elevation Model (DEM) of the terrain, GIS layers including the pre-processing of the available data and of the numerical results, etc., as detailed above. These data should be provided instead of just model outputs. All files should be provided in flexible formats to be read easily by FEMA and formats that do not require specific proprietary software.

• **How does FEMA ensure they have trained reviewers for the different models?**

In the current way FEMA handles the review process the private company, Baker Corp., is in charge of making the decision on who reviews new studies, and if a given reviewer has the qualifications to do a review (von James, personal communication).

Reviewers can be selected from modelers who have already conducted studies and done reviews. FEMA needs to be aware of modelers who conducted studies whose results were not accurate/satisfactory. Reviewers should also have the knowledge of the local features of the area of the study. It would be best to have at least 1-2 members with local knowledge on the review team. Ideally, the review team needs to have knowledge on the software, hydrology, hydraulics, and the local system.

USACE review process for engineering studies first consists of a local expert looking at any given study; next a broader team looks at the study; finally, a third party does the
final review if the study is large. The team of reviewers should possess a mixture of knowledge on local aspects, hydraulics, hydrology, environmental issues, etc.

USACE process for obtaining qualified individuals for each study is obtaining a list of experts in each field. USACE Headquarters (HQ) sends out a letter requesting a list of experienced individuals for different field areas. USACE HQ sends documents to each listed engineer asking the individual’s expertise and confirming with their manager that they are experts in the field they listed. FEMA could obtain a list of names and resumes from agencies and similarly determine which individual is an expert in each field.

FEMA can ensure they have trained reviewers by checking that reviewers have the proper education and attend online programs, FEMA workshops, and ASCE webinars. It is desirable to transfer knowledge and exercise mentorship from experienced modelers to young new modelers. FEMA should work with UC Davis, or other independent organizations, to set up classes and webinars. Recognizing this need, UC Davis Civil & Environmental Engineering Department has recently developed a graduate course to go beyond theory and teach students in the practice of applying models.

FEMA could also reach out to active members of FMA, and create a small group of current reviewers that can mentor new modelers. FEMA could pay for time of mentors and the activities they do as mentors. Mentors would need to conduct webinars and seminars and train new modelers. University extension and online, remote teaching programs could collaborate with FEMA and FMA to set up some of these webinars and seminars.

- **How does FEMA ensure it is reviewing the elements that matter?**

It seems that FEMA already ensures it is reviewing the elements that matter by having a Technical Study Data Notebook (TSDN). The TSDN contains all of the support data for a community for which FEMA published/revised a flood map. TSDN information can be found in “Guidelines and Specifications for Flood Hazard Mapping Partners, Appendix M: Guidance for Preparing and Maintaining Technical and Administrative Support Data.”

We would like to manifest that we strongly believe that the materials mentioned above are crucial in checking that a satisfactory study has been accomplished.

6. **Many times the models have large data sets. What are the data storage and handling considerations? What needs to be submitted with a model for review?**

For models with large data sets, one solution might be that all information should be stored in a CLOUD. Items that need to be submitted include DEMs, calibration details in GIS format, results in terms of velocity vectors and contours of water heights, and comparisons with data in several cross sections; etc. A report containing all considerations developed to decide for one particular approach and values of roughness need to be provided as well.
There should be a minimum requirement to convert the LiDAR data to some resolution so anyone could take the data and reproduce the results. For CVFED, the LiDAR data were converted to a 1-foot grid, where raw data was 4.5 TB and in Global Mapper format was 250 GB. In the Global Mapper format, one can open the entire Central Valley and zoom into any area. Global Mapper is a GIS Data processing application that offers access to an unparalled variety of spatial datasets and “provides just the right level of GIS functionality to satisfy both experienced GIS professionals and mapping novices.” It also has capabilities of image rectification and contour generation from surface data (Global Mapper).

FEMA does have an ongoing collaborative effort with USGS to establish the topography standard for models. Also, FEMA is going toward having a portal for data storage in collaboration with DWR.

7. **FEMA has the requirement to make the data freely available to review. How does FEMA handle the issue of proprietary software?**

All data should be submitted in an established universal format; not in a format which belongs to unique proprietary software. The issue is what format to select. Within the discussions of the BRP, some members advocated the use of hdf5 format, a format developed by NASA some years ago for time series data. Also, FEMA can require Vendors to provide 30 day trial versions of their software.

8. **Software vendors work to improve their modeling software with new versions. What are the issues that arise from version changes?**

Issues with version changes are a common problem with all software packages. Sometimes new versions of software are released to correct errors produced in previous versions. Sometimes new versions are released to provide new code features. FEMA should consider software that will remain backwardly compatible or where utilities to convert data are provided.

- **How does FEMA track software version changes?**

FEMA should track software version changes of the authorized software, by requiring the model developer to give information on software changes from version to version. Model developers should indicate clearly the nature of the change (improvement of any given capacity or addition of new feature), and should show that the change does not affect previous capabilities of the code. This is where model validation is absolutely necessary. If a model is well calibrated and has produce good results compared to historical data, it is assumed to be a good model. When a new version of the software comes out, the new results also need to be compared to the historical data. If the results compared to the historical data are better, then there is no problem. If the results are worse, then the model must be further calibrated with the new version to reproduce acceptable results (or establish that an error was made in the earlier work), as long as the parameters of the new calibration are within physical margins. If no calibration data
were available, and the new version produces different results, then a detailed analysis of the results, by an experienced engineer/modeler must be performed to decide which results he/she thinks are better. If it is deemed that the older versions results are better, the model may have to be modified in order to use the new version, such that it produces the same results as the previous software version.

FEMA should consider a requirement that all vendors produce backwardly compatible products or utilities that would translate older boundary conditions into the new format. FEMA could request the consultants/communities to turn in the inputs and outputs of the model for any given study, developed with the last version of the software and in a previously approved universal format. At least, they should show that the solution is compatible with the last version of the code.

- **How does FEMA ensure that the developed models remain usable through software version changes?**

FEMA could require that the latest 2-D model results be “close” in some mathematical sense to those coming from previous 2-D modeling approaches. If the original results are not available, the flooded areas of the new model implementations need to be close to available flood maps. Issues will arise if the original flood map was done in 1-D (addressed later). FEMA could require model developers to provide standard data sets with the models that encompass a wide range of modeling situations and complexities to check software changes. These data sets and their results should be documented and provided with the software. New versions can be run on these data sets to see if they give the same or better results. Also, FEMA could request a set of benchmark tests to be passed by the new versions of software. In this regard, this research effort intends to recommend specific benchmark tests.

- **How does FEMA ensure that modeling programs continue to produce valid results?**

FEMA could ensure that software programs continue to produce valid results by requiring developers to provide results of benchmark tests to FEMA, as discussed in the previous issue.

- **Should FEMA continue with software Accreditation?**

FEMA has eliminated software accreditation and has new Operational Standards that new flood studies must follow, however we strongly believe that there should be a framework established to ensure accreditation of models.
9. What are the issues associated with social justice and model complexity? What happens when communities do not have the technical resources to review, use, or modify the 2-D model?

Communities have, in general terms, few resources to address flood issues. In some cases, the staff has limited knowledge and experience with flood models. The cost associated with conducting a flood study is the cost of software, the cost to train the modelers, and the cost to develop and use the model. The latter two costs are what primarily make it difficult for some communities to conduct a satisfactory flood study.

During the discussions at the BRP, it was mentioned that there are communities with budget constraints for which there would not be enough resources to undertake a complicated 2-D model, for instance, and a 1-D model could be in principle accepted as a solution on grounds of “social justice.” Whereas this is easy to understand, caution should be exercised, because in some cases a 1-D approximation is utterly inadequate, as stated in the response to issue 1. Therefore, budgetary limitations should NOT be an excuse for poor judgment in the decision of the modeling approach.

10. How are floodways addressed in 1-D unsteady or 2-D model? What happens when numerical instabilities prevent 1-D unsteady model from being valid?

The first portion of this issue has been addressed in the responses provided to above issues.

Unstable model solutions are not acceptable; the solutions need to be free from numerical instabilities to be considered valid. This concept is well grounded in the field of numerical simulation of problems in many branches of science. If the solution has numerical instabilities, it is because there is an intrinsic problem with the solution, and it is equivalent to not having a solution in first place.

The stability of a model is best described by examining the Courant number \((Cr)\), through the so-called Courant Criteria. The Courant number is defined as follows (Abbott, 1979; Engineers Australia, 2012):

\[
Cr = \frac{v + \sqrt{gd}\times dt}{dx}
\]

where \(v\) is velocity (m/s), \(g\) is gravity, \(d\) is depth, \(dt\) is the time step and \(dx\) is the grid/mesh dimension. Theoretically, it is only explicit models that must satisfy the criteria that \(Cr\) shall be no greater than one (1) in order to avoid “unstable” or inaccurate results. A fully implicit scheme should be able to utilize Courant numbers that are infinitely high provided that the time step is sufficiently small to depict the shape of the boundary time series. Semi-implicit schemes, such as those used for fixed grid finite difference solutions are able to run at \(Cr\) conditions greater than one, but are unlikely to produce stable and reliable results for \(Cr\) values greater than 10. Note that \(Cr\) varies throughout the model domain, so it is the areas that experience the highest \(Cr\) value that limit the time step. These areas are usually the deepest flow areas where the hydraulic...
friction from the bed of the model is at a minimum within the model domain. However, for implicit solutions, the actual Courant number can be much smaller in practice and may approach one (1), depending on the application. The introduction of 1-D elements can introduce into an implicit model a $Cr$ limitation much smaller than for the rest of the domain. In general, model stability and accuracy is compromised where the $Cr$ value is too high for the numerical scheme being used. On investigation of the Courant equation, this typically occurs when the velocity, depth or time step is too large in combination with an element size, which is relatively small. A key issue in the model development process is to gain an understanding of the time step that is likely to be required (and hence overall run time) for a specific mesh/grid resolution. This analysis will require an estimate of the likely modeled flow depth and velocities” (Engineers Australia, 2012). Although we do not completely share all ideas of stability in the lines quoted above (basically because the notion of accuracy should not be discussed for unstable solutions), they summarize some of the issues of stability in flood models.

Instabilities can occur at or near structures, especially if the discretization is poor. The process for resolving the instability will vary between software and techniques. For example, a finer grid/mesh size might be needed in some cases (Engineers Australia, 2012).

At the same time, consultants should do mesh convergence tests so the model is proven to be accurate.

11. How are model parameters established? How does a modeler/reviewer know if the correct parameters have been used?

- In 1-D models n-values are a lumped parameter for uncertainty. How is the n-value different for 2-D models and what are the appropriate n-values?

Manning's $n$ in Manning’s equation is based on empirical data of rivers around the world, but Manning’s formula has been obtained through theoretical means based on the phenomenological theory of turbulence (see Appendix 4: Gioia and Bombardelli, 2002). Manning's $n$ is less sensitive to changes in water levels than other roughness coefficients (Yen, 2002), and it does not vary too dramatically as the hydraulic radius changes, in comparison with other roughness coefficients. Manning's $n$ also varies in time because vegetation grows or is modified by the flow, in addition to the variability in space. Manning's $n$ values range from 0.02-0.15 for channel (Chow, 1959).

We do not fully agree with Manning’s $n$ being a “lumped parameter for uncertainty.” We believe that the coefficient should be kept within plausible values as much as possible. If that is not the case, it is because the coefficient is being charged with other flow scales which are not being modeled correctly. In fact, many modelers unfortunately use Manning's $n$ as a “garbage dump” to include all the scales a given model is not capturing. This excessive “tweaking” of the roughness coefficients should not be accepted by FEMA.
In 1-D, $n$ is used for cross sections (the original assumption), whereas $n$ is used in 2-D for each mesh/grid element (cell). In most cases, Manning’s $n$ values are relatively close for 1-D and 2-D approaches. A few exceptions are where there are rapid changes in flow direction and magnitude (e.g., at a structure, sharp bend or embankment opening). Software that solves the full 2-D equations will model the loss of energy due to the water changing direction and magnitude through a change in roughness. Therefore, the $n$ value in a 1-D model at, for example, a sharp river bend may be higher than for the 2-D model to represent the losses (Engineers Australia, 2012). Values of Manning’s $n$ (roughness) can be determined with the help of aerial or satellite imagery, textbooks, guidelines found in manuals, and site inspections (surveying).

In what follows, we comment on the main ideas contained in the paper, Knight et al. (2010), which received an award by the International Association for Hydro-Environment Engineering and Research (IAHR).

Roughness appears in simplified or discrete numerical models to account for momentum and energy dissipation at wall boundaries, and it is a measure of flow resistance. Thus, it is needed for models of all dimensions. The representation of the flow resistance categories is, however, different for 1-D, 2-D, and 3-D approaches. In 1-D and 2-D models, roughness is more uncertain and less accurate in terms of the definition and sizing than in 3-D models (Knight et al., 2010). Flow resistance can be classified as skin drag, form drag, or shape drag. Skin drag is produced by “roughness” due to surface texture; form drag is “roughness” due to surface geometry, and shape drag is “roughness” due to overall channel shape (Knight et al., 2010).

In order to approximate the notion of roughness, simple experiments such as flow of liquids in circular pipes were used to gain knowledge on roughness in early times of fluid mechanics. The 1-D Darcy-Weisbach equation for flow in circular pipes relates channel geometry, flow, and pipe characteristics (relative roughness, $k_s/d$, where $k_s$ is the Nikuradse equivalent sand roughness size). The extension of this simple concept to compound channels is not easy. For instance, in open channels, the surface can be mobile, composed of loose boundary material such as sand or gravel, and/or flexible, composed of deformable material such as vegetation or instream weeds (Knight et al., 2010). In addition, channels can be compound. The Moody chart should not be used when specifying head losses in compound channels or complex shaped ducts because the hydraulic radius, $R$, is not a suitable characteristic geometric parameter, as is in the case of simple 1-D pipe flow. The wetted parameter, $P$, will have an unexpected change at the bank-full stage and the cross-sectional area, $A$, will not, which gives rise to discontinuous relationships in $f$ versus the $Re$ domain (Knight et al., 2010).

Unsteadiness in the flow affects the turbulence structure and as a result alters the bed shear and the resistance. Vegetation alongside the river channel also affects the resistance to flow. This is why seasonal growth/decay or managed weed cutting mean that vegetation will vary spatially. In alluvial fans, sediment has an effect on flow resistance. Alluvial resistance relationships are much more complex because geometry bed forms change with velocity and depth of flow. Some models include turbulence but
other models do not, so it is not correct to use the same friction factor for both 1-D and 2-D models (Knight et al., 2010).

As said the 2-D shallow water equations are derived from vertical integration of the Navier-Stokes Equations, whereas 1-D models are obtained by integrating equations over the cross-section. Friction is characterized by using Manning’s $n$ or Chezy’s $C$. 1-D Manning’s $n$ and Chezy’s $C$ friction coefficients represent the shear stress exerted by the entire bed and banks bounding the flow while 2-D models use the friction factor to represent shear stress exerted at the base of a vertical column of water. In 3-D, a no-slip condition applies at the walls meaning the tangential and normal velocities to the wall are zero, whereas turbulence is modeled explicitly by the code. For 3-D models, the roughness appears in the boundary condition rather than as a term in the equations. Therefore, the impact on the solution is not as significant as in 1-D or 2-D models (Knight et al., 2010).

It is difficult to understand roughness without considering the numerical discretization of the model, the theoretical model, and the flow characteristics. The main difference for roughness is the amount of physics that each dimensional level of flow model includes (lateral/vertical velocity, density, and turbulence). Each approach for defining roughness coefficients has its own definition. Roughness implicitly means it is a function of the flow it represents and depends on the water depth as well. Modelers need to keep in mind that increasing the value of the roughness parameters has significant impacts on the algebraic equations (Knight et al., 2010).

- **How should infiltration be handled?**

Evaporation and infiltration should be considered in 2-D unsteady simulations for long lasting floods (Central Valley) if it has not been considered in the hydrology already. In the Sierras, floods that last a short time do not need to include this. Also, floods over alluvial fans, or other high permeability surfaces, need to consider infiltration. Guidelines should be based on duration of the flood and permeability of the bottom.

If hydrology and hydraulics are solved together, infiltration can be computed in the rainfall runoff model and the flow in the river system. Rainfall and infiltration should not be double counted by being included in both the hydraulics and hydrology.

- **How should the hydrology be handled?**

The question regarding hydrology during the 2012 2-D Symposium was the following: Should the model integrate direct rainfall-runoff conversion (i.e., hydrology) or should the modeler use an external program to bring in such processes?

Some 2-D models do include direct rainfall-runoff conversion in the model. The modeler needs to fully understand both the hydrology and the hydraulics in the model. He/she should calibrate and validate both levels of the model. Hydrology might be simpler to calibrate because in some cases there are rural and urban areas where a historic event has
taken place and the rainfall data, and water surface elevations are known for the event. The modeler can thus run the hydrological model and obtain calibrated hydrographs. The problem is that the overall accuracy will be governed by the lower accuracy of either model.

- **Should there be guidance regarding when to consider sheet flow and when to consider point discharge? In urban areas, sheet flow may be more prevalent.**

FEMA defines sheet flow as the flow where there are “inadequate” or not clearly defined channels; in those cases, floodwater spreads out over a large area at a somewhat uniform depth. Sheet flows occur after an intense or prolonged rainfall during which the rain cannot soak into the ground. During sheet flow, the floodwaters cover a wide area (FEMA). This should be explicitly handled in the software.

- **How is grid/mesh element size determined? What makes that size appropriate for certain applications?**

It is difficult to establish absolutely general guidelines. What is clear is that more “cells” (i.e., volumes or elements or points, according to the numerical method employed) are needed where the higher gradients in velocity are located. Velocity gradients are usually linked to gradients in topography.

We strongly believe that convergence tests should be performed to ensure that the numerical result is independent of the mesh, because this is a required step in the field of numerical methods. At least two mesh resolutions must be tested to provide any guidance on efficacy.

The theoretical approach for determining grid size is to try different grid resolutions until the results converge (i.e., no further change in results are found). New 2-D models are allowing the modeler to specify different grid sizes and run the model with each grid size specified which make comparing results with two different grid resolutions easier. In spite of the fact that comparing results coming from the use of different meshes may be more problematic for some practitioners, we believe it is absolutely needed.

- **How should time steps be established and reviewed?**

The model simulation time step is dependent on the model grid/mesh resolution. Poor model discretization can lead to inefficiently small time steps, which in turn produce high run times. In some explicit codes, time steps are automatically determined through stability criteria. There are generally 2 choices for selecting a model time step, a fixed regular time step or an adaptive time step.

The fixed regular time step allows the modeler to pre-determine the model run time and to set the output saving step (in which model results are saved) as a regular multiple of the simulation time step. However, the time step will need to be set at the shortest time interval necessary for stability of the model during the most energetic or deepest flows.
during the simulation. This typically occurs for only a very short period of time during the peak of the flood hydrograph.

The adaptive time step uses the Courant condition (in explicit models) to determine the appropriate time step. The modeler sets a maximum and minimum time step, which allows the model to use larger time steps where flow is shallow and smaller time steps during the flow peak. In theory, “this should allow the shortest run time for the simulation to be achieved whilst maintaining model stability.” In reality, this can cause really computationally long model runs. “This is due to the impact of a few minor locations in the model where short lived energetic fluctuations in the flow can lead to the minimum time step being selected for excessively long periods of time. Run times can also become excessive if the period that it takes for the flood wave to propagate through the model is very long. For example, simulations of large river systems or of flat terrain, where the critical rainfall duration is long, will have propagation times in the order of days, if not weeks” (Engineers Australia, 2012).

The advantages of using an adaptive time step is that a longer time step can be used when flow is uniform and a smaller time step can be used with rapid changes in flow (Engineers Australia, 2012).

Volume conservation error is a standard output in most programs (compares inflow and outflow) to show the accuracy of the model. Some software calculates the volume conservation errors and for the next time step adds the error in volume conservation back into the solution, which is a trick that (vendors argue) minimizes the error in volume conservation. Unfortunately, this process is creating/destroying mass. Software should not generate or destroy mass.

• What about the other parameters?

This question is not completely clear to us. We interpret that it refers to other parameters affecting the grid size and distribution. Other parameters include:

• Hydraulic roughness parameters.
• Energy losses at structures/bends.
• Inflow hydrographs.
• Downstream boundary location and assumptions, particularly stage-discharge boundaries.

• How are the grids established? Can/should mesh convergence be developed to ensure grid is appropriate for physics used? Can/should grid size be tied to the level of accuracy of the topography?

A modeler should always use multiple grid sizes to see if answers converge, as discussed earlier. Grid definition/distribution depends on features of each particular problem. When mapping urban areas, the modeler should determine the location of grid sizes as a function of the terrain. A modeler should use smaller grid sizes where greater detailed is needed and where sharper gradients are located. If the study is mapping property by
property, then grids require small discretization so it is possible to capture flood elevation for each property.

The topography needs to be known with a level of resolution greater than or equal to the grid size used in the hydraulic analysis. This should be always the case.

- **How should islands and ponds be handled?**

Stranded islands can be treated as solid boundaries. If an island or a pond is important to the solution of the floodwave movement, timing, and magnitude (water surface elevations, flows, velocities), then the model should be developed in a way to include it in a correct physical manner in the computations.

An important aspect in the treatment of these matters is given by the relative size of the mesh element (or volume) with respect of the size of the island or pond. If the latter are smaller than the mesh size, they should be included through an ad-hoc treatment of the bathymetry and/or roughness coefficient. Otherwise, they can be analyzed in an explicit manner.

12. **How are levee systems modeled?**

- **How are levees modeled?**

Levees can be treated, in 2-D, in different ways according to the code employed. For instance, they could be treated as internal conditions if the grid size is much larger than the width of the levee. 2-D models can thus use break-lines to define floodplain features such as levees, embankments and channels. Break-lines need to be specifically included in the Digital Terrain Model (DTM) and the hydraulic model due to their critical hydraulic importance (Engineers Australia, 2012). An alternative would be to actually reduce the grid size locally to represent the levee explicitly if adequate information in the DTM is available, and when submergence is important. However, this last methodology cannot be trusted when the DTM of the levee is off by 1 foot or more. In addition, the results will not be accurate in cases of strong slopes in the lee side of the levee, because the solution of shallow water equations down a steep slope is not accurate.

USACE manages a National Levee System database where it has surveyed levees over the last 5 years.

Many Central Valley levees fail due to under-seepage, rather than overtopping. All known models represent levee failure through overtopping but not through under seepage. This is something that should be included in all models if levee failure is of special concern for the study.
• **How are non-levee embankments and roads modeled?**

Non-levee embankments and roads are modeled differently by various models and often can be handled in multiple ways within a model. DWR guidelines require that any embankment that has an effect on the extent of the flooding should be included in the model. Any non-levee embankment also should be included if it affects the properties of the flood.

13. **How should local drainage and in-ground piping systems be modeled? When you put them in, when do you leave them out?**

The treatment of local drainage and in-ground piping depends on the specific problem at hand and of the code capabilities. If water needs to be removed then it needs to be considered carefully as mass needs to be conserved.

The first question should be how big the volume of the drainage system is compared to the volume of the flood. Typical storm systems handle 10-year storm events. This will definitely be a concern when highly urbanized areas are being modeled such as Los Angeles and Ventura County.

14. **How are the results mapped?**

Current FEMA maps results are presented through Hard Copy Flood Insurance Rate Maps (FIRM). Flood risk information presented on FIRM s is based on historical, meteorological, hydrologic, and hydraulic data, as well as open-space conditions, flood control works, and development. To prepare FIRM s that illustrate the extent of flood hazard in a flood prone community, FEMA conducts engineering studies referred to as Flood Insurance Studies (FISs). Using information gathered in these studies, FEMA engineers and cartographers delineate Special Flood Hazard Areas (SFHAs) on FIRM s. SFHAs are those areas subject to inundation by a flood that has a 1-percent or greater chance of being equaled or exceeded during any given year (100-year flood). This type of flood is referred to as a base flood. The base flood is a regulatory standard used by Federal agencies, and most states, to administer floodplain management programs, and is also used by the National Flood Insurance Program as the basis for insurance requirements nationwide.

• **What needs to be done to correctly convey the results of the analysis on a FEMA map?**

It is believed that FEMA maps should include both water depths as well as velocities. Additionally, some measure of risk could be shown. In this way, the main factors characterizing the purpose of the map will be given.
- Are the current AE/AH/AO zones appropriate? Should there be a special zone that includes the velocity?

According to the FEMA website, “A” zones are the second most volatile Special Flood Hazard Area (SFHA). These zones are usually located adjacent to a lake, river, stream or other water course and are subject to rising waters. Flood insurance is also mandatory in most all A zones. Also included are AE, AH, AO, AR, and A99 designations, named for the way in which they might be flooded. Zone A - subject to inundation by the 100-year flood, no base flood elevations shown, mandatory flood insurance applies. Zones AE and A1-30 - subject to 100-year flood, base flood elevations are shown, mandatory flood insurance applies. (On new maps, Zone AE is used in place of Zones A1-30.) Zone AO - subject to 100-year shallow flooding (usually sheet flow on sloping terrain, 1 to 3 foot average depth), mandatory insurance applies (FEMA website). Flood hazard zones are defined by NFIP and adding/modifying any special zones needs to be done by modifying NFIP regulations.

FEMA should consider incorporating high velocity hazard zones into the mapping and analysis of floodways to provide information on evacuation and personal safety issues. Another possibility is to use Depth times Velocity, which might be a better way to classify flow hazards.

15. If different models give different results how does FEMA handle “model shopping” or “tweak it until it goes away” issues? Is this a judgment call? If so, can it be applied in the mapping transition of all models? What would be the guidelines?

FEMA should not accept “tweak it until it goes away” methods. Some modelers obtain results using different software models and then compare the results to find the more ‘convenient’ answer. When conducting this type of comparison, one should verify that the results from the first model are correct and they are not “forcing” the other models to get the same results. Thus, this should not be a “judgment call.” FEMA should consider including a full disclosure requirement to ensure work is done in a proper fashion. These are serious matters and should be treated as such.

16. How are issues related to model calibration and general lack of calibration data to be addressed?

A more detailed sensitivity and uncertainty analysis can solve issues in relation to model calibration and general lack of calibration data. If the model results are not too sensitive to values of certain parameters, that condition could indicate that results are robust and that the lack of data does not have devastating effects influencing the calibration/validation of the model. However, this issue should be taken with a lot of care since all models are robust as long as they have been properly calibrated and validated. If these stages have not been adequately done, due to lack of calibration data, the results of the model cannot be considered reliable enough.
17. How are modelers trained? Should there be some sort of certification program?

Modelers should have reasonable education and experience, and FEMA should require certain level of education, and amount of hours of attending seminars or taking classes related to floodplain modeling. FEMA could also develop an Accreditation Program for modelers.

18. How should issues related to run times be addressed?

In the authors’ opinion, the issue of run times is something of less importance as long as the result is fine. Mesh convergence studies need to be developed, and results need to be checked, regardless of the time they take. This might be the difference between a successful or erroneous study.

19. What kinds of hardware and software investments must FEMA make to view and use the models?

FEMA should require modelers to provide all input files to FEMA, including DEMs and Manning’s n values, for instance, as detailed above. Also, the code developers should provide the software to FEMA so they can review the model and duplicate results. Hardware requirements are then similar to those employed by the consultants in their runs.

An option is to have input and output files in a standard format so that results could be duplicated with a different model. USACE input files are standard text files and can easily be read. USACE has binary output file format using hdf5. The hdf5 file format is a data model, library, and file structure for storing and managing data (http://www.hdfgroup.org/HDF5/). Also, USACE uses hdf5 for some input data files. USACE standard for time series data and paired data is DSS (http://dssdatacenter.com/). It allows users to develop rating curves and plot data, but DSS is not good for larger masses of gridded data like hydraulics and hydrology (H&H) model data so USACE uses hdf5.

It is likely best to require that software vendors supply raw data in a general format. Most programs read ARC-Grid format but these data sets are too large. One recommendation might be that all the boundary condition data can be in DSS and output data can be done with hdf5.

20. How can model results be most effectively communicated to the public?

One example of how model results can be most effectively communicate to the public is by outputting files to Google Earth and superimposing images with numerical results. Also, colored depth inundation maps, velocity maps, hazard (e.g., depth times velocity) maps, water surface profiles, and computed hydrographs can be used to communicate model results to the public.
In some cases, it could be argued that the public does not have a general understanding of floods to interpret the significance of the results. In these cases, it could be judged as best to educate the public on how the results were obtained and how they should be interpreted. However, there is ample experience with weather maps which suggests that people understand from the news services complex systems such as evolution of storm fronts and similar atmospheric structures; thus, similar approaches should be followed in the case of floods.

21. How are LOMA determinations made for homeowners requesting a LOMA? How are BFEs established for a specific location?

According to the FEMA website, if a property owner thinks its property has been inadvertently mapped in a Special Flood Hazard Area, he/she may submit a request to FEMA for a Letter of Map Change (LOMC). A SFHA is defined as the area that will be inundated by the flood event having a 1-percent chance of being equaled or exceeded in any given year. A LOMC reflects an official revision/amendment to an effective Flood Insurance Rate Map. If the LOMC request is granted, property owners may be eligible for lower flood insurance premiums, or the option to not purchase flood insurance. Applicants can now use the Online LOMC, an internet-based tool, to easily request a Letter of Map Amendment. A LOMA is a letter from FEMA stating that an existing structure or parcel of land - that is on naturally high ground and has not been elevated by fill - would not be inundated by the base flood. This new tool is a convenient way for applicants to upload all information and supporting documentation and check the status of their application online. Users can submit LOMA requests through this tool instead of filing the MT-EZ paper form via mail (http://www.fema.gov/letter-map-amendment-letter-map-revision-based-fill-process). Base flood elevation contours on the FIRMs/DFIRMs are the primary reference. Also, the computed water surface elevations at grid cells are preferred in LOMA determination.

The computed elevation from a flood model to which the floodwater is anticipated to rise during the base flood (100-year flood) is the Base Flood Elevation (BFE). The BFEs on a Flood Insurance Rate Map (FIRM) are rounded to the nearest foot.

Base flood elevations (BFEs) shown on the FEMA Insurance Flood Map (FIRM) are directly related to elevation data shown on the flood profiles (from the flood insurance study). Within the limits of map accuracy, it could be possible to obtain the same elevation whether you a map or profile is used (please see discussion below). However, the flood profiles should always be used to determine flood elevations along rivers and streams (FEMA).

Currently FEMA uses the level cross-section water surface elevations predicted by steady-state models to establish the BFE. A 2-D model will produce an uneven water surface elevation across the same location, however rather than providing data only at cross-sections the 2-D model will provide the results calculated for the nodes (or center) of each defined cell in the discretized grid. Further the values are then typically smoothed along the predicted flooded boundary. Likewise the longitudinal profiles used
by FEMA, which in 1-D would have been the same across each cross-section, will now vary both longitudinally and laterally. The result will affect how FEMA will need to address the results.

22. Should the FEMA maps make some effort to display the uncertainties? If so how?

In opinion of most of the BRP members, FEMA maps should display uncertainties and results should be shown to include a range of parameters as a result of the sensitivity analysis. Currently, when FEMA produces a new map, FEMA Headquarters also produces depth grid maps as well, and provides the probability of 30-year flood (life of mortgage) and 100-year flood. Maps also include both the 95% and 50% confidence intervals. The confidence intervals define the uncertainty of the proposed solution. During the BRP discussions, it was mentioned that this could evolve to more sophisticated uncertainties maps. One of the plots which could be offered could be stage-frequency curves; maps with range of uncertain parameters such as the roughness coefficients. An important aspect is to show flow depths in relation to the location of the free surface, so the depth does not need to be computed by the person “reading” the map.

23. Can FEMA address some of the uncertainties with extra freeboard requirements that are specified in local ordinances?

Freeboard is a factor of safety usually expressed in feet above the base flood level. Freeboard can compensate for many unknown factors such as wave action, flood intensity increases due to upstream watershed development, uncertainties in flood modeling, topography, and mapping (FEMA). NFIP does not require freeboard but it is recommended as it increases safety and significantly reduces flood insurance rates. For example, for one foot above the base flood elevation, insurance rate can be reduced from $0.45 for $100 coverage to $0.26. Nineteen states have stricter construction requirements than NFIP. Freeboard requirements for communities range from six inches to four feet (FEMA). FEMA can address some of the uncertainties with extra freeboard requirements because it adds to the degree of safety, but ultimately it should be up to the community to decide if it is worth building at higher elevation to reduce flood insurance rates.

24. FEMA requires that new models tie into existing models to within 0.5 feet. What challenges does this pose for 2-D models.

The 2-D models should duplicate the calibration made with 1-D models. The newer model needs to prove that either better data, better calibration, and better decision making process was used than the previous model. The baseline model should tie into the existing model but if the modeler can prove the new model has produced better results with better data and provide documentation of why it is better, then it should be acceptable.
25. How are profiles and profile base lines established when 2-D models are used?

Since the hydraulic model establishes the flood elevations at each cross section in a 1-D model, flood elevations at the locations between the cross sections need to be established. The elevations at the cross sections are then plotted on a graph and the plotted points are connected. This is referred to as the flood profile (FEMA). In a 2-D model, each grid cell will have a center point, corner points, or edge points where elevation is calculated.

In order to show results, profiles can be requested down the centerline of the main channel. It could also be requested down the center of mass of the left overbank and right overbank. For more complex flow paths, in which there are many flow paths, FEMA could request centerline of the main channel, and also other dominant flow paths, i.e. the ones with the most flow.

26. How does FEMA cost effectively integrate the regulatory 2-D mapping results with the non-regulatory 2-D mapping results? (Edges are smoothed for regulatory, not smoothed for non-regulatory.)

After a flood study is completed, the flood elevation data from the model are transferred onto a map showing ground elevation data. This is called a topographic map or contour map because points with same elevation are connected by contour. Most of the topographic maps are provided by the United States Geological Survey (USGS). The base flood elevation from cross sections and profiles are plotted on the topographic map. The contour lines are used as a guide to help draw the floodplain boundary lines. The floodplain boundary lines are drawn connecting the plotted points (FEMA).

New modeling techniques are producing smooth edge boundaries directly from the model computations, so they are not computed under the assumption that the entire cell is wet, and they are not visualized or mapped with the entire cell wet. Models that make the assumption that the entire cell is wet or dry based on a depth threshold provide more approximate results for the floodplain boundary, and they must have a post processing phase to produce a reasonable boundary. This should be provided by the software developer.

27. Can Alluvial Fans be adequately/safely modeled in 2-D software? If so, what guidelines are necessary when doing so?

1-D or 2-D models typically assume rigid bottom boundaries during the simulation of the flooding event. However, this is not generally the case when significant flooding events occur on alluvial fans. Sediment erosion and deposition can occur and evulsions/break out flows can occur with no way to accurately predict where. Even the state of the art multi-dimensional sediment transport models cannot accurately predict what happens on an alluvial fan during a significant event. The state of practice is not able to perform predictive modeling on an alluvial fan, unless the event causes no significant movement of sediment, and stays within its current channel banks. This generally does not happen for significant events.
More importantly, an alluvial fan is a highly porous body of material where infiltration needs to be taken into consideration. Modeling infiltration on alluvial fans has a very high degree of difficulty and uncertainty surrounding initial moisture content. Different models implement various routines to remove water but the a priori knowledge to properly predict infiltration is fleeting. Infiltration will continue to produce highly uncertain results.

For inactive alluvial fans, 2-D modeling can be a useful tool for the hydraulic analysis. The floodplain could be modeled by combined hydraulic and geo-morphological analysis. As an example, the Cornet Creek Watershed and Alluvial Fan Debris Flow Analysis, conducted by Mussetter Engineering Inc. (MEI) for Town of Telluride, used FLO-2D to model the Cornet Creek and the adjacent alluvial fan (Mussetter Engineering, 2012). The FLO-2D model of the alluvial fan used 4,200 elements and a 50-foot grid size. Channel cross sections were obtained from the previously developed HEC-RAS model of Cornet Creek (MEI, 2008). At channel grids where no surveyed data were available, the cross sections were interpolated between the surveyed cross sections using the PROFILES program in FLO-2D. “The clear-water inflow hydrograph is input to the FLO-2D model at a specified element near the head of the Cornet Creek debris fan. The clear-water hydrograph is then bulked with sediment using a developed sediment concentration (by volume, CV) hydrograph to represent the mudflow hydrograph. The total volume of the water and sediment in a mudflow can be determined by multiplying the clear-water volume by the bulking factor, where the bulking factor is defined by: \( BF = \frac{1}{1 - CV} \). For example, a sediment concentration of 10 percent (CV=0.10) creates a bulking factor of 1.11, indicating the flood volume is 11 percent greater than if the flood was considered to be only water.

The sediment concentration hydrograph was developed to represent the likely variation in sediment concentration throughout the storm hydrograph based on previous studies and recommendations provided in the FLO-2D manual. More information regarding this example can be found in Mussetter Engineering (2012).”

28. Can information like velocity grids be used to assist OES in emergency evacuation planning? If so what is the best way to use this information?

Maps should include water surface contours, depth contours, and velocity grids. Velocity grids can indeed assist OES with emergency evacuation planning.

Velocity grids are very important. Velocity grids and their associated flood depth grids are required for new Risk Mapping Assessment and Planning (MAP) non-regulatory products and datasets. During low-velocity floodwaters, a building must resist primarily hydrostatic pressures from saturated soils and floodwaters. This situation is typical of broad, flat floodplains and floodways along low-gradient rivers and streams. During high-velocity riverine and coastal floodwaters, a building must also be able to resist hydrodynamic forces and impact loads. High-velocity floodwaters are found in floodways along steeper-gradient rivers, sheet-flow down slopes, and coastal areas with storm surge and waves. This is why velocity grid data can be used to improve building
performance during floods. Also, proper foundation design can enhance overall building stability and performance (FEMA). Velocity grids can also help OES determine safe routes during flood evacuation as well as provide guidance for an evacuation priority.

Velocity grids can be established in HEC-RAS, 1-D model, in multiple ways. It can be a single mean velocity value for lettered cross sections and/or hundreds of velocity values for every modeled or interpolated cross-section. In a 2-D model, velocity grids show both magnitude and direction. Sometimes, for a 2-D model velocity grids are developed outside the specific hydraulic model based on another grid, such as depth grid. This is why consistent and precise documentation is important so FEMA knows what method and process was used (FEMA).

29. What are some of the ways this information can be displayed electronically besides through the FEMA maps?

This information can be displayed on Google Maps to facilitate locating where the extreme values of the depths and velocities are located.

30. How do models match the effective modeling? Or should they? 2-D results are different from 1-D but has to match the effective model;

Effective model is the FEMA model that is used to create current FEMA maps. The level of data for the effective model, calibration process, and decision-making process should be properly documented. The newer model needs to prove that either better data, better calibration, and/or better decision making process were used than in the previous model. The baseline model should tie into the existing model but if modeler can prove the new model has produced better results with better data and provide documentation of why it is better, and then it should be acceptable. Consistent and precise documentation for data used, calibration process, and decision-making process should be required for new models.

31. Is it possible to develop a test that compares level of detail with needed accuracy?

To compare the level of detail with accuracy the modeler can conduct a mesh convergence test and compare the results in terms of the plot of mesh size versus relative error.

32. If you run steady state and unsteady model for same reach, is there residual risk in the area that is NOT in the SFHA for the unsteady model?

Since 1-D steady state model is more conservative as opposed to the unsteady state model, we believe that the risk of not accounting flooded areas in the 1-D model is smaller. On the other hand, it could show areas as flooded which would not appear as such in unsteady solutions. The unsteady solution would more accurately calculate the flooded areas in general and the Special Hazard Flood Area (SFHA) in particular. As
mentioned earlier, the modeler needs to also determine if the level of data available will support the unsteady model.

33. Since 2-D calculates flood depth more accurately than 1-D, should 1% boundary be mapped just to the <1 ft depth threshold? Or all the way to 0 ft depth.

Since 2-D calculates flood depth more accurately, in principle it should be mapped to zero feet.
As discussed in issue 1, please keep in mind that 2-D calculations are only as good as the data used to produce the model, the software, and the modeler. If the data do not support the 2-D model, a 1-D model should be used. Also, if better data, better calibration, and better decision making process were used for the 2-D model then it should definitely produce better results.

34. It is not possible to get a Duplicate Effective (DE) model when revising a 2-D model. Sometimes this is due to a versioning issue. Is a DE model needed if you are revising a 1-D study using a 2-D model?

This issue follows the same requirements as issue 30. The level of data for effective model, calibration process, and decision-making process should be properly documented. The newer model needs to prove that either better data, better calibration, and better decision making process was used than the previous model. The baseline model should tie into the existing model but if the modeler can prove the new model has produced better results with better data and provide documentation of why it is better then it should be acceptable. Consistent and precise documentation for data used, calibration process, and decision-making process should be required for new models.

35. If flooding is contained in the street, can it be mapped as Shaded X?

Shaded X zone is referred by FEMA as Moderate and Minimal Risk Areas. Shaded X refers to moderate to low risk flood areas. Shaded X areas are within 500-year flood, within 100-year flood where flood depths are less than 1 foot, or area where contributing drainage is less than 1 square mile (FEMA). Low risk flood areas include streets which are made for drainage and since streets have no property damage so they can be referred to as Shaded X.

36. If the depth is averaged, can detailed depth data from a 2-D study be used to evaluate the application?

In a 1-D model the depth is averaged over the entire cross-section, while in 2-D the average is over the cell. Using detailed depth data from a 2-D study to evaluate the application depends on the terrain and the computational cell sizes used to represent that terrain. If the model has very large cells, then the average depth is not very accurate for flood mapping. But if the cell sizes are small, then using average depths is adequate for mapping accuracy.
37. Should there be a certificate program for modelers? If there is, what should it include? Who should administer the program?

There should be no need for a certificate program for modelers because a Professional Engineer signs off on the model when it is valid. The focus should be on the selection of a qualified modeler and the review of the model. Here we need to differentiate training versus certification.

38. Collecting field data. Field data collection for a 2-D study would be different that for a 1-D study. In addition to collecting data linearly downstream, data should be collected across the horizontal extent of the floodplain.

For a 1-D model cross-section data may only be measured and developed at a few critical locations along the river and floodplain. Intermediate cross-sections are then interpolated and any errors are corrected in calibration with roughness coefficients. For 2-D models it is more appropriate to have higher resolution data from which smaller cells can be established in the grid.

There is an issue with data collection and how data are used. For instance, topographic data required to construct bathymetry need to be used carefully in order not to diminish the intrinsic accuracy with manipulation in the model.

39. How do you ensure that the modeler’s assumptions are fully documented? How do you ensure that these assumptions are collected and stored with the correct model version?

The modeler’s assumptions must be reasonable and documented; the modeling approach should be easy to understand and be easily reproduced. Information to provide to FEMA should include the study methods, reason for method selection, input data and parameters, and justification for computer flood parameters. To ensure that the modeler’s assumptions are fully documented, FEMA needs to define a common format/standard and modelers need to define their hypothesis clearly in the report. This can also be conveyed through generic GIS layers.

40. Should FEMA give more latitude to communities that are actively advancing or innovating in the area of 2-D studies?

In principle we believe this should not be a policy in the short term, but a possibility in the long term, if the mentioned “latitude” is clearly specified. FEMA can certainly take advantage of early adapters to streamline the eventual guidelines and standards.

41. Should there be an on-line forum?

Yes, we believe that there should be an on-line forum, as happens with all good software packages. Users seem to feel fine with this. The on-line forum could be on both the FEMA and FMA websites. Also, experts can weigh in on issues, and can correct misleading information that could be posted by the less informed model users. Cases of
successful flood mapping and reviews could be uploaded to those websites as examples, with the due permissions of the developers of those modeling studies.

42. Can FEMA establish an open source 2-D software system that anyone can adapt and improve upon? Something in line with Linux system?

Currently, there is no successful open source H&H code. The basic approach of open source is to have the source code open to anyone to modify, which is difficult to justify because of the amount of time and cost to develop it. For example, Linux is open source but took a lot of money and time to develop and most successful applications are commercial modifications.

A less-open approach to open source is to use plug-ins, DLLs, or library approaches. The main code is not exposed to change (solver) but allows modelers to write around the solver to modify the code to fit the user’s needs and allows the modeler to be creative. Software can have open MI (Model Interface). HEC-RAS is open MI compliant and has been linked with MODFLOW. The modeler needs to have great understanding of both programs and open MI to connect/marry the software. FEMA should keep in mind that it takes a lot of money and time to test software to make it open MI compliant.

43. Can FEMA collaborate with other agencies to develop common set of guidelines, or is there a way for the agencies to provide input?

This research effort intends to provide direction for such guidelines to FEMA. Also, the guidelines should be open for public comments so other agencies can provide input before finalizing the guidelines.

44. How can FEMA provide opportunities for communities to provide input and comment on new 2-D policies or procedures?

FEMA can facilitate opportunities for the modeling community to provide input and comment on new 2-D policies or procedures through disseminating the discussions of the BRP, by hosting extended 2-D Symposiums at FMA conferences such as that at Anaheim, and through the California Water and Environmental Modeling Forum (CWEMF). The CWEMF is a non-profit, non-partisan organization whose mission is to “increase the usefulness of models for analyzing California water-related problems.” CWEMF facilitates an open exchange for California water issues, resolves disagreements in professional settings, and ensures technical work considers the needs of stakeholders and decision makers. CWEMF also maintains a “modeling clearinghouse” that provides an open forum for the improvement of models and modeling information (CWEMF 2013). In addition, the guidelines should be open for public comments so other agencies can provide input.
45. “If it’s not too deep, it’s 2D.” – Canadian colleague

The statement above is valid in general, because in shallow water bodies the vertical accelerations are in general negligible. 2-D approaches implicitly assume that the distribution of horizontal velocities in the vertical direction does not differ significantly from a “rectangle” (i.e., the value is the same). This assumption is more likely to hold in shallow water bodies.

46. All equations and models are approximations: 1-D, 2-D or 3-D. Must understand how Manning’s values vary from 1-D to 2-D. In 2-D, Manning’s is a function of the cell/grid rather than (average for) the entire cross-section (as in 1-D). 1-D – will get similar answers from different folks; 2-D – will likely get different results because we differ on really small inputs (like Manning’s) which could drastically change the results (have to know what’s going on in field for 2-D because it is calculated at the cell level).

We agree essentially with these assessments. In first place, it is very clear that all models are approximations, even 3-D models.

Manning’s $n$ is basically different in 1-D and 2-D. The value of Manning’s $n$ is NOT necessarily an average of values in the cross section, but rather a weighted value in this cross section. Yen (2002) provided a table, included as Appendix 3, indicating all approaches to compute the cross-sectional Manning’s $n$ when having different roughness values across it.

Model results developed by different modelers are in general different, regardless of the dimensionality of the model. Each modeler might select different values of Manning’s $n$, and approach the model in different ways. Models are by definition a representation of reality. That representation can differ among codes and the way in which different modelers find ways to represent reality with the features of those codes.
Chapter 4: Numerical Model Verification

1. Introduction

Growth in computational power in recent decades, in addition to cost and scale effects of physical models, increased the popularity of computational simulation in design, analysis, and optimization of engineering projects (Roache, 2009). Unfortunately, coding and simulation errors are not uncommon among computational models. These computational errors may seriously affect the professional career of a code user (see for example: Miller, 2006; Keenan, 2012) or they could end up in extreme engineering tragedies. For example, faulty numerical modeling of snowmelt led to a severe flood in Colorado River in 1983 (Hatton, 1997). Another catastrophic case happened on August 23, 1991, in the North Sea where a 700 million dollar oil rig sank due to inaccurate design of a condeep\(^2\). Post-audit of the numerical model showed that the failure was due to a problematic finite element code computation by NASTRAN (Collins et al., 1997). The examples above strongly suggest that, without reliable “verification,” modelers should not put too much trust in numerical modeling.

In the literature of numerical modeling of Partial Differential Equations (PDEs), verification is defined as “solving the equations right” versus validation which is defined as “solving the right equations” (Roache, 1997). Verification is the set of activities which has to be done to build confidence in the results of a numerical model. Verification has two fundamental aspects: a) solver (code) verification and b) solution (calculation) verification. Solver verification is a procedure of checking the code for bugs, inconsistencies and imperfections (Roache, 2009; Oberkampf and Roy, 2010). Solver verification is a one-time activity which must be conducted by code developer(s) or whoever modifies a code for his/her particular use. There is no need to repeat solver verification activities unless the source code is changed. On the other hand, solution verification is the process of quantitatively assessing the error in numerical solutions which has to be performed in each and every application of the code which is considerably changed compared to each other. Solution verification is among the duties of the model user. In the following section, the techniques of verification in computational fluid mechanics are briefly discussed.

2. Verification Criteria for Numerical Solvers

Formal proof of correctness of a solver is not forthcoming; however, there are activities which are able to provide a framework for a reliable testing of a computer code. The criteria for code verification are suggested as follows (in order of increasing rigor, Knupp and Salari, 2003):

- Expert judgment after extensive use in wide range of problems.
- Error quantification.
- Consistency-convergence test.

\(^2\) Abbr. concrete deep water structure
• Order of convergence test.

The expert's judgment is used when results are given to an expert and he/she will assess it to make sense. This criterion is very loose and with all the respect for Dr. Patrick Knupp as one of the pioneers of "V&V" we do not recognize it as "formal" verification method. Error quantification consists in assessing misfit between numerical results and a reliable benchmark (reliable benchmark will be discussed in the rest of this section). This criterion is more advanced and objective compared to the expert's opinion; however, there are many cases in which there is a hidden bug in the code, inconsistency between parts or a mis-implementation of a solver where the issue does not amplify error such that modeler can notice that there is problem evident (for example, see Ateljevich et al., 2011). The third level is checking if the solver results "converge" as the discretization size in time and space shrink. Finally, the most rigorous measure is similar to the third level (consistency and convergence test) while the quantitative measure of convergence is being studied. Comparing observed order of convergence with formal order of convergence of the scheme is the most restrictive verification test and many researchers agree on the extreme efficiency of this method in uncovering imperfections in coding and implementation of numerical software (Roache, 1997, 2009; Knupp and Salari, 2003; Wang et al., 2008; Graziani, 2008; Oberkampf and Roy, 2010; Ateljevich et al., 2011; Zamani and Bombardelli, 2014). The next section provides insight into the techniques of performing mesh convergence test.

3. Mesh-Convergence as a Verification method in Computational Hydraulics

The crux of mesh convergence test is rooted in the Lax Equivalence Theorem. This theorem states that convergence implies both consistency and stability in the analysis of Finite Difference Method of numerical discretization of PDEs (Lax and Richtmyer, 1956). Later on many researchers employed the same concept of Lax Equivalence Theorem for Finite Element Method (FEM), Finite Volume Method (FVM), and Boundary Element Method (BEM) for the purpose of code verification (Roache, 2009). Mesh-convergence tests are well-accepted methods for verifying numerical solvers in computational fluid mechanics. They consist of reducing the mesh size (both spatial and time steps) and in checking the evolution of the ratio of error metrics. Grid-convergence tests check if the formal grid accuracy of the numerical scheme is reproduced by the code. Coding bugs and implementation errors could be detected via this vehicle. The question here is how to determine error when conducting mesh convergence tests since, for most of the PDEs of interest in engineering, there is no analytical solution. The general roadmap for any verification study (solver or solution) is shown in the Figure 1. The first preference for conducting mesh convergence studies is the Method of Exact Solution (MES). In this method, the evolution of errors is observed based on an exact solution as a benchmark (Zamani and Bombardelli, 2014). MES is the most convenient code verification method. Unfortunately, we are not fortunate enough to have an analytical benchmark for all system of PDEs. Therefore, other alternatives of code verification should be considered. In the absence of analytical solutions, if the source
code is open to modifications, the Method of Manufactured Solution\(^3\) (MMS) could be utilized (Roache, 2009; Zamani and Bombardelli, 2014). MMS has its own drawbacks and it must be performed with special care.

MMS is considered a very strong error probe. If the source code is not accessible, which is the case in most of the commercial CFD packages, we have to do "Black Box Testing" (Ateljevich et al., 2011). Black Box Testing constitutes a set of practices for computational code testing in the situation where only input and output of results are available. The first common method of Black Box Testing is checking the error evolution versus the results of the exact same problem (similar initial and boundary condition) solved with higher resolution, already verified code on the dense mesh. And, finally, in the case where the corresponding analytical solution does not exist, access to the source code is restricted, and another reliable high resolution solver of the same governing equation is not available, the verification study must be performed via Richardson Extrapolation (Roache and Knupp, 1993). Careless application of the above mentioned

---

\(^3\) In the CFD verification literature variations of this method is called "Man Made Solution" or "Prescribed Forcing Method" which are identical to the MMS in principles.
techniques may result in vague outcomes or erroneous interpretation. The next part discusses the assumptions and prerequisites of mesh convergence studies.

4. Ambiguity in the Mesh Convergence Test

Due to the fact that the derivation of the Lax Equivalence Theorem makes several assumptions, any violation of these assumptions may affect the results of the mesh convergence study. A modeler may find an order of convergence lower than the expected order of convergence in a simulation because of the issues not related to bugs or imperfection of the code. In the following paragraphs, we briefly discuss common pitfalls a modeler may encounter in the verification process:

- Smoothness of the solution: shock presence or shock formation in a numerical solution may strongly impact the convergence of the solution in various ways. For example, a shock may trigger an auxiliary numerical scheme to deal with the effect of shock (van Leer, 1979) or it may directly affect the order of global convergence measures (Zamani and Bombardelli, 2014).
- Iteration error: the final outcome of all discretizations of a PDE such as FEM, FDM, FVM, etc. is a sparse system of algebraic equations. In real scale problems, this sparse matrix is solved by iterative methods (Saad, 1996). In a mesh convergence study, it is usually assumed that the iteration error in the solution is at least two orders of magnitude lower than discretization error. However, this issue could cause crucial confusion in several cases (Roache, 2009).
- Difficulties induced by the scale of the problem: if the test problem for the mesh convergence study is chosen in such a way that one phenomenon is dominant and its effect is larger than the others in orders of magnitude, the imperfection in the low effect phenomenon may be concealed by the dominant effect. For example, code verification of a transport solver should not be performed in very low and very high Peclet numbers (Zamani and Bombardelli, 2014).
- Appropriate convergence zone: a mesh convergence study has to be done for mesh sizes which are fine enough to capture the problem completely. In coarser mesh sizes, oscillatory convergence-behavior occurs which may cause serious misinterpretations of the mesh convergence results. Figure 2 shows the convergence-behavior versus the mesh size.
Figure 3: Schematic of asymptotic zone of convergence (after Graziani, 2008). In the zone one, round off error and iteration error take over; zone two is appropriate zone for numerical modeling and mesh convergence test; and zone three is mesh sizes where the problem is under resolved.

- Problem of PDE identicalness: a mesh convergence test has to be done on exactly identical differential equations which mean the exact same geometry, initial condition and boundary conditions. In that regard, implementation of boundary-fitted cells may cause problems of solving slightly different domains in a mesh convergence study (Wang et al., 2008). Other common problems in environmental fluid mechanics are cases where, with mesh refinement, a different physical problem is solved, like Large Eddy Simulation (LES) turbulence models (Roache, 1997).

- Problem of order of accuracy of other pre/post-processing algorithm(s): any extra pre-processed or post-processed value which is used in verification must be computed via a high order method. For example, in the case of Richardson Extrapolation of lid driven cavity flow problem (example one below), since the formal convergence order of the Navier-Stokes solver we used is 2\textsuperscript{nd} order, the interpolation of the fine results to the location of coarser grid points must be developed with an interpolation method of an order three or higher. Although this issue seems trivial, overlooking it may cause confusion in code verification activities.

- Nature of integrated values: the ultimate goal of any verification study is uncovering bugs or weaknesses in the model. Along that line, any fragmented information on the convergence behavior would be more fruitful than the integrated information. For example, local values of velocity convergence will reveal more information than global values of the same phenomenon. Therefore, we suggest separate analysis of the boundary and inner domain values of interest.
for rigorous mesh convergence studies. Global metrics for mesh convergence studies are given below.

5. Error Measures and Their Qualification

When dealing with numbers we can identify them as being large or small. Arrays and vectors are functions of many elements but we need to measure their size - an index for them to be small or large. Norms are used as a measure in this context. Realizing that the size of a vector or matrix should depend on the magnitude of all elements in the arrays, we arrive at the definition of matrix norms. By definition, a norm is a single number that depends on the magnitude of all elements in the matrix. A norm of matrix \( v \) should satisfy the following three conditions:

\[
\|v\| \geq 0 \text{ and } \|v\| = 0 \text{ if and only if } v = 0 \tag{21}
\]

\[
\|cv\| = |c|\|v\| \text{ for any scalar } c \tag{22}
\]

\[
\|v + w\| \leq \|v\| + \|w\| \text{ for matrices } v \text{ and } w \tag{23}
\]

The following three vector/matrix norms are commonly used and called the maximum norm (infinity norm), energy norm (second norm), and first norm (taxicab norm):

\[
L_{\infty} = \|v\| = \max |v_i| \tag{24}
\]

\[
L_2 = \frac{1}{n} \left( \sum_{i=1}^{n} |v_i|^2 \right)^{\frac{1}{2}} \tag{25}
\]

\[
L_1 = \frac{1}{n} \sum_{i=1}^{n} |v_i| \tag{26}
\]

We can define \( v \) as any state variable of interest in fluid mechanics (velocity, pressure, depth, temperature, etc.) or the error between \( v_{\text{num}} \) and \( v_{\text{exact}} \), or error between coarse and fine solution of state variables \( |v_{\text{coarse}} - v_{\text{fine}}| \). \( L_\infty \), Equation (24), is the most restrictive norm for mesh convergence studies, as it produces the "worst" situation in the domain so the lower "convergence rate" will be obtained by employing this norm. In contrast, \( L_2 \) and \( L_1 \) act as a "smoother" norms, so the convergence rates obtained via these norms are less restrictive than the convergence rate calculated via the infinity norm or even the original values of the state variable of interest.

6. Observed Order of Accuracy

"Observed order of accuracy" is the actual order of accuracy delivered by a numerical solver (versus the "formal order of accuracy" which is the order of the employed numerical scheme). For the sake of simplicity, we assume cases in which a benchmark for error calculation exists. Calculation of the order of convergence in cases in which the exact solution is not available is conceptually similar and details can be found in Knupp and Salari (2003) or Roache (2009). Let's assume a Taylor series expansion for the
numerical solution to the discrete equation \( u_h \) based on the grid spacing \( h \), as \( h \) goes to zero:

\[
    u_h = u_h \big|_{h=0} + h \frac{\partial u}{\partial h} \big|_{h=0} + \frac{h^2 \partial^2 u}{2 \partial h^2} \big|_{h=0} + \cdots + \frac{h^{p-1} \partial^{p-1} u}{(p-1)! \partial h^{p-1}} \big|_{h=0} + O(h^p)
\]  

(27)

The above equation is the basis for calculating \( p \) (observed order of accuracy). As the mesh size tends toward zero, the higher order terms of error become negligible, therefore we may write:

\[
    \epsilon_h = C h^p + O(h^{p+1}) \approx C h^p
\]  

(28)

where \( \epsilon_h \) is the error on the mesh size \( h \), and \( C \) is a constant which depends on the numerical method of discretization and other factors. If we repeat the calculation for mesh size of \( 2h \) we would have:

\[
    \epsilon_{2h} \approx C (2h)^p
\]  

(29)

eliminating \( C \) between the above two relation we can calculate "\( p \)" (observed order of convergence) between the two mesh sizes as follows:

\[
    p = \frac{\ln(\epsilon_{2h} / \epsilon_h)}{\ln(2)}
\]  

(30)

This concept can be generalized to non-integer mesh refinement ratio \( r = \frac{h_{coarse}}{h_{fine}} \) as well. For more details and derivation, please see Oberkampf and Roy (2010), Chapters 5 and 8.

7. VAVUQ: A Toolkit for Verification and Validation, and Uncertainty Quantification in Environmental-Fluid-Mechanics Simulations

To conduct code and calculation verification of models of interest in Environmental Fluid Mechanics, a Matlab code was developed at the Department of Civil and Environmental Engineering of the University of California, Davis. VAVUQ has a GUI (graphical user interface) with the capability of loading, visualizing and post-processing files in text, binary and excel format. The toolkit can be utilized for the following purposes:

- Code and calculation verification studies, including: Richardson extrapolation, verification against a benchmark: MES, MMS, and Dense Mesh Solution or Cross-solver-verification. VAVUQ is able to post-process results of structured (uniform and non-uniform) and unstructured meshes. In addition, this toolkit processes the verification in integrated (globally on all cells) and disintegrated domains (point-wise, inner, and boundary cells) and provides individual log of the verification for each part (please see examples one and two).
- Model validation: VAVUQ calculates common metrics of quantitative model skill validation based on the numerical results and measured benchmark values:
RMSE, Scatter Index, Bias Index and Coefficient of Determination (please see example three).

- Visualization of the error distribution over the domain via high-order spline extrapolation (please see example four below).
- Uncertainty Quantification: VAVUQ calculates the upper and lower limit of the numerical simulation error for 99% and 95% confidence in the results based on the Roache's Grid Convergence Index "GCI" (please see example five).

VAVUQ is coded in the most general form and it can be employed for verification studies of any PDE of interest in water resource engineering including: Navier-Stokes Equation, Saint-Venant's Equation, Richards Equation, Burgers' Equation, etc. In the following sections, some examples of the application of VAVUQ are presented.

8. **Example one: Richardson Extrapolation**

A two-dimensional lid-driven cavity flow is a classical problem for code testing associated with Navier-Stokes solvers (Ghia et al., 1982). The problem domain and boundary conditions are straightforward. The standard case is a viscous fluid inside a square domain with Dirichlet boundary conditions on all sides, with three stationary sides and a moving lid (with velocity tangent to the side). The schematic of this physical set up is shown in the Figure 3. In this example, we solved a 2-D lid driven cavity flow with a commercial CFD code (Flow-3D).

![Figure 4: Schematic of lid-driven cavity flow (after Ghia et al., 1982)](image)

We solved the flow with four different mesh sizes and conducted Richardson extrapolation on three state variables (pressure, x-wise velocity and y-wise velocity components). Given that local discontinuity may affect mesh convergence results, the study was conducted for three categories: a) Inside the domain b) on the boundaries and, c) globally. Table 1 shows the formal order of convergence of the pressure and the two velocity components inside the domain, on the boundary and globally produced by VAVUQ. The results of the mesh convergence study show that in the inner-domain cells the solver almost reaches 2nd order convergence in pressure and approximately $p \approx 1.7$ in
velocity simulation. However, in the boundary cells, the order of convergence dramatically drops to near zero. Direct implementation of velocity boundary condition on the boundary cells next to the lid explains why the lowest convergence ratio happens in lid-wise velocity component. Lastly, the global order of convergence is something in between the inner domain and outer skin of the domain.

Table 5: Convergence rates of two dimensional lid driven cavity flow on the grid resolutions of $h_4 = 92 \times 92; h_3 = 128 \times 128; h_2 = 182 \times 182$ and $h_1 = 256 \times 256$ and $r_{12} = \frac{h_1}{h_2}; r_{23} = \frac{h_2}{h_3}; r_{34} = \frac{h_3}{h_4}$.

<table>
<thead>
<tr>
<th>Refinement comparison</th>
<th>Variable</th>
<th>Inner Cells</th>
<th>Boundary Cells</th>
<th>Global</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$P(L_1)$</td>
<td>$P(L_2)$</td>
<td>$P(L_{\infty})$</td>
</tr>
<tr>
<td>$h_3 - h_2$ - $h_1$</td>
<td>Pressure</td>
<td>2.020</td>
<td>2.077</td>
<td>2.124</td>
</tr>
<tr>
<td></td>
<td>$u$</td>
<td>1.750</td>
<td>1.645</td>
<td>1.679</td>
</tr>
<tr>
<td></td>
<td>$w$</td>
<td>1.764</td>
<td>1.650</td>
<td>1.792</td>
</tr>
<tr>
<td>$h_4 - h_3$ - $h_2$</td>
<td>Pressure</td>
<td>1.943</td>
<td>2.026</td>
<td>2.116</td>
</tr>
<tr>
<td></td>
<td>$u$</td>
<td>1.792</td>
<td>1.646</td>
<td>1.691</td>
</tr>
<tr>
<td></td>
<td>$w$</td>
<td>1.778</td>
<td>1.665</td>
<td>1.818</td>
</tr>
</tbody>
</table>

9. Example Two: Mesh Convergence Test versus a Known Benchmark (MES)

In this example, VAVUQ is used to find the order of convergence of an ADR solver versus an analytical solution of advection reaction in tidal flow in a dead-end harbor. Details of the solution and implementation can be found in Zamani and Bombardelli (2014).

Figure 5: Numerical and analytical solution of concentration plume in a dead-end harbor subjected to tidal forcing (Zamani and Bombardelli, 2014).
10. Example Three: Quantitative Model Skill Assessment (Validation)

In this case, VAVUQ is used for validation of a numerical simulation through comparison against laboratory measurements. The physical phenomenon is a two-phase flow of air-water mixture in hydraulic jump (Zamani and Bombardelli, unpublished). Figure 6 shows the profile of the free surface simulated with three different resolutions versus the measurements by Murzyn et al. (2007) at University of Southampton. VAVUQ calculated the numerical values in the positions as data points via a high-order extrapolation. Then, four common metrics of model validation were calculated to provide an objective comparison benchmark for comparison (Willmott et al., 2012; Zamani and Bombardelli, 2014). The metrics employed are as follows:

\[
Bias = \frac{1}{N} \sum_{i=1}^{N} (M_i - B_i)
\]

\[
R^2 = 1 - \frac{\sum_{i=1}^{N} (M_i - B_i)^2}{\sum_{i=1}^{N} (M_i - B)^2}
\]

\[
RMSE = \sqrt{\frac{1}{N} \sum_{i=1}^{N} (M_i - B_i)^2}
\]

\[
SI(\%) = \frac{1}{N} \sum_{i=1}^{N} \frac{(M_i - B_i)^2}{\sum_{i=1}^{N} B_i} \times 100
\]
where \( M \) refers to the results of numerical model value and \( B \) is the value corresponding to the experimental data (benchmark). This is a good example of how a finer mesh solution, which is converging, does not necessarily lead to better representation of physical phenomenon. Or in other words “verification” and “validation” are individual activities.

![Graph showing modeled free surface profile versus laboratory measurements](image)

**Figure 7:** Numerical model of hydraulic jump: Modeled free surface profile versus laboratory measurements by Murzyn et al. (2007).

**Table 6:** Metrics of models skill validation for three different simulation of free surface profile of air-water mixture in hydraulic jump.

<table>
<thead>
<tr>
<th>Mesh Size</th>
<th>Bias</th>
<th>Scatter Index</th>
<th>RMSE</th>
<th>( R^2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>500 × 80</td>
<td>0.183</td>
<td>0.353</td>
<td>0.222</td>
<td>87.4%</td>
</tr>
<tr>
<td>750 × 120</td>
<td>0.271</td>
<td>0.438</td>
<td>0.314</td>
<td>76.7%</td>
</tr>
<tr>
<td>1000 × 160</td>
<td>0.267</td>
<td>0.431</td>
<td>0.308</td>
<td>78.9%</td>
</tr>
</tbody>
</table>
11. Example four: Visualization of the numerical results’ difference in various meshes

VAVUQ is able to manipulate numerical results coming from different mesh sizes and visualize the error. This visualization is valuable to the code/calculation verification process as it helps the modeler efficiently narrow down the problematic part of the modeling. Figure 7 shows the visualization of misfit (difference) between mesh sizes of $128 \times 128$ and $182 \times 182$ for the simulation of pressure in lid driven cavity flow.

Figure 8: Difference in pressure numerical calculation between two different meshes sizes, for a lid-driven cavity flow. Most of the difference happens in the top right and top left corners next to the moving lid.

12. Example five: Numerical uncertainty quantification

By definition, uncertainty in numerical modeling has three components: uncertainty in initial and boundary conditions, $U_{\text{input}}$; uncertainty in the mathematical model, $U_{\text{model}}$; and uncertainty in numerical solution of the PDE $U_{\text{num}}$. The overall uncertainty can be computed as follows:

$$U_{\text{value}} = \sqrt{U_{\text{num}}^2 + U_{\text{input}}^2 + U_{\text{model}}^2}$$  \hspace{1cm} (35)

From the above components of the uncertainty, quantifying $U_{\text{input}}$ is out of the scope of this work. In turn, $U_{\text{model}}$ is within the scope of "model validation" not "model verification". However, VAVUQ is able to calculate the uncertainty in the numerical simulation. Then based on the most reliable results (finest mesh) it is able to calculate the lower and upper range of the numerical error. The value of these bounds of the error depends on several factor including: a) number of mesh refinements, b) required confidence interval (99% or 95%), c) mesh type (structured or unstructured), and d) the
observed order of the convergence of the solver (Roache, 2009). In general, the safety factor "FS" for numerical uncertainty quantification considered to be between one and five. And for uncertainty quantification, at least results of four different mesh sizes are needed. Figure 8 shows the relative misfit between the finest mesh and the second finest mesh in a vertical cross section of a 3D modeling of Gust Chamber (Yuen, 2014, unpublished data). Figure 9 shows the upper and lower range of the numerical difference based on the Roache's Grid Convergence Index (GCI), both calculated via VAVUQ toolkit.

Figure 9: Pressure numerical solution difference between the finest mesh and the second finest mesh.

Figure 10: Upper and lower range of numerical calculation difference based of GCI for confidence interval of 99% and four mesh refinements (structured mesh).
Chapter 5: Question Form for 2-D Software Vendors

1. a) Is the software solving the Full Dynamic Wave Equation?
   b) Diffusion Wave (Non-inertia) Equation?
   c) The Kinematic Wave Equation?
   d) Is the model restricted to only the use of the Full Dynamic Wave Equation?

2. a) Is the software based on the Finite Elements Method?
   b) Finite Differences Method?
   c) Finite Volumes Method?
   d) Other(s)? Please explain.

3. a) What is the numerical solution scheme for your software model?
   b) Is the numerical scheme explicit, implicit, or semi-implicit?
   c) What is the order of accuracy of the numerical scheme?

4. a) Is the software based on structured or unstructured grid/mesh?
   b) Triangles, quadrilaterals, or a combination?

5. a) Does the software conserve mass in a local sense?
   b) Does the software conserve mass in a global sense?

6. Is turbulence being addressed? It is important to include turbulence closure when simulating flows in urban areas, sharp corners, etc.

7. What data formats does the software support for input and output?

8. Please provide a list of peer reviewed publications that explicitly describe, justify and validate the software code’s methods.

9. a) How does the software handle flow exchanges between the 1-D main river and the 2-D overbank?
   b) How does the software handle flow exchanges between the 2-D main river and the 2-D overbank?

10. Does the software explicitly handle hydraulic structures (culverts, weirs, low levees, curbs and gutters, etc.) within the floodplain?

11. a) Is the levee breach controlled by time input or physics (stage and flow) of the problem?
    b) Or is it static, in or out?

12. Does the software display a floodplain map, or is other software needed to accomplish that? In other words, does the software come with pre- and post-processing capabilities?
Chapter 6: 2-D Flood Modeling Recommendations

1. 2-D Models

FEMA should not obligate all communities to implement 2-D models. If the data do not exist, it is **not** recommended to use a 2-D model because the model effort will not provide improved results over a 1-D representation.

**2-D models should be considered under the following circumstances:**

- When levees become overtopped or breach since the water goes in many directions.
- When the flow is truly bi-dimensional, such as in estuaries and bays, where tidal fluctuations play an important role in wetting and drying variable zones.
- In highly braided water courses.
- In alluvial fans.
- In flows past abrupt bends.
- In very flat and wide flood plains.
- Where it is important to obtain detailed velocities and there are data to support such modeling.

When using a 2-D model, FEMA should ensure the 2-D software model has:

**For the software code:**

- Peer reviewed publications that explicitly describe, justify and verify the software code methods.
- Good level of accuracy, at least second order.

**For the model application:**

- Mesh convergence tests to ensure that the numerical result is independent of the mesh, because this is a required step in the field of numerical methods. At least two, preferably three, mesh sizes should be tested.
- Adequate calibration and validation

2. Qualifications for Consultant Preparing the Model

The following qualifications should be required:

1) A M.S. degree from a Department of Civil and Environmental Engineering with a focus on modeling hydraulics and hydrology (or equivalent experience); 2) work experience of no less than 2 years on modeling floods and developing flooding maps; 3) experience with the selected software; 4) knowledge of the local system. GIS experts are more than welcome to prepare models but they need to have sufficient knowledge of hydrology and hydraulics.
3. FEMA Model Review and Model Documentation

FEMA could ask for the draft TSDN up front and ask for decision-making processes before the study starts. During the BRP meetings, it was determined that requested documents can include the discovery process, a detailed work plan (more detailed than the scope of work) and have a component in the report for the modeler experience and decision for choosing the selected software. FEMA needs to ensure that the modeler(s) developing the model properly document/s: a) the data used in the implementation of the model, b) the decision-making process in the selection of modeling approach and parameters, and c) the boundary conditions used, and d) the considerations during the calibration process. The uncertainty in the hydrology data needs to be reported (if applicable) and, within the parameters, the uncertainty in the determination of Manning’s $n$ (for example) should be properly assessed and documented. Also, FEMA can require the agency to submit their QA/QC review document as part of the standard submittal package.

FEMA could also reach out to active members of FMA and create a Reviewer Working Group. This group would help define what it takes to be a good reviewer and this group could be responsible for peer reviewing expedited flood studies and Letter of Map Revisions (LOMRs). A concern that many agencies have is that it takes a lot of time to approve a flood study or LOMR. If the flood study is approved by the Reviewer Workgroup then all FEMA would have to do is validate the documentation process since it has the seal of approval from the Reviewer Working Group. This could help expedite the process to approve an agency’s flood study. Agencies can pay a fee to fund the efforts of the Reviewer Working Group since the review of their study will be expedited. Care would need to be taken to avoid any possibility of quid pro quo relationships in these types of arrangements.

FEMA could also reach out to active members of FMA, and create a small group of current reviewers that can mentor new modelers. FEMA could pay for time of mentors and the activities they do as mentors. Mentors have to conduct webinars and seminars and train new modelers. Universities (even UC Davis) could collaborate with FEMA and FMA to set up some of these webinars and seminars.

4. Software Version Changes and Model Results

FEMA should request the consultants/communities to turn in the inputs and outputs of the model for any given study, developed with the last version of the software and in a previously approved universal format. At least, they should show that the solution is compatible with such last version of the code. In addition, the baseline model should tie into existing model results for the area/site, but if modeler can prove the new model has produced better results with better data and provide documentation of why it is better, then it should be acceptable. Consistent and precise documentation for data used, calibration process, and decision-making process should be required for new models.
Also, FEMA could request a set of benchmark tests to be passed by the new versions of software. UCD is providing initial benchmark tests with this project.

5. Flood Maps

FEMA should start incorporating high velocity hazard zones into the mapping and analysis of floodways. FEMA can also include depth times velocity results, which might be a better way to classify flow hazards. One example of how model results can be most effectively communicated to the public is by outputting files to Google Earth and superimposing images with numerical results. Also, colored depth inundation maps, velocity maps, hazard (depth times velocity) maps, water surface profiles, and computed hydrographs can be used to communicate model results to the public.
Chapter 7: References


Chapter 8: Appendix


Gary W. Brunner, P.E., D. WRE
Hydrologic Engineering Center
U.S. Army Corps of Engineers
Gary.w.brunner@usace.army.mil
(530) – 756-1104

"Where are 2D and 1D Unsteady Flow models, respectively: needed absolutely; preferred; a waste of effort?"

To answer the above questions requires additional information regarding the purpose of the model. Each river system will have site specific information that must be considered in order to answer the questions of Steady versus Unsteady flow and 1D versus 2D model. The following is a partial list of some of the things that I typically consider when trying to make a modeling approach decision:

1. Physical description of the river channels, floodplain areas, bridges/culverts, other hydraulic structures, levees, roads, etc. that the model will be applied to.
2. What is the typical size, length, and complexity of the systems that these models will be applied to? Is it a 1 mile, 10, 50, 100, 500, or 1000 mile river system?
3. Will this model be used for Planning type studies, or will it be used for real time modeling and mapping?
4. What type of events (hydrology and boundary conditions) will the models be used to analyze?
5. What is the typical duration of a flood event on this river system? (1/2 day, 1-day, 3-days, 1-week, 1-month, or 6-months)
6. Are there unique aspects of the system that will significantly affect the computed results? Such as: is the river tidally influenced; do wind speed and directions affect the water surface elevations; is the river affected by floating ice or ice jams; does there tend to be debris issues during flood, and does the debris tend to pile up at hydraulic structures; will levee overtopping, breaching, and interior flow routing need to be addressed, Are there any significant bridges and culverts that will cause water to backup behind them during significant flood events, etc...
7. What is the level of accuracy of the terrain data and hydraulic structure data?
8. What is the general level of accuracy of the hydrology used to drive the models?
9. What are the required outputs from the model (water surface elevations, water depths, arrival times, average velocities, detailed velocities in two dimensions at specific point locations, etc...)?
10. What is the model purpose and expected level of accuracy required?

With that said, I will try to offer both a theoretical opinion and a practical application opinion to the questions posed above.
Steady Versus Unsteady Flow Models

Steady flow models (1D or 2D) should generally not be used when the following situations exist in the river system being analyzed (this is not an exhaustive list):

- The river is tidally influenced, and the tide has a significant effect on the stage for the area of interest.
- The events being modeled are very dynamic with respect to time (e.g., Dambreak flood waves; flash floods; river systems in which the peak flow comes up very quickly, stays high for a very short time, and then recedes quickly).
- Flow reversals occur during the event.
- Dynamic events such as levee overtopping and breaching occur during the event.
- Extremely flat river systems, where gravity is not necessarily the only significant driving force of the flow.

In addition to the specific items listed above, a successful application of any steady-flow model requires that flow rates have already been accurately computed by a hydrologic model or measured by an accurate and complete set of stream gages. If a hydrologic model is used to not only compute the rainfall-runoff over the watershed, but perform all of the routing within the system, then the flow rates used in the steady-flow model are only as accurate as the hydrologic model could compute them. So, the use of a steady-flow hydraulic model is predicated on the fact that a hydrologic model was considered to be appropriate for not only developing the flow rate from rainfall runoff computations, but also routing all of the flows through the system during the event. Therefore, a large part of the decision of steady-flow versus unsteady flow hydraulic modeling comes down to the question: is hydrologic stream flow routing accurate enough to produce flow rates that can be used in the corresponding steady-flow hydraulics models.

Even considering all of what is stated above, there are still many areas in which a good hydrologic model (one that is representative of the watershed and has been well calibrated) can be used in conjunction with a steady-flow hydraulics model to perform real-time river forecasting and mapping, with reasonable accuracy.
1D Versus 2D Hydraulic Modeling

The question of 1D versus 2D hydraulic modeling is a much tougher question than steady versus unsteady flow. There are definitely some areas where 2D modeling can produce better results than 1D modeling, and there are also situations in which 1D modeling can produce as good as or better results than 2D models... with less effort and computational requirements. Unfortunately, there is a very large range of situations that fall into a gray area, and one could list the positive and negative aspects of both methodologies for specific applications. Here are some areas where I think 2D modeling can give better results than 1D modeling:

- When modeling an area behind a levee system, and the levee will be overtopped and/or breached, the water can go in many directions. If that interior area has a slope to it, water will travel overland in potentially many directions before it finds its way to the lowest point of the protected area, and then it will begin to pond and potentially overtop and/or breach the levee on the lower end of the system. However, if a protected area is small, and ultimately the whole area will fill to a level pool, then 1D model is fine for predicting the final water surface and extent of the inundation.
- Bays and estuaries in which the flow will continuously go in multiple directions due to tidal fluctuations and river flows coming into the bay/estuary at multiple locations and times.
- Highly braided streams
- Alluvial fans — however, this is very debatable that any numerical model can capture a flood event accurately on an alluvial fan, due to the episodic nature of flow evolutions that can change the whole direction of the channels during the event.
- Flow around abrupt bends in which a significant amount of super elevation will occur during the event.
- Very wide and flat flood plains, such that when the flows goes out into the overbank area, the water will take multiple flow paths and have varying water surface elevations and velocities in multiple directions.
- Applications where it is very important to obtain detailed velocities for the hydraulics of flow around an object, such as a bridge abutment or bridge piers, etc...

The following are areas in which I think 1D modeling will produce as good as or better results than 2D modeling for real time flood forecasting applications, with less effort (both from a model development, calibration, and application viewpoint, as well as a computational time viewpoint):

- Rivers and floodplains in which the dominant flow directions and forces follow the general river flow path. This covers a lot of river systems in my opinion, but it is obviously debatable as to the significance that lateral and vertical velocities and forces impact the computed water surface elevations and the resulting flood inundation boundary.
- Steep streams that are highly gravity driven and have small overbank areas.
River systems that contain a lot of bridges/culvert crossings, weirs, dams and other gated structures, levees, pump stations, etc... and these structures impact the computed stages and flows within the river system. I have not seen any 2D model yet that has a comprehensive set of hydraulic structure modules/capabilities that can handle the full range of hydraulic flow situations that can come up on many of our river systems. This is an area that the current state of the art in 1D models is far ahead of the 2D models. This statement does not mean that these capabilities cannot be incorporated into a 2D model. It just means that I have not seen a widely used 2D model that has such a comprehensive set of capabilities.

- Medium to large river systems, where we are modeling a large portion of the system (50 or more miles), and it is necessary to run longer time period forecasts (i.e. 2 week to 6 month forecasts). Even with the tremendous advancements in multi-processor computing, and GPU (Graphics Processor Units) computing, there are still significant spatial and simulation time limitations on what we can effectively use 2D models for in the real time forecasting domain. This will obviously be changing over time.

- Areas in which the basic data does not support the potential gain of using a 2D model. If you do not have detailed overbank and channel bathymetry, or you only have detailed cross sections at representative distances apart, many of the benefits of the 2D model will not be realized due to the poor accuracy of the terrain data.

With all of that said, there are many areas in which it will be highly debatable as to the relevant accuracy of using a 1D or a 2D model for a specific application. There are many aspects to consider, other than purely “am I solving the full Saint Venant equations in one dimension or two dimensions”. I believe that there are both knowledge gaps in understanding when 1D versus 2D should be used, and there are tool gaps. I personally believe that combined 1D/2D models will play an important role in our modeling efforts in the near term. This is an area where the hydraulic modeling tools need to be improved.

I am also of the viewpoint that the majority of uncertainty and ability to accurately forecast stages and flows in river systems is mostly due to poor estimation of rainfall both spatially and temporally, and hydrologic modeling, which often includes large portions of ungauged areas in which little to no calibration could be performed. This is more often than not a much greater contributor to forecast/modeling error than any differences arising from 1D versus 2D model choices.
2. **Engineers Australia, 2012**

We quote herein the descriptions found in one of the references. These ARE NOT our words.

**“6.2.1. Fixed Grids**

The fixed grid approach is usually based on a finite difference solution to the governing equations. This approach requires that a grid of same size and shaped elements is developed as illustrated in Figure 6-1. A variant on the rectilinear grid is the “boundary-fitted coordinate” or “finite difference curvilinear” method. Generally referred to as a “curvilinear” model, this method allows the grid to be slightly distorted so as to align the elements with the streamlines or flow paths of importance (Engineers Australia, 2012).

The solution schemes for fixed grid models are numerically more efficient to solve than their flexible mesh counterparts, especially those schemes that are implicit and utilise matrix solvers. This is due to the “well-organised” nature of the coefficient matrix that must be solved mathematically at each time step (refer Chapter 3). Generally, the fixed grid models are between 4 and 8 times faster to solve than flexible mesh models where the same number of elements exist in each type of model. In some cases, flexible mesh models have the ability to reduce the number of elements (and thereby reduce run times) by utilising larger elements in areas away from the area of interest or where larger elements adequately depict the hydraulic behaviour. The trade off between resolution and model efficiency must always be considered when choosing a model type.

**Figure 6-1 Regular Fixed Grid and Curvilinear Grid**

![Figure 6-1 Regular Fixed Grid and Curvilinear Grid](image-url)
6.2.1.1. Rectilinear Grids

The rectilinear fixed grid technique is particularly popular for flooding and urban drainage studies as they have historically been more stable and faster than finite element and finite volume models that use a flexible mesh. The rectilinear models are relatively simple to develop compared to the flexible mesh model as there is no requirement to manually develop a model mesh. This can save significant time and effort in the model development process. However, in some situations the rectilinear models are not suitable for resolving channel flow due to the poor representation of flowpaths by the grid. For example, Figure 6-1 shows how the channel becomes staggered where the alignment is angled to the fixed grid. This problem can be resolved by increasing the resolution with a finer grid size but it comes at the significant cost of increased computational time. Generally, the reduction of the grid cell size by a factor of 2 will lead to an 8 fold increase in the model run time. If the resolution of channel hydraulics is important objective for the model then the use of a 1D representation of the hydraulic feature can be included with linkages made to the 2D grid (see Chapter 9).

The number of grid cells chosen to represent a channel is dependent upon the objectives of the modelling exercise and the importance of the channel’s conveyance in the conceptual model. A minor or insignificant channel may not require accurate representation and thus be modelled as a simplified channel with fewer grid cells. The model’s ability to represent the conveyance of a channel will also be dependent upon the orientation of the grid to the direction of flow in the channel. A meandering channel through a fixed grid may require more grid cells than a relatively straight channel directly aligned with the grid orientation. Generally a minimum of five (5) grid cells across the width of a channel is required to produce an adequate representation of channel flow.

6.2.1.2. Curvilinear Grids

A curvilinear grid model solves the governing flow equations on a transformed reference system based on polar coordinates. This allows the model “grids” to curve slightly to fit within predefined flow paths as shown in Figure 6-1. The use of curvilinear grids is preferred where the flow is predominately uni-directional such as a river or drainage channel with generally unidirectional flow in floodplains. In these situations, the total number of computational points can be significantly reduced compared to a rectilinear grid whilst still maintaining the benefits of having a finite difference solution scheme. In addition, the grid is aligned to the flow path, which enhances the representation of the channel conveyance. A typical example of a curvilinear model on a river system is shown in Figure 6-2. The curvilinear mesh has some restrictions that must be considered when selecting the model type. The first consideration is the grid dimension. The overall grid dimensions must be maintained throughout the model domain, as is the case in a rectilinear model. Therefore, the number of cells in the longitudinal direction and the transverse direction must be maintained throughout the model domain. If a large number of cells are required to represent a large floodplain in the transverse direction in one part of the model; the same number of cells must be maintained in the transverse direction through more narrow parts of the model. This can produce a very small grid
resolution size in the narrow part of the model and result in significantly large numbers of grid cells. This in turn requires a reduced model timestep, resulting in longer run times. The second consideration is the development of the mesh. The curvilinear models require that the angle between crossing grid lines is maintained at 90 degrees for all elements and the aspect ratio (the grid cell length to width ratio) for the distorted element should not exceed five, with three being a recommended value. This geometry restriction requires greater effort from the modeller in developing the mesh.

**Figure 6-2 Example of the Use of a Curvilinear Grid**

![Curvilinear Grid Example](image)

### 6.2.2. Flexible Mesh

Models that utilise a flexible mesh are becoming increasingly popular, particularly those using a finite volume solution. The flexible mesh method involves the discretisation of the model domain into triangular or quadrilateral elements with a single element type being applied exclusively or in combination with other types. The elements applied in a mesh can be of varying size (area), as shown in Figure 6-3. The flexible mesh technique provides a great amount of flexibility in the representation of a complex geometry and enables the mesh to be customised to resolve specific details such as boundary conditions, physical features and channels. Higher resolution can be provided with smaller elements in areas of interest or where there are rapid changes in flow behaviour. In other areas such as wide open floodplains, larger elements can be developed to reduce the number of computational elements and thereby reduce the model run time.

However, the numerical solution scheme used for flexible mesh models is not as efficient as that used for finite difference models. As previously stated in Section 6.2.1, the flexible mesh models are between four and eight times slower than an equivalent finite difference model with an equivalent number of computational points. Nevertheless, flexible mesh models can be an attractive alternative when the number of computational elements can be reduced to the extent that they offset the reduced numerical efficiency.

**Figure 6-3 Examples of Flexible Mesh Elements**

![Flexible Mesh Elements](image)
### 3. Open Channel Flow Resistance, Ben Yen, 2002

**Table 3. Equations for Composite or Composite Channel Resistance Coefficients**

<table>
<thead>
<tr>
<th>Eqn.</th>
<th>( n_j )</th>
<th>Concept</th>
<th>Equation</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>( \frac{\sum P_i A_i}{A} )</td>
<td>Sum of component is weighted by area ratio; total shear stress is weighted sum of subarea shear stresses</td>
<td>( Q = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>U.S. Army Corps of Engineers Los Angeles District Method, see Con (1973)</td>
</tr>
<tr>
<td>B</td>
<td>( \frac{\sum \eta_j A_j}{A} )</td>
<td>Total resistance force is equal to sum of subarea resistance forces, or ( n_j ) weighted by ( A_j )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>C</td>
<td>( \frac{A}{\sqrt{R S}} )</td>
<td>Total discharge is sum of subareas discharges</td>
<td>( Q = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>D</td>
<td>( \frac{\sum (n_j A_j)^{1.2}}{A} )</td>
<td>Same as Horton and Einstein's Eq. 4 but derived empirically</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>E</td>
<td>( \frac{1}{n} \sum (n_j A_j) )</td>
<td>Total cross-sectional mean velocity equal to subareas mean velocity</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>F</td>
<td>( \frac{F}{P} )</td>
<td>Total discharge is sum of subareas discharges</td>
<td>( Q = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>G</td>
<td>( \frac{1}{n} \sum \eta_j )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>H</td>
<td>( \frac{\sum (n_j P_j)}{F} )</td>
<td>Total shear velocity is weighted sum of subarea shear velocities; ( \sum \eta_j ) is linearly proportional to wetted perimeter</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>I</td>
<td>( \frac{1}{n} \sum \eta_j )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>J</td>
<td>( \frac{1}{n} \sum \eta_j )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>K</td>
<td>( \frac{F}{P} )</td>
<td>Total discharge is sum of subareas discharges</td>
<td>( Q = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>L</td>
<td>( \frac{1}{n} \sum \eta_j )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>M</td>
<td>( \frac{\sum \eta_j}{n} )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>N</td>
<td>( \frac{\sum \eta_j}{n} )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>O</td>
<td>( \frac{\sum \eta_j}{n} )</td>
<td>Total shear stress, ( \eta_j ) is sum subareas resistance forces, ( 2 F_1 )</td>
<td>( F_1 = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
<tr>
<td>Z</td>
<td>( \sum \eta_j )</td>
<td>Logarithmic velocity distribution over depth ( a ) for wide channel</td>
<td>( Q = \frac{P}{\sqrt{R S} - \frac{P}{\bar{P}}} )</td>
<td>Colebatch (1941)</td>
</tr>
</tbody>
</table>

28 / JOURNAL OF HYDRAULIC ENGINEERING / JANUARY 2002

Scaling and Similarity in Rough Channel Flows, G. Gioia and F.A. Bombardelli

4. Scaling and Similarity in Rough Channel Flows, G. Gioia and F.A. Bombardelli
used by Carr [3] to adapt the tabulated values of \( n \) to the gravitational field of Mars. The scaling \( n \sim r^{1/6} \) was proposed by Strickler [2] based on the analysis of extensive experimental data.

In deriving Manning’s formula we expect to verify (i) that an incomplete similarity in \( r/R \) prevails for \( r/R \ll 1 \); (ii) that the similarity exponent is \( a = -1/6 \); and (iii) that for rectangular channels the hydraulic radius surfaces to characterize the geometry of the cross section.

We start by considering a rectangular channel of slope \( S \). Then, the streamwise component of the gravitational force per unit length of channel is \( F_r = \rho g b h S \), where \( \rho \) is the density of the fluid. Let us call \( S \) a wetted surface tangent to the peaks of the roughness elements, Fig. 1. (For the time being we need only consider roughness elements of uniform size \( r \).) Under conditions of fully developed turbulence, the streamwise component of the force on \( S \) per unit length of channel is \( F_r = (b + 2h) r \). In this expression, \( b + 2h \) is the wetted perimeter and \( r = \rho [u_\tau^2] \) is a Reynolds shear stress, where \( u_\tau \) and \( v_\tau \) are the fluctuating velocities normal and tangent to \( S \), respectively, and an overbar denotes time average. We study \( u_\tau \) first, and start by making a crucial observation: when the relative roughness is small \( (r/R \ll 1) \), turbulent eddies of sizes larger than, say, \( 2r \), can provide only a negligible normal velocity to \( S \), Fig. 1. On the other hand, turbulent eddies smaller than \( r \) are in the space between successive roughness elements, and they can provide a normal velocity to \( S \). However, when these eddies are smaller than, say, \( r/2 \), their characteristic velocities are negligible compared with the characteristic velocity of the eddies of size \( r \). Thus, \( u_\tau \) is dominated by \( u_\tau \), which is the characteristic velocity associated with the eddies of size \( r \) (a suitable mathematical expression for \( u_\tau \) is given in [8]). In other words, \( u_\tau \sim u_\tau \), where the symbol “\( \sim \)” means “scales with.” We now turn to \( v_\tau \). Eddies of all sizes can provide a velocity tangent to \( S \). It follows that \( v_\tau \) is dominated by \( V \), which is the characteristic velocity associated with the largest eddies, and \( v_\tau \sim V \). We sumise that \( [u_\tau] \sim u_\tau V \), which together with the equation of balance of momentum transfer, \( F_r = F_r \), lead to

\[
u_\tau V \sim \left( \frac{bh}{b + 2h} \right) g S = R g s.
\]

We now seek to relate \( u_\tau \) and \( V \). To that end we use Kolmogorov’s scaling. This scaling can be easily derived for isotropic turbulence. It has been proved, however, that the scaling applies as well to turbulence which is not only anisotropic, but also inhomogeneous [9] the turbulence is inhomogeneous in the vicinity of the wall. If the eddies of size \( r \) are within the inertial range (i.e., \( r \gg \eta \), where \( \eta \) is the Kolmogorov length), then \( u_\tau^2 / r \sim e \), where \( e \) is the rate of dissipation of turbulent energy per unit mass. According to Kolmogorov’s theory of turbulence, \( e \) equals the rate of production of turbulent energy per unit mass, and is independent of the viscosity [8,10]. It follows that a scaling expression for \( e \) can be obtained in terms of \( V \), \( b \), and \( h \). The largest eddies possess an energy per unit mass \( \sim V^2 / R \); of these, the ones with horizontal vorticity vector are characterized by a turnover time \( (b/2) / V \). We conclude that

\[
\frac{u_\tau^2}{r} \sim e \sim \frac{V^2}{h} \frac{b}{2V} = \left( \frac{b + 2h}{bh} \right) V^3 = \frac{V^3}{R},
\]

whereupon

\[
u_\tau \sim \left( \frac{L}{R} \right)^{1/3} V.
\]

This equation indicates that \( u_\tau \) is self-similar in \( r \) with exponent \( 1/3 \), a well-known result of Kolmogorov’s theory [8]. More surprisingly, \( r \) appears normalized by the hydraulic radius \( R \). Substituting (6) into (4) yields

\[
V \sim \left( \frac{L}{R} \right)^{-1/6} \sqrt{R g s},
\]

which is the leading term of (3) with \( a = -1/6 \), as expected. This concludes our derivaton.

We have derived Manning’s formula for the case of channel walls with roughness elements of uniform size \( r \). We now generalize our derivation to the case of channel walls with roughness elements in a range of sizes. Consider

\[
\begin{array}{c}
\text{FIG. 1. Immediate vicinity of a channel wall with roughness elements of characteristic size } r. \text{ The dashed line is the trace of a wetted surface } S \text{ tangent to the peaks of the roughness elements.}
\end{array}
\]

\[
\begin{array}{c}
\text{FIG. 2. Largest-length-scale eddies in a rectangular channel of width } b \text{ and depth } h. \text{ The velocity of these eddies scales with the mean velocity of the flow } V.
\end{array}
\]
a channel wall $\mathcal{W}$ characterized by a probability distribution $p(\sigma)$, $\sigma_0 p(\sigma) d\sigma = 1$, where $p(\sigma) d\sigma$ measures the probability of finding a roughness element of a size between $\sigma$ and $\sigma + d\sigma$. Assume that for the channel wall $\mathcal{W}$ the average roughness element is of size 1, i.e., that $\sigma_0 \sigma p(\sigma) d\sigma = 1$. Then, we can use $\mathcal{W}$ to generate a family of geometrically similar channel-wall surfaces $\{\mathcal{W}_r\}$. For a generic member $\mathcal{W}_r$ of this family of channel-wall surfaces the average roughness element is of size $r$, i.e., $\sigma_0 \sigma p(\sigma/r) d(\sigma/r) = r$. (The concept of geometrically similar channel-wall surfaces dates back to the early Twentieth Century; see, e.g., [11]). We now redefine Manning’s formulas for a generic member $\mathcal{W}_r$ of the family of geometrically similar channel-wall surfaces $\{\mathcal{W}_r\}$. The average Reynolds stress on the wetted surface $S$ of Fig. 1 is $\tau = \rho V \int_0^\infty u_\rho p(\sigma/r) d(\sigma/r)$, and we can rewrite (4) in the form

$$V \int_0^\infty u_\rho p(\sigma/r) d(\sigma/r) = \frac{\text{Rgs}}{\alpha}. \quad (8)$$

On the other hand,

$$u_\rho = \frac{\sigma^2}{r} V = \left( \frac{\sigma}{r} \right)^{3/2} \frac{V}{R}. \quad (9)$$

Substituting (9) into (8) leads to the leading term of (3) with

$$K = K_0 \left( \int_0^\infty \xi^{1/2} p(\xi) d\xi \right)^{-1/2}. \quad (10)$$

where $K_0$ is a constant.

We obtained (7) based on three assumptions. The first one is that $\eta < 1$. In keeping with this assumption, (7) corresponds to the leading term in the power-law asymptotics of Eq. (3). The second assumption is that the turbulent eddies in the vicinity of the walls are governed by Kolmogorov’s scaling (6). This is justified because Kolmogorov’s scaling has been shown to apply to inhomogeneous turbulence. The third assumption is that the spaces between roughness elements are occupied by eddies of size $r$, in the form shown in Fig. 1. We now discuss this third assumption.

It is apparent that an eddy of size $r$ could be found between any two successive roughness elements. In deriving (7) we have assumed, however, that one such eddy does occupy the space between each pair of consecutive roughness elements. Our assumption could be justified by recalling that in Kolmogorov’s theory eddies of any given size within the inertial range are space filling (this is required for $\eta$ to be scale invariant within the inertial range [11]). It is perhaps more illuminating to think of the assumed set of eddies of size $r$ as akin to the arrays of parallel vortices that have long been documented in the vicinity of smooth channel walls, and which constitute the most common form of coherent structures. (Note, however, that the eddy of size $r$ in Fig. 1 need not have a vorticity vector oriented streamwise.) We know from theoretical work on the etiology of coherent structures that numerous instabilities are possible leading to arrays of vortices of specific wavetlengths [12,13]. Interestingly, it has been conjectured that the presence of periodic forms of wall roughness (such as, for instance, riblets) may excite instabilities of similar wavelength [12]. This conjecture affords a compelling explanation for the incomplete similarity in the relative roughness, $r/R$, displayed by Eq. (3). In fact, this similarity is quite puzzling: given that turbulence involves a wide spectrum of wavelengths, spanning many orders of magnitude, why would $r$, which is just one wavelength somewhere within that spectrum, appear so conspicuously in (3)? The puzzle is explained if the wall roughness induces eddies of size $r$ in the immediate vicinity of the wall and if, as suggested by our derivation, these eddies effect most of the momentum transfer. Thus, if $r$ diminishes, the capacity for momentum transfer also diminishes; as a result, the fluid friction diminishes, and the mean velocity increases, as indicated by Eq. (7). Given that the size of the eddies is bounded below by the Kolmogorov length $\eta$, it is interesting to investigate what happens when $r$ approaches $\eta$. To that end, we start by recalling that $\eta = \nu^{-3} e^{-1/4}$, where $\nu$ is the kinematic viscosity. From (5) we have $r \sim \eta^{3/2}$, and therefore $\eta/\eta \sim (\nu/\eta^{3})^{3/2} = \text{Re}^{-3/4}$, where $\text{Re} = \nu R / \nu$ is the Reynolds number. Therefore, as the roughness approaches the Kolmogorov length (i.e., as the channel walls become hydraulically smooth), we expect (7) to become

$$V \sim \text{Re}^{-3/8} \sqrt{\text{Rgs}}. \quad (11)$$

The appearance of the Reynolds number in (11) indicates that in the limit $r = \eta$ the momentum transfer is viscous. It is convenient to write (11) in terms of the resistance coefficient, $f = \text{Rgs}/V^2$; the result is $f \sim 4^{-1/4}$, which we recognize as Blasius’s classical empirical scaling for hydraulically smooth channels [2]. This result unveils the existence of a relationship among the three well known, and apparently unrelated, scalings due to Blasius, $f \sim \text{Re}^{-1/4}$, Kolmogorov, $\eta = \nu^{-3} e^{-1/4}$, and Manning, $V \sim \eta^{-1/2}$.

We have provided a derivation of Manning’s empirical formula. Besides the final result, we have reached a number of interesting conclusions. For example, in rectangular channels the Reynolds stress in the immediate vicinity of the walls depends on (i) the mean velocity of the flow, (ii) the local wall roughness, and (iii) the depth and width of the cross section through the hydraulic radius only. This conclusion suggests ways of formulating generalized Manning formulas for channels in which different portions of the channel walls are characterized by different families of geometrically similar channel-wall surfaces, of considerable interest in applications. It also has momentous geomorphological implications, since it allows for the determination of absolutely stable aspect ratios, $b/h$, in natural channels. We shall study these and other related issues in a separate paper.
G. G. is very grateful to Professor W. R. C. Phillips for encouragement, for many illuminating discussions, and for his comments on several versions of the manuscript.


[10] D. Lohse, Phys. Rev. Lett. 73, 3223 (1994). The existence of an upper bound on $c$ that is independent of the viscosity has been proved mathematically; see Ch. R. Doering and P. Constantin, Phys. Rev. Lett. 69, 1648 (1992).


5. Snapshots from VAVUQ GUI for Verification, Validation and Uncertainty Quantification

Choose between Verification and Validation subroutines

Verification with and without known benchmark solution

Interpolation method for conversion of results to same grid points
Selection of output level of details

Confidence level for uncertainty quantification