Tutorial on the use of TELEMAC-2D Hydrodynamics model and Pre-/Post-processing with BlueKenue for flood-inundation mapping in Unsteady Flow Conditions

Christopher Gifford-Miears, Arturo S. Leon[†]



Oregon State University School of Civil and Construction Engineering CE 540 **Unsteady flows in Rivers and Pipe Networks**, Fall 2013

Instructor: Arturo Leon, Ph.D., P.E. E-mail: arturo.leon@oregonstate.edu Office phone: (541) 737-2606 Mail address: 220 Owen Hall School of Civil and Construction Engineering Oregon State University Corvallis, OR 97331-3212 Research Web page: http://web.engr.oregonstate.edu/~leon

Geometric data accessed and adapted from U.S. Army Corps. Hydrologic Engineering Center - HEC-GeoRAS Tutorial

Objectives of this Tutorial: The objective of this tutorial is to give a brief introduction on the use of TELEMAC-2D and BlueKenue for setting up and analyzing flood inundation in unsteady flow conditions.

KeyWords: Unsteady hydraulic routing; Two-dimensional modeling; Flood-inundation; BlueKenue; Finite-element mesh generation

^{*}Graduate Research Assistant, School of Civil and Construction Engineering, Oregon State University, 233 Owen Hall, Corvallis, OR 97331, USA. E-mail: cgiffordmiears@gmail.com (Corresponding author)

[†]Assistant Professor, School of Civil and Construction Engineering, Oregon State University, 220 Owen Hall, Corvallis, OR 97331-3212, USA. E-mail: arturo.leon@oregonstate.edu

Contents

1	Pre-	requisites	9
	1.1	Computer Requirements	9
	1.2	Data Requirements	9
2	Extr	act bathymetry from ArcGIS surface	11
	2.1	ArcGIS TIN to ASCII text file	11
	2.2	Converting ArcGIS ASCII text file to XYZ text file	15
	2.3	Export ArcMap data as a Shape File	18
	2.4	Representing TIN elevation data using Shape file	21
	2.5	Convert source data from US Survey Feet to Meters	22
3	Pre-	processing utilizing BlueKenue	23
	3.1	Importing data to BlueKenue	23
	3.2	Viewing data in BlueKenue	25
	3.3	Mesh Generation	27
	0.0	3.3.1 Steps 1 and 2: Create Open- and Closed-Lines in BlueKenue	$\frac{-1}{28}$
		3.3.2 Steps 3 and 4: Create Channel Meshes in BlueKenue	30
		3 3 3 Steps 5 and 1. Create Combined Mesh in BlueKenue	35
	34	Interpolate bathymetry to the mesh	38
	5.4	3.4.1 Levee height adjustment	<u>4</u> 3
		3.4.2 Viewing cross-sections of the mesh	$\Lambda\Lambda$
			44
4	TEL	EMAC-2D input file generation	47
	4.1	BlueKenue	47
		4.1.1 TELEMAC-2D Geometry File	47
		4.1.2 TELEMAC-2D Boundary Condition File	52
	4.2	FUDAA Pre-pro	61
		4.2.1 TELEMAC-2D parameters file (.cas)	61
		4.2.2 Example unsteady parameters file (.cas)	68
5	Run	ning TELEMAC-2D simulation	71
	5.1	TELEMAC-2D from the DOS Command Prompt	71
	5.2	TELEMAC-2D Steady-state simulation	75
		5.2.1 Create a HOTSTART file from previous computation	75
	5.3	TELEMAC-2D Unsteady simulation	81
		5.3.1 Unsteady boundary conditions	81
6	Post	-processing utilizing BlueKenue	83
	6.1	Flood inundation view settings	85
	6.2	Animation of Flood-wave propagation	85
	6.3	Outflow hydrographs using MATLAB	86
	6.4	TELEMAC-2D vs HEC-GeoRAS Flood Inundation	87

Append	ices	89
Append	ix A Create Simple Meshes in Blue Kenue	89
A.1	Supporting figures for simple mesh tutorial	89
Append	ix B Self-installation of TELEMAC hydrodynamics suite	95
B.1	Pre-requisite programs for TELEMAC installation	95
B.2	TELEMAC source code checkout using Tortoise SVN	95
B.3	Environment variables for running/compiling TELEMAC	96
B.4	Testing your PATH variables	96
B.5	Compiling TELEMAC	96

List of Figures

1	Contents of unzipped TELEMAC tutorial data. Note: extract this data to a direc-	
	tory with no spaces	10
2	Activate 3D Analyst extension from ArcMap Customize > Extensions menu	11
3	Add TIN data to ArcMap and initialize the ArcToolbox	12
4	Extract and export ASCII xyz bathymetric data utilizing the 3D Analyst Tools	
	toolbox	13
5	Prompt from Step 1 , converting <i>TIN</i> to <i>TIN-Node Feature Class</i>	14
6	Prompt from Step 2, converting TIN Node Feature Class to ASCII text	15
7	Open the Baxter River ASCII text file using Excel, importing as a space delimited	
	file	16
8	Format the cells in order to avoid truncating the data	17
9	Node IDs are not necessary for <i>xyz</i> format; delete the entire column of IDs	17
10	Save altered bathymetric ASCII text file as a Tab delimited text file	18
11	BlueKenue utilizes a (.xyz) file extension, therefore the extension is simply up-	
	dated to (.xyz)	18
12	Open ArcMap and add the RASGeometry data layer from baxter10.mbd database	19
13	Select dataset of interest from RASGeometry and Export Data	20
14	Exported data features, as Shape Files (*.shp), are simple to import using BlueKenue	21
15	BlueKenue is capable of importing ArcMap/ArcView Shape Files directly	22
16	Import the Baxter bathymetry using either the <i>Open</i> command or the <i>Import</i> ;	
	ArcView Shape file command	23
17	When importing the bathymetry text file, toggle file types to <i>All Files</i> (*.*)	24
18	To view Data Items imported to BlueKenue, drag item of interest to a Views object	
	(e.g. 2D View (1))	25
19	Create a new viewing window using the toolbar icons or Windows command and	
	drag the data object to the new view (e.g. 3D View (2))	26
20	Step 1; create a New Closed Line representing the domain extents	28
21	Step 2; Right and Left banks are necessary for all channels; either import the 2D-	
	Lines (.i2s), or create them using New Open Lines	29
22	For each channel mesh (i.e. 3 in total) a new Channel Mesher object should be	
	initialized	30
23	Specify the Open Lines representing banks of the channel feature of interest (e.g.	
	Baxter River) and specify the meshing parameters (Table 3)	31
24	Rename and Run the Channel Mesher	32
25	Resulting Baxter River Channel Mesh and corresponding parameters	33
26	Resulting Baxter River Channel Meshes	34
27	Baxter River Channel Meshes components	34
28	Settings for Baxter tutorial T3 Mesh Generator	35
29	Mesh components added to Mesh Generator	36
30	Execution of the mesh generator	37
31	Resulting 2D triangular mesh generated using T3 Mesh Generator	38
32	Create a new 2D Interpolator object	39

33	Drag elevation datasets to 2D Interpolator object	40
34	Select the mesh and execute the Map Objects command (e.g. Tools ; Map Ob-	
	jects)	40
35	Select the elevation source data to map, or interpolate, onto the selected object (mesh)	41
36	When Map Object command is finished, a progress window will appear	41
37	Drag the interpolated mesh onto a 2D View Object to view the mapped bathymetry	42
38	Drag the interpolated mesh onto a 3D View Object to view the mapped bathymetry	42
39	Adjust the levee height prior to interpolating the domain mesh by using the BlueKenue	
	calculator	43
40	Apply adjusted levee heights using a 2D Interpolator	43
41	Create a new line where cross-section view is desired	44
42	Resample the newline to increase number of sampling points	45
43	Once resampled, use command Map Object to choose the source surface to sam-	
	ple from	46
44	Drag the mapped, resampled line onto a new 1D View to see the cross-section	46
45	Create a new geometry object using either the toolbar icon or File command	48
46	Double-click the New Selafin object, rename, and select Apply	49
47	Right-click the geometry object and select Add Variable	49
48	Select the computational domain mesh and add as a BOTTOM variable	50
49	Geometry object with BOTTOM variable added	50
50	Non-interpolated / Mapped BOTTOM - BAD	51
51	Correctly interpolated / Mapped BOTTOM - GOOD	51
52	Comparison of bad versus good BOTTOM interpolation	51
53	Save geometry object under the TELEMAC simulation directory	52
54	Create a new Boundary Conditions file for the BOTTOM object	53
55	Assign the new Boundary Conditions file to the BOTTOM object	54
56	Rename the BOTTOM BC object for ease of identification	55
57	All boundary nodes are assigned as Closed boundary (wall) by default	55
58	Zoomed in view of the default bc baxter object	56
59	TELEMAC-2D Boundary condition node types	56
60	Boundary condition overview for the Baxter River tutorial	57
61	Select the starting edge node where the boundary condition segment begins	58
62	Select the end node of the segment and right-click to Add Boundary Segment	58
63	Apply an Open boundary with prescribed Q BC code to the Baxter River up-	
	stream reach	59
64	Boundary condition applied to bc baxter object	59
65	Save both boundary condition objects to the simulation directory	60
66	FUDAA Supervisor initialized screen	61
67	Populate this window with necessary hydraulic project files	62
68	Under the Steering file field, specify the name of your parameters file (e.g. bax-	
	ter_unsteady.cas	63
69	Under the Boundary conditions field, specify the file location of bounary condi-	
	tions file, bc_baxter.cli	64
70	Under the Serafin file field (i.e. TELEMAC geometry file format), specify the	
	location of geometry_baxter.slf	65

71	Select the General Parameters tab to access the project parameters	66
72	Under General Parameters the files associated with the project are shown	66
73	Select the Results File field to specify where to write the T2D results	67
74	The Keywords tab is used for viewing and understanding the hydraulic project	
	parameters	68
75	Project files for example TELEMAC-2D unsteady parameters file	69
76	Using the filters under the Keywords tab can help view pertinent parameters to	
	your project quickly	69
77	Once changes are saved from FUDAA, the parameters file is well organized and	
	ready for the TELEMAC-2D simulation	70
78	initialize new DOS command prompt	71
79	Test Python installation	72
80	Check Python version, should be 2.7.3 or 2.7.5	72
81	Test gfortran installation	72
82	Copy the full-path directory address to the TELEMAC_simulation_files folder	73
83	Using the change-directory DOS command, cd , change the directory to the loca-	
	tion of the TELEMAC_simulation_files folder	73
84	Within the input file working directory, execute TELEMAC-2D using the com-	
	mand, telemac2d.py name_of_input_file.cas	74
85	If there are no errors, the simulation will execute until finished	74
86	Open the previous results file in BlueKenue (e.g. Baxter_SS_hotstart.slf)	75
87	Create, and rename a new Selafin file for the HOTSTART components to be stored	76
88	If the previous results file has several timesteps, be sure to Animate the results,	
	and fast-forward to the final frame	76
89	BlueKenue Calculator method for extracting V component velocities as a new ob-	
0.0	ject hotstart_velocity_V	77
90	BlueKenue Calculator method for extracting U component velocities as a new ob-	
	ject hotstart_velocity_U	77
91	Add copies of the current domain mesh as new variables to the Selatin file	78
92	Unmapped new variable VELOCITY U, prior to Map Objects command	/8
93	Map previous result file components to new components using Map Objects com-	70
0.4		79 70
94 05	Successfully mapped VELOCITY U object	/9
95	Successful HOISTART file generation should contain the above four child-objects	80
90 07	Outflow ratio a sume for this tutuorial	81 82
9/	Outnow rating curve for this tutuorial	82
98	Applying a vertical exaggeration to datasets with nonzontal scales much larger than the vertical scale (e.g. POTTOM, EPEE SUBEACE, etc.) halps to visualize	
	than the vertical scale (e.g. BOI IOW, FREE SURFACE, etc.) helps to visualize	02
00	Changing the Style and engoity of the object is easily performed by double eligibing	03
99	changing the style and opacity of the object is easily performed by double-clicking	01
100	After initializing the Animate item under properties and select Anniv , then the	04
100	After initializing the Annuale from under properties and select Apply, then the	Q1
101	To view only the Flood Inundation depths, apply Clin Contours and make sure	04
101	that Style is set to Filled Contours	85
		05

102	Resulting view from Figure 101 settings with Animate enabled	85
103	Control sections within the Baxter River domain	86
104	Outflow hydrographs resulting from unsteady TELEMAC-2D simulation	87
105	Example comparison of TELEMAC-2D and HEC-GeoRAS inundation extents	88
A106	Create closed line for the exterior boundary of the mesh.	90
A107	Create New T3 Mesh	90
A108	BDrag newClosedLine into newT3Mesh Outline	91
A109	Select edge length for elements and run.	91
A110	The new T3 mesh!	92
A111	Create New 2D Interpolator.	92
A112	2Drag subset of xyz data into newInterpalator2D	93
A113	Choose Tools>Map Object and select newInterpolator2D	93
A114	Give a new name and choose put M for units.	94
A115	The new interpolated mesh	94

List of Tables

1	Individual xyz point information from ArcMap ASCII bathymetry file	15
2	Conversion equivalents from US Survey Feet to Meters	22
3	BlueKenue Channel Mesher and Mesh Generator tutorial values	30
4	Baxter tutorial inflow hydrograph values	81
5	Rating curve for downstream boundary condition	82

1 Pre-requisites

1.1 Computer Requirements

- 1. ArcGIS 10.1
- 2. OpenTELEMAC CFD suite (via auto- or self-install) Successful installation/compilation requires:
 - (a) Python 2.7.3
 - (b) gfortran compiler
 - (c) TortoiseSVN client
- 3. BlueKenue pre-/post-processing software
- 4. FUDAA pre-processing software
- 5. 64-bit operating system (ideal, but not required if self-installing)

You can download the Open TELEMAC automatic installer and FUDAA pre-processor from the TELEMAC-MASCARET website at:

http://www.opentelemac.org/

The geometric dataset can be obtained from the course webpage or from the US Army Corps of Engineers Hydrologic Engineering Center website at:

http://www.hec.usace.army.mil/software/hec-georas/downloads.aspx

BlueKenue Pre-/Post-processing program is available from the "Canadian Hydraulics Centre of the National Research Council Canada" website at:

http://www.nrc-cnrc.gc.ca/eng/solutions/advisory/blue_kenue_index.
html

If performing a self-installation of the TELEMAC-MASCARET suite, it is recommended to follow the **Python installation guidelines** along with a summary of installation/compilation tips for a gfortran/Python/Windows specific build (**Appendix B**).

1.2 Data Requirements

The data required for this tutorial is available at:

http://web.engr.oregonstate.edu/~leon/Teaching_transients.html Download both zip files on your local drive (e.g. C:\TELEMAC_tutorial), and unzip their contents. The TELEMAC and GeoRAS Data folder contain three sub-folders; one TIN dataset, a RASModel and one aerial image (as raster grid) as shown in Figure 1.

rganize 🔻 🛛 Include	in library 🔻 Share with 💌 Burn	New folder		- 61 (
Favorites	Name	Date modified	Туре	Size
🗾 Desktop	🌗 Aerials	10/22/2013 12:01	File folder	
🐌 Downloads 👘	📕 RASModel	10/22/2013 12:01	File folder	
Recent Places	🔋 🕕 Terrain	10/22/2013 12:01	File folder	
	baxter10.mdb	4/27/2012 2:43 PM	Microsoft Access	6,248 K
Libraries	🔕 Baxter10.mxd	6/8/2012 2:10 PM	ArcGIS ArcMap D	8,031 K
Documents				
🚮 Git				
J Music				
🔄 Pictures 🛛 👻				

Figure 1: Contents of unzipped TELEMAC tutorial data. Note: extract this data to a directory with no spaces

2 Extract bathymetry from ArcGIS surface

Accurate representation of the waterway geometry is a crucial component for successful hydraulic modeling. In this section, two methods are illustrated for extracting a bathymetric data from an ArcGIS Triangulated Irregular Network (TIN) terrain model. The first method involves extracting *xyz* bathymetric points from the surface, into an *ASCII text file* and the second method exports the bathymetric data as an *ArcView Shape File (.shp)*.

2.1 ArcGIS TIN to ASCII text file

Start ArcMAP, and enable the *3D Analyst* extension from the Customize > Extensions menu as shown in Figure 2.

Extensions		X
Select the extensions you want to use.		
Description:		
3D Analyst 10.1 Copyright ©1999-2012 Esri Inc. All Rights Reserved		
Provides tools for surface modeling and 3D visualization.		
	Close	•

Figure 2: Activate 3D Analyst extension from ArcMap Customize > Extensions menu

Add the Baxter river TIN data set to the data frame using the 'Add Data' command, and open the 'ArcToolbox' window, as shown in Figure 3.



Figure 3: Add TIN data to ArcMap and initialize the ArcToolbox

In order to generate the finite-element mesh, the bathymetric and topographic information needs to be extracted or imported from a CAD program. A universal method to represent the elevation data of a domain is to have the xyz coordinates of each node representing the surface. This can be accomplished in three steps using ArcMap and Excel:

- 1. Extract TIN-Nodes using 3D-Analyst Toolbox command (Figure 4): ArcToolbox > 3DAnalystTools > Conversion > FromTIN > TINNode
- 2. Convert TIN-Node feature class to ASCII text via Toolbox command (Figure 4): ArcToolbox > 3DAnalystTools > Conversion... > FromFeatureClass > FeatureClassZtoASCII
- 3. Open text file using Excel, remove point index numbering column, and save-as a new text file (e.g. Baxter_geometry.xyz)

Steps 1 and 2 are illustrated in Figures 4 through 6, and Step 3 details are illustrated in Figures 7 through 11



Figure 4: Extract and export ASCII xyz bathymetric data utilizing the 3D Analyst Tools toolbox

TIN Node	Select original ArcMap TIN
C:\CE540_telemac2d_tutorial\BaxterExample10 -	smaller\Terrain\baxter_tin
Output Feature Class	
C:\CE540_telemac2d_tutorial\BaxterExample10 - sm	aller\Terrain\Baxter_TIN_nodes.shp
Spot Field (optional)	
Tag Value Field (optional) Tag_Value	
ОК	Cancel Environments Show Help >>

Figure 5: Prompt from Step 1, converting TIN to TIN-Node Feature Class

Seature Class Z to ASCII	Select TIN-Node shape file
Input Feature Class	
C:\CE540_telemac2d_tutorial\BaxterExar Output Text File	ample 10 - smaller \Terrain \baxter_xyz_data
baxter_xyz.txt File Format (optional) GENERATE	Specify converted file location
Delimiter (optional) Specify nam SPACE file to be ex	me of text xported \checkmark
Decimal Notation (optional) AUTOMATIC	
Digits after Decimal (optional)	3
Decimal Separator (optional) DECIMAL_POINT	~
	OK Cancel Environments Show Help >>

Figure 6: Prompt from Step 2, converting TIN Node Feature Class to ASCII text

2.2 Converting ArcGIS ASCII text file to XYZ text file

Now that the xyz point information has been converted from the ArcMap TIN to the corresponding ASCII format, the next step requires a minor change in formatting and unit conversion using Microsoft Excel. Import the ASCII bathymetric information from ArcMap to Excel (e.g. *File ; Open ;* baxter_xyz.txt); each row of data represents an individual xyz point in the form illustrated in Table 1:

Table 1: Individual xyz point information from ArcMap ASCII bathymetry file

point-id	x-coord	y-coord	z-coord
(#)	(ft)	(ft)	(ft)
1	6417302.24	2048668.67	32.5123
•			•
•	•	•	•
238547	6417290.38	2048732.06	31.5000

For this tutorial, the original source data (e.g. **baxter tin**) had to be converted from US Survey Feet to the SI unit equivalent for use in TELEMAC-2D. This was done using the conversion described in Section 2.5.

Text Import Wizard -	Step 2 of 3
This screen lets you se below.	t the delimiters your data contains. You can see how your text is affected in the preview
Delimiters <u>T</u> ab Semicolon Comma Space <u>Other</u>	✓ Treat consecutive delimiters as one Text gualifier:
Data preview	Specify Space delimitation Finish text data import
0 6417302.2 1 6417310.8	4501799 2048668.67309242 32.5123 2806509 2048663.69465323 32.3907
2 6417310.1 3 6417302.6 4 6417292.6	9090414 2048689.17344517 32.4391 5689635 2048648.77956685 32.4745 4509903 2048666.82527429 31.5316
	Cancel < <u>B</u> ack <u>N</u> ext > <u>F</u> inish

Figure 7: Open the Baxter River ASCII text file using Excel, importing as a space delimited file

Format Cells					? 🔀
Number Alig	gnment Font	Border	Fill	Protection	
<u>Category:</u> General Number Currency Accounting Date Time Percentage Fraction Scientific Text Special Custom	Update Cell F avoid data tru (12 (12 (12) (12) (12) (12) (12) (12) (nple 00000000 mal places: <i>Format</i> to ncation 14.76543210) 34.76543210) 34.76543210)	3 🚔	Select 8 decir	mal places
				ОК	Cancel

Figure 8: Format the cells in order to avoid truncating the data

	<u>ຈ</u> ~ (× - ∥≂		baxter_xyz	.txt - Microsoft Excel	_		_			
File	Home Insert	Page Layout Formula	s Data Review \	/iew Developer	Team				∾ 🕜 🗆	e x
Paste	Calibri B I U -	$\mathbf{x}_{11} \mathbf{x}_{A} \mathbf{x} \equiv$	≡ ≡ ≫ · · ■ N	umber nited file	Condition	al Format	Cell	nsert ▼ Σ Delete ▼ 3	Sort & Find	1&
Clipboard	i G Fo	nt	(.i.xi) tuo dein	inted ine	Formattin	Styles	styles -	Cells	Editing	
	A1 -	<i>f</i> * 0								~
	А	В	С	D	E	F	G	Н	I.	=
1	0.00000000	6 17302.24501799	2048668.67309242	32.51230000						
2	1.00000000	6 17310.82806509	2048663.69465323	32.39070000						
3	2.0000000	6 17310.19090414	2048689.17344517	32.43910000						
4	3.00000000	6 17302.65689635	2048648.77956685	32.47450000						
5	4.00000000	6 17292.64509903	2048666.82527429	31.53160000						
6	5.0000000	6 1 D L G	1 4 0 1	, 000						
7	6.0000000	$_{611}$ Delete C	olumn A; Selec	t 100						
8	7.0000000	⁶ ¹ Column	A header, right	-click.						
9	8.00000000	6 1	1 11000001, 11g.11	000						
10	9.00000000	₆₁ and delet	e	000						
11	10.0000000	6 17301.83313964	2048688.56661798	32.55010000						
12	11.00000000	6 17301.29000000	2048714.80000000	32.60000000						
13	12.00000000	6 17300.14860729	2048729.08211065	32.60000000						
14	13.00000000	6 17309.39000000	2048721.20000000	32.50000000						
15	14.00000000	6 17327.39000000	2048723.00000000	33.50000000						
16	15.00000000	6 17326.32623054	2048687.07621770	33.39580000						
17	16.00000000	6 17335.37004546	2048688.62652166	33.30540000						-
	🛛 baxter_xyz / 🞾	7	1							► I
Ready	Calculate 🛅		Average: 119220.500000	000 Count: 238443	Sum: 284271	74461.0000000) 🗐 🗆 🗉] 115% —	0	- + _;;

Figure 9: Node IDs are not necessary for xyz format; delete the entire column of IDs

🔀 Save As				٤	×
😪 🌍 🗸 🕌 « BaxterEx	ample10 - smaller 🔸 Terrain 🔸 baxter_xyz_data		Search baxter_	xyz_data	P
Organize 🔻 New fold	der			!≡ ▼ (2	
Microsoft Excel	Name	Date modified	Type	Size	
★ Favorites	baxter_xyz.txt	10/22/2013 10:38	Text Document	11,549 KB	
Desktop					
📳 Recent Places		Save <i>xyz</i> dat	ta as a Tab deli	mited file]
ز Libraries		L			
Documents					
Git 👻					
File name: baxt	er xvz.bxt	K			-
Save as type: Text	(Tab delimited) (*.txt)				•
Authors: Giftor	rdMiears, Christop Tags: Add a tag		litle: Add a tr	tie	
Alide Folders		Tools	▼ Save	Cancel]

Figure 10: Save altered bathymetric ASCII text file as a Tab delimited text file

🔆 🌑 🗕 🔢 « CE540_telemac2d_tutorial 🕨 Baxter&	xample10 - smaller 🕨 Terrain 🕨 baxter_xyz_data	✓ 4y Search	baxter_xyz_data	x = .
Organize 🔻 🧻 Open 🔻 Print Burn	New folder		•== •	
🔆 Favorites 💻 Desktop	Name Daxter_geometry.xy2	Date modified 10/22/2013 10:43	Type XYZ File	Size 6,776 K
🐌 Downloads 📃 Recent Places	baxter_xyz.xyz	10/31/2013 9:33 PM 10/22/2013 10:38	Text Document XML Document	10,959 K 1 K
 □ Libraries □ Documents □ Git □ Music □ Pictures □ Subversion □ Videos 	Change the file ex this can be done by and pressing the F	tension to (.xyz) y selecting the fi 2 key to rename	; le	
Computer	• • • • • • • • • • • • • • • • • • •	m		
Text Document Size: 10.7 MB	5755 TH 564C COLOR 10/22/2015 10:50 TH			

Figure 11: BlueKenue utilizes a (.xyz) file extension, therefore the extension is simply updated to (.xyz)

2.3 Export ArcMap data as a Shape File

The ability to import **ArcView Shape Files** into BlueKenue is a powerful feature. The previous topographic data for the *HEC-GeoRAS* tutorial (e.g. levees, bridges, reservoirs, etc.) can readily

be included for analyzing TELEMAC-2D results and incorporating these features into the computational mesh.

For example, exporting the *HEC-GeoRAS* cross-sections, 3D levee and bridge information, can each be accomplished in a single step from ArcMap, as shown in Figures 12, 13 and 14.



Figure 12: Open ArcMap and add the RASGeometry data layer from baxter10.mbd database



Figure 13: Select dataset of interest from RASGeometry and Export Data

Export Data	X
Export: All features	-
Use the same coordinate syst	em as:
 this layer's source data the data frame the feature dataset you export (only applies if you export) 	Select layer's source data and provide an appropriate filename to a feature dataset in a geodatabase)
Output feature class: C:\CE540_telemac2d_tutoria	I\Bridges3D.shp
	OK Cancel

Figure 14: Exported data features, as Shape Files (*.shp), are simple to import using BlueKenue

Once the ArcView Shape Files are exported, importing them to BlueKenue is achieved through the File ; Import ; ArcView Shape File command, detailed in the subsequent section.

2.4 Representing TIN elevation data using Shape file

If the raw *xyz* values are not required, then you can simply perform Step 1 of the process described in Section 2.1 (i.e. Figure 4), save the newly created Shape File (.shp), and import the Tin-Node file directly into BlueKenue. To import an ArcMap Shape File to BlueKenue, use the *File ¿ Import ¿ ArcView Shape File* command as depicted in Figure 15.



Figure 15: BlueKenue is capable of importing ArcMap/ArcView Shape Files directly

2.5 Convert source data from US Survey Feet to Meters

The geographic projection of the original dataset is Oregon State Plane Coordinates in United State (US) Survey Foot units (ft). The exact project information in ArcMap:

NAD_1983_StatePlane_California_III_FIPS_0403_Feet

TELEMAC-2D exclusively uses International System of Units (SI), therefore the projection of the elevation data **must** be converted to equivalent SI units (i.e. meters). The conversion of US Survey foot to meters is shown below in Table 2:

Table 2: Co	onversion	equivalents	from US	Survey	Feet to	Meters
10010 2. 00		equivalents	nom ob	Juivey	1 001 10	10101015

US Survey Foot	Meter	Meter
(ft)	(m)	(m)
1.0	1200/3937	0.3048006096

Resulting information and datasets can be viewed in ArcMap after conversion so long as the geographic projection is changed to the meter equivalent:

NAD_1983_StatePlane_California_III_FIPS_0403

3 Pre-processing utilizing BlueKenue

BlueKenue is developed by the *Canadian Hydraulics Centre of the National Research Council* and is utilized in this tutorial to accomplish the following:

- 1. Generate the Baxter Finite-Element mesh
- 2. Create the boundary conditions influencing the system
- 3. Visualizing the TELEMAC-2D hydrodynamic results

3.1 Importing data to BlueKenue

Start BlueKenue and open the (.xyz) geometry data extracted from the ArcMap TIN, or import the Tin Node Shape File (Figure 16). In order to see your XYZ dataset, change the *Filetype* drop-down menu from *Selaphin* to *AllFiles* as shown in Figure 17.

🚟 BlueKenue -		
File Edit View Tools Run V	Vindow Help	
N 🛎 🗉 🎒 📲 💳 🤻 🖞	🎥 🧎 🏒 🟹 🗰 🔜 🐼 🗾 🗜 🐼 📰 🎙	?
WorkSpace	🗾 2D View (1)	
2D View (1)	Option 1: Select the bathymetry text	
	file (.xyz) using the <i>Open</i> command	
	Option 2: Import the <i>ArcView Shape</i>	
	<i>file</i> (e.g. TIN_Nodes.shp)	

Figure 16: Import the Baxter bathymetry using either the *Open* command or the *Import ¿ ArcView Shape file* command



Figure 17: When importing the bathymetry text file, toggle file types to All Files (*.*)





Figure 18: To view **Data Items** imported to BlueKenue, drag item of interest to a **Views** object (e.g. **2D View** (1))

The bathymetric data can also be viewed in an iso-metric **3D** View by opening a new **3D** View icon or through Window ; New **3D** View as shown in Figure 19.



Figure 19: Create a new viewing window using the **toolbar icons** or **Windows** command and drag the data object to the new view (e.g. **3D View (2)**)

3.3 Mesh Generation

TELEMAC-2D solves the depth-averaged Navier-Stokes equation utilizing both Finite-Element (FE) and Finite-Volume (FV) formulations. All of these formulations require that a spatial representation of the domain be created using a computational mesh. BlueKenue has several tools for mesh generation and editing. Mesh types that BlueKenue can generate are unstructured and regular (via **T3 Channel Mesher**) triangular meshes.

The basic requirements for creating a simple mesh in BlueKenue are outlined in Appendix A. For this tutorial, the following steps are illustrated for generating a mesh incorporating the Baxter River, Tule Creek, Flood-plains, and Levee embankment components:

- 1. Create New Closed Line around computational domain of interest
- 2. Create New Open Lines outlining the river channel, and levee, right and left extents
- 3. Open a new Channel Mesher via File ¿ New ¿ T3 Channel Mesher...
 - (a) Specify the number of cross-channel nodes
 - (b) Specify the mesh element length along the channel
 - (c) Drag and drop the right and left channel bank open-lines to the **RightBank** and **Left-Bank** objects
 - (d) Double click the T3 Channel Mesher and select Run
 - (e) This will be performed for both the Baxter River and Tule Creek channels
- 4. Represent the levee utilizing a new T3 Channel Mesher...
 - (a) Specify the number of cross-channel nodes
 - (b) Specify the mesh element length along the levee
 - (c) Drag and drop the right and left channel levee open-lines to the **RightBank** and **Left-Bank** objects
 - (d) Double click the T3 Channel Mesher and select Run
- 5. Open a new T3 Mesh generator via File ¿ New ¿ T3 Mesh Generator...
 - (a) Drag domain outline to **Outline** child-object
 - (b) Drag main-channel and creek mesh onto the SubMeshes child-object
 - (c) Drag levee mesh onto the **SubMeshes** child-object
 - (d) Drag 3D Line Set representing bridge abutments to HardLines child-object
 - (e) Double click the T3 Mesh Generator and specify Default Edge Length and Edge Growth Ratio

Following these steps will result in the generation of a hybrid unstructured triangular mesh that incorporates major topographic features in this flood-inundation scenario. The steps outlined above are described in additional detail, in their respective order, in the following subsections.

3.3.1 Steps 1 and 2: Create Open- and Closed-Lines in BlueKenue

Steps 1 and 2 involve creating a bounding polygon of the entire domain and the delineation of the river channel, and levee, right and left banks. Figure 20 shows the process of creating a **New Closed Line** to represent the extents of the computational domain. Once the **New Closed Line** icon is selected, you can use the cursor to define the points representing the outline. Figure 20 also shows the imported *HEC-GeoRAS* cross-section cuts as a reference for creating the outline (see Section 2.3 for details how)



Figure 20: Step 1; create a New Closed Line representing the domain extents



Figure 21: Step 2; Right and Left banks are necessary for all channels; either import the 2D-Lines (.i2s), or create them using **New Open Lines**

3.3.2 Steps 3 and 4: Create Channel Meshes in BlueKenue

The mesh for this tutorial includes three channel mesher components:

- 1. Baxter River channel mesh
- 2. Tule Creek channel mesh
- 3. Levee represented using channel mesh

Table 3 presents the values that will be used for the tutorial mesh parameters. Generating a channel mesh is illustrated in Figures 22 through 25

Table 3: BlueKenue Channel Mesher and Mesh Generator tutorial values						
BlueKenue Mesh Type	Mesh Component	Channel Interval/Edge Length				
		(-)	(M)			
T3 Channel Mesher	Baxter River	10	50			
T3 Channel Mesher	Tule Creek	10	30			
T3 Channel Mesher	Levee Mesh	3	20			
T3 Mesh Generator	Full Mesh	-	100			



Figure 22: For each channel mesh (i.e. 3 in total) a new **Channel Mesher** object should be initialized



Figure 23: Specify the **Open Lines** representing banks of the channel feature of interest (e.g. Baxter River) and specify the meshing parameters (Table 3)

Properties of: newT3ChannelMesh	× 1
Mesh Parameters	
CrossChannelNodeCount 3 (includ	ing banks)
AlongChannelInterval 1	Run
LeftBank RightBank Thalweg Mesh Met	a Data
Keyword Value	Once the Right and Left Bank
Name newT3ChannelMesh	objects are placed, and mesh
Type T3 Channe Mesh Generator	parameters are specified, Run will
Directory	generate the channel mesh
Filename Each channel mesh (e.g. should be renamed for ea identification	3 total) ise of
OK Apply	Cancel

Figure 24: Rename and **Run** the Channel Mesher



Figure 25: Resulting Baxter River Channel Mesh and corresponding parameters



Figure 26: Resulting Baxter River Channel Meshes



Figure 27: Baxter River Channel Meshes components

3.3.3 Steps 5: Generate Combined Mesh in BlueKenue

Once the submesh components are created, the domain mesh combining all elements can be generated. Due to the large size of the domain (~ 20 KM), a coarse mesh will be generated for demonstration purposes. The objects needed for creating this mesh include:

- 1. Baxter River channel mesh
- 2. Tule Creek channel mesh
- 3. Levee channel mesh
- 4. Baxter domain outline (Baxter_outline_20m_resampled.i2s)
- 5. Bridge 3D elevation line (Bridges3d_20m_resampled.i3s)

All of the above items are available from the tutorial dataset directory, BK_baxter_tutorial_files.

Open a new T3 Mesh generator via File ; New ; T3 Mesh Generator... and enter the settings specified in Figure 28 and select Apply, then exit.

Properties of: B	Baxter_tutorial_	mesh		8
- Mesh Parame	ters	Deselee	<u>et</u> Resample C	Outline
🔲 Resample	Outline		📝 Auto Smooth	Mesh
🔽 Edge Grov	wth Ratio 1.2		🔲 Resample Lii	nes Only
Default Edge	Length 1		B	un
Meta Data Keyword	Value Set	Default kimum el	Edge Length lement size des	to sired
Title			1	+
Name	Baxter_tutor	al_mesh		
Type Directory	13 Mesh ve	nerator	_	
Filename	Change T	3 Mesh		
	Generato	r Name		
			_	
OK		Apply	Cancel	

Figure 28: Settings for Baxter tutorial T3 Mesh Generator

BlueKenue mesh generator utilizes a Delaunay triangulation method and can accommodate a wide range of complex topographic and bathymetric features. Following Step 5 of the Mesh Generation process, the location of each component in the **T3 Mesh Generator** are specified. Figure 29 illustrates the component locations from Step 5.



Figure 29: Mesh components added to Mesh Generator
Properties of: Bay	ter_tutorial_mesh		8			
-Mesh Paramete	Mesh Parameters					
📃 Resample O	utline	🔽 Aut	o Smooth Mesh			
Mesh Parameters Mesh Parameters Resample Outline Edge Growth Ratio 1.2 Default Edge Length 100 Run Meta Data Keyword Value Title Name Baxter_tutorial_mesh Type T3 Mesh Generator Directory Filename						
Default Edge Le	Default Edge Length 100 Run					
Meta Data						
Keyword	Value					
Title			+			
Name	Baxter_tutorial_mesh					
Туре	T3 Mesh Generator					
Directory						
Filename						
OK	Apply		Cancel			

Figure 30: Execution of the mesh generator



Figure 31: Resulting 2D triangular mesh generated using T3 Mesh Generator

3.4 Interpolate bathymetry to the mesh

Section 3.3 describes the process of creating a two-dimensional triangular mesh to represent the domain extents. It can be noticed that the mesh initially has a constant value assigned to each mesh node (e.g. EL. 0 m). In order to map elevation values to the nodes of a mesh, BlueKenue has a tool called **2D Interpolator...**.

A primary function of BlueKenue is to project, or interpolate the values of one dataset, onto another. In this tutorial, the bathymetric and topographic information are contained in several components, one of which is <code>baxter_xyz_meters.xyz</code>. Mapping the bathymetry to a mesh can be completed through the following steps:

- 1. Create a new 2D Interpolator through File ; New ; 2D Interpolator...
- 2. Drag and drop all relevant elevation datasets onto the 2D Interpolator object
- 3. Select the mesh of interest, and go-to Tools ¿ Map Objects...
- 4. Find the 2D Interpolator object on the list, select and press OK
- 5. View the resulting interpolation in a 2D-View or 3D-View

Step 1, create a new 2D Interpolator object through File ; New ; 2D Interpolator... .

🚈 BlueK	enue - [2D View (6)]					
🗾 File	Edit View Tools Run Window	Help				
Ľ	New	×		SELAFIN Object	t	
	Open			Boundary Con	ditions (Conlin	n)
.	Import	•		T3 Mesh Gener	rator	
	Base Maps			T3 Channel Me	esher	
	Save			Triangulation		
	Save Copy As			Regular Grid		
	Load WorkSpace			Table		
	Save WorkSpace			Points Open Line Closed Line		
	Print					
	1 Briders 2d 20m second ad 25					
	2 geometry, byster olf			2D Interpolator		
	3 Levee xvz pts.xvz			SED Run		
	4 levee_xyz_pts.xyz		POO			
	5 levee_density_polygon.i2s		L .			
	6 Bridges3d.i3s		L .			
	7 tule_right_bank.i2s		poo)		
	8 tule_left_bank.i2s					
	Exit					
	Tule Creek mesher Mesh	61	- 8000			
BlueKenue - [20 File Edit Vi C New Open Import Base Map Save Save Cop Load Wo Save Wo Print 1 Bridges 2 geome 3 Leve_y 4 levee_x 5 levee_d 6 Bridges 7 tule_rig 8 tule_lef Exit Exit Exit C New C Con C Con C C C C	Baxter_River_mesher_Mesh			,		
	New Mesh (NodeType)					
-4	newInterpolator2D			1954000	1956000	1958000
	ConstraintOutline	Crea	ate a r	new 2D Interpola	tor object	

Figure 32: Create a new **2D Interpolator** object

Step 2, drag and drop all relevant elevation datasets onto the 2D Interpolator object.



Figure 33: Drag elevation datasets to 2D Interpolator object

Step 3, select the mesh of interest, and utilize the **Tools ; Map Objects...** command to interpolate elevation data to the mesh nodes.



Figure 34: Select the mesh and execute the Map Objects command (e.g. Tools ¿ Map Objects)

Step 4, find the **2D Interpolator** object on the list, then select, and press OK.

Available Objects
Levee_mesher_Mesh
Ridges3d_20m_resampled
✓ Levee_mesher_Mesh ✓ Tule_Creek_mesher_Mesh
River_mesher_Mesh
newInterpolator2D
OK Cancel

Figure 35: Select the elevation source data to map, or interpolate, onto the selected object (mesh)

Processing		
Interpolating from: newInterpolator2D		
Target mesh: New Mesh		*
Processing 24469 Nodes Using Inverse Distance Interpolation.		
Interpolation finished in: 0.000000 sec.		
		Ŧ
Done		
ОК	Cancel	

Figure 36: When Map Object command is finished, a progress window will appear

Step 5, once the elevation data is mapped to the mesh, drag the mesh object to a **2D-View** or **3D-View** to inspect the interpolation.



Figure 37: Drag the interpolated mesh onto a 2D View Object to view the mapped bathymetry



Figure 38: Drag the interpolated mesh onto a **3D View** Object to view the mapped bathymetry

3.4.1 Levee height adjustment

Utilizing the steps outlined in Section 3.4, new levee heights can be imposed and mapped, or interpolated, to the bathymetric mesh. A method to adjust the crest height of the levee involves adjusting the **xyz** point file **levee_xyz.xyz**. This is accomplished utilizing the BlueKenue calculator tool. Select the **levee_xyz.xyz** object, go-to **Tools** *¿* **Calculator...**, and the xyz dataset can be vertically adjusted. As the calculator directly alters the **xyz** point file, it is recommended to make copies of the levee elevation data file prior adjustment.

Calculator	
Object levee_xyz	
Function Plus 0	Units M
ОК	Cancel

Figure 39: Adjust the levee height prior to interpolating the domain mesh by using the BlueKenue calculator



Figure 40: Apply adjusted levee heights using a **2D Interpolator**

If creating a new levee using the **T3 Channel Mesher**, the following are the steps used to generate **levee_xyz.xyz** used in Section 3.4.

- 1. Create T3 Channel Mesh representing levee
- 2. Use **2D Interpolator** to map original levee elevations (from Exported Shape File)
- 3. Save mapped levee mesh as a text file (*.t3s)
- 4. Open levee mesh file in text editor, delete mesh face info, leaving xyz vertex info at each levee mesh node
- 5. Save as a xyz file, and include in **2D Interpolator** used for combined mesh to incorporate changes exactly

3.4.2 Viewing cross-sections of the mesh



Figure 41: Create a new line where cross-section view is desired

Resample LineSet	Disht	
Source LineSet Name cross_section_view	object	and select resample to control ing density of cross-section
Attribute	Value	
Line Count	1	
Point Count of Line	2	
Length of Line	1530.0	
Average Delta of Line	1529.98	Select Equal Distance and a spacing of 1 M ; this will create a line with points every 1 m along
Resample Options		the original cross-section
Method Equal Distance Delta 1	•	
Create New LineSet New Name Resampled cross_s	ection_view Cancel	

Figure 42: Resample the newline to increase number of sampling points



Figure 43: Once resampled, use command **Map Object** to choose the source surface to sample from



Figure 44: Drag the mapped, resampled line onto a new 1D View to see the cross-section

TELEMAC-2D input file generation 4

A TELEMAC-2D hydrodynamics simulation requires a minimum of three input files:

- 1. Geometry File (*.slf)
- 2. Boundary Conditions File (*.cli)
- 3. TELEMAC-2D simulation parameters file (*.cas)

Items 1 and 2 are prepared in BlueKenue, and FUDAA Pre-Processor will be used to set up the TELEMAC-2D parameters file.

BlueKenue 4.1

4.1.1 **TELEMAC-2D Geometry File**

The geometry file for TELEMAC-2D can be represented in several formats. Using BlueKenue, a formatted binary file will be created that includes the bathmetric mesh. The process of creating a geometry file is achieved through the following steps:

- 1. Create a New SELAFIN File and rename (e.g. geometry_baxter) Step 1 - Page 47
- 2. Add the bathymetric mesh as a New Variable specified as BOTTOM Step 2 - Page 49 Step 3 - Page 50
- 3. Map the bathymetry to the new child-object mesh, **BOTTOM**
- 4. Save geometry file to the TELEMAC-2D simulation directory Step 4 - Page 51

Step 1, create new Selafin File utilizing either the New SELAFIN Object icon or through File : New ¿ SELAFIN Object (i.e. Figure 45).



Figure 45: Create a new geometry object using either the toolbar icon or File command

roperties of:r	newSelafin	
Meta Data		
Keyword	Value	
Title	newSelafin	+
Name	geometry_baxter	
Туре	Telemac 2D Selafin File	
Directory		
Filename		
ОК	Apply	Cancel

Figure 46: Double-click the New Selafin object, rename, and select Apply

Step 2, add the bathymetric mesh as a **New Variable** specified as **BOTTOM** as shown in Figures 47 and 48.



Figure 47: Right-click the geometry object and select Add Variable

Add New	SELAFIN Variable	Select New Mesh from the drop-down list
Source	Mesh and Attribute Data	
Mesh	New Mesh	
Attribut	e Name NodeType A	Assign to Bottom variable
New Va	riable Properties	K
Name	BOTTOM	▼
Units	М	
Cop De	y Node Values from Source fault Node Value 0	
	OK Cancel	

Figure 48: Select the computational domain mesh and add as a **BOTTOM** variable



Figure 49: Geometry object with **BOTTOM** variable added

Step 3, map the bathymetry to the new object **BOTTOM** as shown in section 3.4. Again, inspect that the elevation data is represented on the geometry **BOTTOM** child-object in a **2D View** or **3D View** window. An example of a non-interpolated mesh and correctly interpolated mesh are shown below in Figure 52.



Figure 51: Correctly interpolated / Mapped BOTTOM - GOOD

Figure 52: Comparison of bad versus good BOTTOM interpolation

Step 4, save the geometry file under the TELEMAC_simulation_files directory.

•					
🚟 Save As		_			8
C C C C C C C C C C C C C C C C C C C	_Baxter_tutorial_files	es 🕨 👻	Search TELEMA	NC_simulation_f	<mark>P</mark>
Organize 🔻 New folde	r				0
Documents ^	Name	Date modified	Туре	Size	
Git Music	Ctrl_section_results	11/10/2013 7:36 PM	File folder		
	📗 inflow_hydrograph	11/10/2013 7:36 PM	File folder		
Pictures	AC_Baxter_tutorial_files > TELEMAC_simulation_files > AC_Baxter_tutorial_files > TELEMAC_simulation_files > der BET Pate modified Type Size Ctrl_section_results 11/10/2013 7:36 PM File folder inflow_hydrograph 11/10/2013 7:36 PM File folder results 11/10/2013 7:36 PM File folder results 11/10/2013 7:36 PM File folder metry_baxter.slf mate 2D Selafin File (BigEndian) (*.slf) Save Cancel				
	Source Add Organize New folder Image: Computer Image: Computer <th></th> <th></th>				
E Computer Computer Ieon (\attic) (A:) Constant Cons					
File name: geom	etry_baxter.slf				•
Save as type: Telema	ac 2D Selafin File (BigEndian) (*.slf)				•
) Hide Folders			Save	Cancel	

Figure 53: Save geometry object under the TELEMAC simulation directory

4.1.2 TELEMAC-2D Boundary Condition File

1.	Create a New ¿ Boundary conditions (conlim) file for BOTTOM	Step 1 - Page 52
2.	Rename the boundary conditions file (e.g. bc_baxter)	Step 2 - Page 54
3.	Drag the boundary conditions object to a 2D-View object	Step 3 - Page 55
4.	Prescribe boundary conditions at the inflow and outflow domain locations	Step 4 - Page 56
5.	Save boundary conditions file to the TELEMAC-2D simulation directory	Step 5 - Page 60
6.	Save boundary conditions child-object to the T2D simulation directory	Step 6 - Page 60

Step 1 , create a New ; Boundary conditions (conlim) file for the geometry child-object Bottom



Figure 54: Create a new Boundary Conditions file for the BOTTOM object



Figure 55: Assign the new **Boundary Conditions** file to the **BOTTOM** object

Step 2, rename the boundary conditions file (e.g. bc_baxter). This can be done by doubleclicking the **BC BOTTOM** object and changing the **Name** under the **Meta Data** tab (Figure 56).



Figure 56: Rename the **BOTTOM BC** object for ease of identification

Step 3, drag the boundary conditions object to a **2D-View** object.



Figure 57: All boundary nodes are assigned as Closed boundary (wall) by default



Figure 58: Zoomed in view of the default bc baxter object

Step 4, prescribe boundary conditions at the inflow and outflow domain locations

Each node along the boundary has a specific code and color representing what type of boundary it is. For example, **Closed boundary (wall)** nodes are brown, **Open boundary with prescribed Q** nodes are blue, and **Open boundary with prescribed H** nodes are green, where Q is volumetric flowrate, and H is water depth. The full list of boundary nodes are shown below in Figure 59

	CONLIM Boundary Segment Editor					
Boundary Name newBoundary (19564 - 19384)						
Boundary Co	ode	Closed boundary (wall) 👻				
Tracer Code		Closed boundary (wall) Open boundary with prescribed Q Open boundary with prescribed H				
HBOR	U	Open boundary with prescribed Q and H				
0		Open boundary with prescribed UV and H				
0		Open boundary with incident Waves				
0		Custom				

Figure 59: TELEMAC-2D Boundary condition node types



Figure 60: Boundary condition overview for the Baxter River tutorial

Updating and adding boundary condition segments is performed on the **bc baxter** object within a **2D View** window. To create a segment of nodes, representing a single boundary condition, **select the starting edge node** of the domain where the boundary condition begins, then holding the **Shift** key, **select the end node** where the boundary condition segment ends. For example, Figure 64 shows the delineation of the upstream Baxter River boundary condition as an **Open boundary with prescribed Q**. The process of prescribing a boundary condition nodes is illustrated in Figures 61 through 64 below.



Figure 61: Select the starting edge node where the boundary condition segment begins



Figure 62: Select the end node of the segment and right-click to Add Boundary Segment

CONLIM Bound	ary Segme	nt Editor						x
Boundary Name newBoundary (19606 - 19338)								
Boundary Code	Closed b	oundary (w	all)		- 222	2		
Tracer Code	Tracer Code Open boundary (wai) Open boundary with prescribed Q							
HBOR	U Open bo	Open boundary with prescribed Q and H				BTBOR	NBOR	
0	Open bo Open bo	undary with undary with	prescribed	UV UV and H	0	0	19606	
0	Open bo	undary with	incident Wa	aves	0	0	19602	
0	Custom	-			0	0	19600	-
0	0	0	0	0	0	0	19596	=
0	0	0	0	0	0	0	19591	
0	0	0	0	0	0	19588		
0	0	0 0 0 0				0	19584	
0	0	0	0	0	0	0	19578	
0	0	0	0	0	0	0	19573	
0	0	0	0	0	0	0	19564	

Figure 63: Apply an **Open boundary with prescribed Q** BC code to the Baxter River upstream reach



Figure 64: Boundary condition applied to bc baxter object

Step 5 and Step 6 , save boundary conditions file and boundary conditions child-object (.cli) to the TELEMAC-2D simulation directory



Figure 65: Save **both** boundary condition objects to the simulation directory

4.2 FUDAA Pre-pro

For this tutorial, the TELEMAC-2D parameter files will be provided, however creating a new hydraulic project is detailed below.

4.2.1 TELEMAC-2D parameters file (.cas)

- 1. Open FUDAA Pre-Processor using executable (*.jar)
- 2. Launch the hydraulic project editor,
- 3. Create a new TELEMAC-2D hydraulic project,
- 4. Specify the TELEMAC-2D files to be used:
 - (a) boundary conditions
 - (b) geometry
 - (c) results filename, etc.
- 5. Save the hydraulic project, $\begin{bmatrix} 1 \\ save \end{bmatrix}$

Creating a new TELEMAC-2D project is illustrated below in Figures 66 through 74

<u>File E</u> dition <u>Applications</u> <u>Bookmarks</u> <u>H</u> elp	
🚖 🖸 R, M, 🗏 🗉 🔟	
C:\CE540_telemac2d_tutorial\TELEMAC_Baxter_tutorial_files\TELEMAC_simulation_files	5 🖌
All	
S C() V Name	
bexter bc2 bruter bc2 bc_bter cli bc_bter cli crating_curve rating_curve results Navigate to TELEMAC_simulation_files directory	
22M	

Figure 66: FUDAA Supervisor initialized screen

If creating a new hydraulic project, launch the **hydraulic project editor** by selecting the icon, Create a new TELEMAC project using the **Create...** icon,

E Create a new Telemac project	tt X
Serafin file :	tutorial_files\TELEMAC_simulation_files\geometry_baxter.slf
Steering file :	
Dictionary :	telemac2d 🗸
• version of the dico file :	v6p1
Coad a dico file :	
Language :	English
Boundary conditions file:	Baxter_tutorial_files\TELEMAC_simulation_files\bc_baxter.cli
Optimise with OLB	Configure
0	✓ <u>V</u> alidate 🔀 Ca <u>n</u> cel

Figure 67: Populate this window with necessary hydraulic project files

E S	ering file :	x
Boo	narks: 😭 🔍 💌	/ 🛛
Sa	In: TELEMAC_simulation_files	
	trl_section_results	
	nflow_hydrograph	
	ating_curve Specify steering file name (* cas)	
	esults	
	oc_baxter.bc2	
	oc_baxter.cli	
	jeometry_baxter.slf	
File	lame: baxter_unsteady.cas	
File	of <u>Type</u> : All Files	-
	Save Cano	cel

Figure 68: Under the **Steering file** field, specify the name of your parameters file (e.g. **bax-ter_unsteady.cas**

E File		
Bookmarks:	1	▼ ♥ Z
Save In:	TELEMAC_simu	lation_files ▼ 🗟 🔂 🗂 📴 🗖
ctrl_section	on_results	
inflow_hyd	lrograph	Select the boundary condition file (*.cli)
📑 rating_cur	ve	J J J J
results		
bc baxter	.bc2	
bc_baxter	.cli	
geometry_	baxter.slf	
I		
File <u>N</u> ame:	bc_baxter.cli	
Files of <u>Type</u> :	All Files	-
		Save Cancel

Figure 69: Under the **Boundary conditions** field, specify the file location of bounary conditions file, **bc_baxter.cli**

E Serafin file :	x
Bookmarks: 📄	▼ ♥ Z
Look <u>I</u> n: 📑 TE	ELEMAC_simulation_files
ctrl_section	_results
inflow_hydr	ograph Select the geometry file
rating_curv	e
results	
bc_baxter.b	0.2
bc_baxter.c	li
geometry_b	axter.slf
File <u>N</u> ame:	jeometry_baxter.slf
Files of <u>Type</u> :	All Files 🗸 🗸
	Open Cancel

Figure 70: Under the **Serafin file** field (i.e. TELEMAC geometry file format), specify the location of **geometry_baxter.slf**

Once the core files have been specified, select **Validate** (e.g. Figure 67) to begin specifying TELEMAC-2D parameters.



Figure 71: Select the General Parameters tab to access the project parameters

File	Edition	Project	Windows	<u>H</u> elp													
D		凹		10	-		0	G		12	(3)	68	8				
Create	e Open	Save	Print	Undo	Redo	Select	Find	Arrang	. Palett.	Export.	Export.	Copy t.	Super.				
8 -	-																
2	diff	88 🖂	R _/ Ma														
=	G	noral para	motore 🔅														
	Desis			4000000000													<u>88</u>
×	Proje	ct Key	words	soundar	y conditio	ons											- 11
	Projec	t's name:															
	Main fi	ile:	C:\CE540	telema	ac2d tuto	rial\TELE	MAC F	Baxter ti	itorial fil	es\TELEN	IAC sim	ulation	files\bay	ter unst	eadv.cas		
										U U U U U U U U U U U U U U U U U U U			inco as as		Judjiede		
	Projec	t s type:	telemacz														
	Last s	ave:	Nov 11, 2	013 10:4	44:25 PM												
	State:		Modified	project:	: model pa	arameter	s and g	graphic p	roperties								
			Valid proj	iect													
			Valia proj													E 11	
			Keyword:		h	c baytor	cli									Files	411
	RE	ESULTS FI	ILE	ISTILL	re	sults/ba	ter un:	steadv r	esults.slf								- 11
	🔌 GE	OMETRY	FILE		g	eometry_	baxter	r.slf									- 11
	PF	REVIOUS	COMPUTAT	ION FIL	E re	sults\ba	ter_SS	_hotstar	slf								
			INDARIES F	FILE	in	flow_hyd	rograpi	h\hydrogr	aphs_ba	xter.liq							- 11
	•																



E File
Bookmarks: 😭 🔍 🗹
Look In: results
File Name: unsteady_baxter_results.slf
Files of <u>Type</u> : All Files
Open Cancel

Figure 73: Select the **Results File** field to specify where to write the T2D results

General parameters Project Keywords	Searching and viewing your hydraulic project parameters are easily accessed						
$\mathbb{N} \otimes$		Name		Value			
Name A	ABSCISSAE OF SOURCES AIR PRESSURE BINARY DATA FILE 1	Under the Keyw	ords tab, a	Ill parameters			
State	BINARY DATA FILE 2 BINARY RESULTS FILE BOTTOM SURFACES DELW BOTTOM TOPOGRAPHY FILE	listed with short	description	18			
Error E	BOUNDARY CONDITIONS FI	OF TRACERS	bc_baxter2.0	cli			
Normal Advanced Expert	COMPUTATION CONTINUED)					
Headings	CORIOLIS COEFFICIENT COST FUNCTION		0.			-	
EQUATIONS EQUATIONS, BOUNDARY	COUPLING PERIOD FOR SIS	SYPHE MAWAC	1				
EQUATIONS, SOURCE T	CRAY NAME DEBUGGER		cicraya 0				
INPUT-OUTPUT, FILES		ITABLE	telemac2dit	el2d_VVV PPP telemac2dl el2d_VVV PPPItelemac2dl DEFAULT EXECU			
INPUT-OUTPUT, INFORM L NUMERICAL PARAMETE V NUMERICAL PARAMETE V NUMERICAL PARAMETE E	Default value: Value: bc_b Error:	axter2.cli	Defat telen	ult value: nac2d tel2d_VVV PPP tele	mac2dMMMVVV.exe		
NUMERICAL PARAMETE PHYSICAL CONSTANTS	Dependent keywords: nor Help	ne					
	Type: Character Level: Normal Name of the file containing t This file is filled automatical through colours that are ass	the types of boundary condit ly by the mesh generator thi igned to the boundary node	ons. ough s.			=	
					-	·	

Figure 74: The **Keywords** tab is used for viewing and understanding the hydraulic project parameters

4.2.2 Example unsteady parameters file (.cas)

To view the example TELEMAC-2D parameters file, initialize FUDAA Pre-pro and launch the hy-

draulic project editor, E. Next, open the baxter_unsteady.cas file from the TELEMAC_simulation_files directory to load and view the project parameters.



Figure 75: Project files for example TELEMAC-2D unsteady parameters file

General parameters	Apply filter to view only the M TELEMAC-2D project parame to clear the filter and view all F	lodified ters; right-click Keywords
	BOUNDARY CONDITIONS FILE	bc baxter2 cli
Name	COMPATIBLE COMPLITATION OF FLUXES	
	OMPUTATION CONTINUED	
[]	ERICTION COFFEICIENT	0.06
State	GEOMETRY FILE	geometry_bayter2 slf
Not modified	GRAPHIC PRINTOUT PERIOD	60
Modified	LAW OF BOTTOM ERICTION	4: MANNING
Error	LIQUID BOUNDARIES FILE	inflow hydrograph/hydrographs baxter lig
U1	LISTING PRINTOUT PERIOD	10
Mode	MASS-BALANCE	
Normal	NUMBER OF TIME STEPS	15240
Advanced	OPTION FOR LIQUID BOUNDARIES	1.1.1
Expert	PREVIOUS COMPUTATION FILE	results/baxter_hotstart.slf
<u>. </u>	RESULTS FILE	results/unsteady_results.slf
Headings	SECTIONS INPUT FILE	ctrl section results/control sections.txt
BOUNDARY CONDITIONS	SECTIONS OUTPUT FILE	ctrl section results/unsteady ctrl results.txt
EQUATIONS	STAGE-DISCHARGE CURVES	0:0:1
EQUATIONS, BOUNDARY	STAGE-DISCHARGE CURVES FILE	rating curve/rating curve So 001.bt
EQUATIONS, INITIAL CO	SUPG OPTION	1:1:1:1
EQUATIONS, SOURCE T	TIME STEP	5
FILES	TURBULENCE MODEL	3: K-EPSILON MODEL
GENERAL	VARIABLES FOR GRAPHIC PRINTOUTS	U.V.B.H.S.F
INPUT-OUTPUT, FILES	VELOCITY PROFILES	5;5;1
INPUT-OUTPUT, GRAPHI		
INPUT-OUTPUT, INFORM	Name: FRICTION COEFFICIENT	A
NUMERICAL PARAMETE	Heading: EQUATIONS	
NUMERICAL PARAMETE	Default value: 50.	
NUMERICAL PARAMETE	Value: 0.06	
PHYSICAL CONSTANTS	Error:	
	Dependent keywords: none	
	Help	
	formulation. It is noteworthy that the meaning of this fi	
	according to the selected formula (Chazy Strickler, et	
	1 : linear coefficient	~J.
	2 : Chezy coefficient	
	2 : Strickler coefficient	
	A : Manning coefficient	
	5 : Nikuradse grain size	
	5. Nikurause grain size	

Figure 76: Using the filters under the **Keywords** tab can help view pertinent parameters to your project quickly

```
🖶 rating_curve_So_001.txt 🗵 📙 baxter_unsteady.cas 🖾 🔚 baxter_unsteady2.cas 🖾 🔚 T2DCAS_fast_impl_
  /-----
1
                                         L
  / TELEMAC2D Version v6p1 Nov 14, 2013
2
                                         ľ
3
  / nom inconnu
4
  /-----
                                         -
5
6
  /-----
7
  / BOUNDARY CONDITIONS
8
  /-----
9
10
  STAGE-DISCHARGE CURVES =0;0;1
11
12
13
  /-----
14
  / EQUATIONS
  /-----
15
16
17
  BOTTOM SMOOTHINGS =1
18
19 FRICTION COEFFICIENT =0.06
20
21 LAW OF BOTTOM FRICTION =4
22
23 TURBULENCE MODEL
              =3
```

Figure 77: Once changes are saved from FUDAA, the parameters file is well organized and ready for the TELEMAC-2D simulation

5 Running TELEMAC-2D simulation

- 1. Open DOS command window
- 2. Test that **Python** and **gfortran** is working
- 3. Change working directory to TELEMAC-2D simulation directory
- 4. Execute TELEMAC-2D simulation

5.1 TELEMAC-2D from the DOS Command Prompt

The following figures (78 to 85) illustrate the process of opening a new DOS command prompt, testing Python and gfortran installations, and executing TELEMAC-2D.

Programs cmd.e cmd.e	(2) xe 03
Creat	te a new DOS Prompt through:
	Start > cmd > enter
₽ See m	esults
cmd	× Log off +
S	🗧 🖸 💓 😰 🔛

Figure 78: initialize new DOS command prompt







Figure 80: Check Python version, should be 2.7.3 or 2.7.5



Figure 81: Test gfortran installation


Figure 82: Copy the full-path directory address to the TELEMAC_simulation_files folder



Figure 83: Using the change-directory DOS command, **cd**, change the directory to the location of the **TELEMAC_simulation_files** folder



Figure 84: Within the input file working directory, execute TELEMAC-2D using the command, **telemac2d.py name_of_input_file.cas**

C:\Windows\system32\cmd.exe	8		
_files\TELEMAC_simulation_files\baxter_unsteady2.cas_2013-11-13-23h37min37s\T2 LI	2DC 🔺		
Munning your simulation :			
C:\CE540_telemac2d_tutorial\TELEMAC_Baxter_tutorial_files\TELEMAC_simulation_f es\baxter_unsteady2.cas_2013-11-13-23h37min37s\out_telemac2d.exe	Fil		
LISTING OF TELEMAC-2D			
T E L E MM MM A A C			
$\begin{array}{ccccc} \mathbf{I} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{M} & \mathbf{M} & \mathbf{A} & \mathbf{A} & \mathbf{C} \\ \mathbf{T} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{M} & \mathbf{M} & \mathbf{A} & \mathbf{A} & \mathbf{C} \\ \mathbf{T} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{E} & \mathbf{M} & \mathbf{M} & \mathbf{A} & \mathbf{A} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} \\ \mathbf{C} & \mathbf{C} & \mathbf{C} $			
2D VERSION 6.3 FORTRAN 90 WITH SEVERAL TRACERS			
COUPLED WITH SISYPHE AND TOMAWAC			
DIFFERENT NUMBER OF PARALLEL PROCESSORS: DECLARED BEFORE (CASE OF COUPLING ?): Ø			
TELEMAC-2D: 1 VALUE ØISKEPT			

* LECDON: * * AFTER CALLING DAMOCLES *			
* CHECKING OF DATA READ * * IN THE STEPPING FILE *			

Figure 85: If there are no errors, the simulation will execute until finished

5.2 TELEMAC-2D Steady-state simulation

Good initial conditions are imperative for reducing overall simulation times as well as ensuring that no additional factors influence the final results. In general, achieving a steady-state solution occurs when the discharge and water surface elevation are no longer changing as a function of time (i.e. $\frac{\partial Q}{\partial t} = 0, \frac{\partial H}{\partial t} = 0$). Here we can utilize the fully-implicit descritization of TELEMAC-2D to speed up the computation for achieving an initial steady state.

An example steady-state parameters file has been included in the **TELEMAC_simulation_files** directory.

5.2.1 Create a HOTSTART file from previous computation

A **HOTSTART** file in this context refers to a simulation, continuing from the final time-step of a previous simulation. For example, a steady-state solution could have been reached for a given mesh configuration. If the computational mesh were changed for the same domain (added levees, bridge piers, etc.), instead of re-simulating from an at rest condition, the values from the previous mesh/solution can be mapped to the new, updated mesh. It requires a few steps, but ultimately saves time, especially when altering meshes on larger domains where reaching a steady-state solution can take hours or days to complete.

Figures 86 through 95 illustrate the process of creating a HOTSTART file from one mesh to another, using the **BlueKenue Calculator** tool.

🚟 Open				8
😪 🌍 🗸 📗 « TELEMAC	_simulation_files 🕨 results	🕶 🍫 Search resul	ts	Q
Organize 🔻 New folde	r		· · ·	0
*	Name	Date modified	Туре	
Cibraries	baxter_SS_hotstart.slf	11/6/2013 9:14 PM	SLF File	
Git				
J Music				
Pictures				
Subversion				
Videos				
🖳 Computer				
🖵 leon (\\attic) (A:)				
🚽 rivers (\\pandora				
SDisk (C:)				
💿 DVD RW Drive (D				
□ GiffordMiears_Ex ▼	<			•
File na	me: baxter_SS_hotstart.slf	✓ All Files (*.*)		•
		Open	Cance	el l

Figure 86: Open the previous results file in BlueKenue (e.g. Baxter_SS_hotstart.slf)

Meta Data		
Keyword	Value	
Title	newSelafin	+
Name	baxter_hotstart	
Туре	Telemac 2D Selafin File	
Directory		
Filename		

Figure 87: Create, and rename a new Selafin file for the HOTSTART components to be stored



Figure 88: If the previous results file has several timesteps, be sure to **Animate** the results, and fast-forward to the final **frame**

Select the **VELOCITY UV** object from the previous results file, **go-to Tools ; Calculator...** and extract a copy of the U velocity component of the object using the calculator method shown in Figures 89 and 90. This is necessary because BlueKenue automatically combines U and V components of velocity when loading result files, where TELEMAC-2D requires that each variable component be available separately.

Calcu	lator	X				
-Varia	ables Start	End				
A	VELOCITY UV V					
В	0.0 👻					
C	0.0 🗸					
D	0.0 🗸					
Ехр	Expression					
A		•				
		~				
Res	Result					
Nan	he hotstart_velocity_V Units	M/S				
	Evaluate Cancel					

Figure 89: BlueKenue Calculator method for extracting V component velocities as a new object $hotstart_velocity_V$

Calcu	ulator	X	3	
-Varia	Variables Start			
Α	VELOCITY UV 🗸 U 🗸			
В	0.0 💌			
С	0.0 🗸			
D	0.0 🗸			
Expr	ression	-		
		*		
		Ŧ		
Res	Result			
Nan	me hotstart_velocity_U Unit	ts M/S		
	Evaluate Cano	cel		

Figure 90: BlueKenue Calculator method for extracting U component velocities as a new object hotstart_velocity_U

Now that the U and V components of velocity across the domain are extracted, new objects have

to be added to the **Selafin** file for interpolating. Add the current domain mesh as the **Source Mesh** and specify the **New Variable** as **VELOCITY U**, for example (Figure 91).

Add New SELAFIN Variable			
Source Mesh and Attribute Data			
Mesh	New Mesh 💌		
Attribu	te Name NodeType 💌		
New Variable Properties			
Name	VELOCITY U 👻		
Units	M/S		
Add copy of domain mesh as new VELOCITY U variable			
OK Cancel			

Figure 91: Add copies of the current domain mesh as new variables to the Selafin file

Once the **New Variables** are added, they will by default be set to **0** and appear flat as shown in Figure 92.



Figure 92: Unmapped new variable VELOCITY U, prior to Map Objects command

Select one of the **New Variables** of the **Selafin** file, and **go-to Tools ; Map Objects** and select the previous result file child-object that corresponds to the selected **New Variable**. This is shown with the child-object **VELOCITY U** in Figure 93.

Available Objects	23
TISSIPATION TO DISSIPATION TO VISCOSITY TO VISCOSITY TO FLOWRATE ALONG XY TO FLOCITY V TO VELOCITY V TO WATER DEPTH	
THEREE SUBFACE	E
<pre> hotstart_velocity_v </pre>	T
OK Cancel]

Figure 93: Map previous result file components to new components using Map Objects command



Figure 94: Successfully mapped VELOCITY U object

The variables required for a complete HOTSTART file to be created are VELOCITY U, VELOC-ITY V, WATER DEPTH, and FREE SURFACE ELEVATION.



Figure 95: Successful HOTSTART file generation should contain the above four child-objects

Once the HOTSTART file is created, it can be included in the parameters file as a **PREVIOUS COMPUTATION FILE**.

5.3 TELEMAC-2D Unsteady simulation

5.3.1 Unsteady boundary conditions

Inflow hydrographs are to be specified at the **Open boundary with specified Q** nodes for both upstream reaches. The downstream boundary will be represented using a rating curve assuming **Normal Flow** and a bed-slope, S_0 , equal to 0.001 m/m.



Figure 96: Inflow hydrographs applied to the Baxter River and Tule Creek

Т	Q(1)	Q(2)
S	m3/s	m3/s
0	566.337	11.327
7200	566.337	11.327
39000	3567.923	102.649
70800	566.337	11.327
76200	566.337	11.327

Table 4: Baxter tutorial inflow hydrograph values



Baxter River Outflow Rating Curve

Figure 97: Outflow rating curve for this tutuorial

Z(3)	Q(3)
m	m3/s
7.266	2.832
8.830	76.399
9.513	149.966
•	
•	•
•	•
16.106	3681.190
16.426	4247.527
17.057	5663.369

Table 5: Rating curve for downstream boundary condition

6 Post-processing utilizing BlueKenue

BlueKenue as a post-processing tool serves as an intuitive and capable platform for viewing and analyzing 2D- and 3D-results. Once the results are opened within the **Workspace**, the following are some guidelines for basic viewing of results.



Figure 98: Applying a vertical exaggeration to datasets with horizontal scales much larger than the vertical scale (e.g. BOTTOM, FREE SURFACE, etc.) helps to visualize geometric features



Figure 99: Changing the **Style** and opacity of the object is easily performed by double-clicking or right-clicking the object



Figure 100: After initializing the **Animate** item under properties and select **Apply**, then the playback controls initialize to view the dataset

6.1 Flood inundation view settings



Figure 101: To view only the Flood Inundation depths, apply **Clip Contours** and make sure that **Style** is set to **Filled Contours**



Figure 102: Resulting view from Figure 101 settings with Animate enabled

6.2 Animation of Flood-wave propagation

Animations of your TELEMAC-2D results are easily exported as video files using BlueKenue. For example, the following steps will illustrate how to export an animation of the flood-wave propagation to a video file.

- 1. Create a new **3D View**
- 2. Drag and drop TELEMAC-2D result objects to animate

- 3. Enable animation on the T2D object and select desired Scale, viewing angle, etc.
- 4. Under properties of the 3D View object, select the Recording tab
- 5. Choose destination filename, number of frames, playback frame-rate, encoding method, etc.
- 6. Create the animation by selecting the record icon,

6.3 Outflow hydrographs using MATLAB

Using control sections defined within the computational domain, it is possible to calculate and track discharge at specified locations. The control sections can be specified by either coordinate pairs, or by start-and-end mesh nodes. The control sections incorporated in this tutorial are shown in Figure 103 below. An example of outflow hydrographs resulting from using control sections is shown in Figure 104. The simple MATLAB plotting routine is included in the **ctrl_section_results** directory as **plot_ctrl_section_results.m**.



Figure 103: Control sections within the Baxter River domain



Figure 104: Outflow hydrographs resulting from unsteady TELEMAC-2D simulation

6.4 TELEMAC-2D vs HEC-GeoRAS Flood Inundation

Comparison between TELEMAC-2D and HEC-GeoRAS flood inundation can be achieved through importing the HEC-GeoRAS inundation map to BlueKenue. The process for exporting an ArcView Shape File from ArcGIS is detailed in Section 2.3. Note that the exported Shape File will need to be converted using the conversion in Section 2.5.

A quick comparison between the TELEMAC-2D and HEC-GeoRAS flood inundation extents shows loose agreement, with TELEMAC-2D showing greater inundation extents for the same flow event. Using the **Animate** playback controls, the TELEMAC-2D max inundation occurs around time 15:00 (HH:MM).



Figure 105: Example comparison of TELEMAC-2D and HEC-GeoRAS inundation extents

Appendix A Create Simple Meshes in Blue Kenue

This section gives brief step-by-step instructions on how to create a simple mesh in BlueKenue for use with the TELEMAC hydrodynamics suite. The content and figures herein are adapted from the work of **Stephen Kwan, MCS, Ph.D**. You can find the original document link at his website: http://river2dm.wordpress.com/telemac2d/

1.	Load xyz data.	
2.	Create closed line (Figure A106)	Page 90
3.	Highlight the xyz data in DataItems	
4.	Select Files>New>T3 Mesh generator (Figure A107)	Page 90
5.	Give length of element side and press OK	
6.	Drag the new ClosedLine into Outline of newT3Mesh (Figure A108)	Page 91
7.	Double click on new T3 Mesh and press run (Figures A109 and A110)	Pages 91 & 92
8.	Select File>New 2D Interpolator (Figure A111)	Page 92
9.	Drag subset into newInterpolator2D (Figure A112)	Page 93
10.	Highlight New Mesh (NodeType)	
11.	Select Tools>Map object	
12.	Choose New2Dinterpolator (Figure A113)	Page 93
13.	Give name, put M for units (Figure A114)	Page 94
14.	Mesh shown in (Figure A115)	Page 94

A.1 Supporting figures for simple mesh tutorial

The figures below correspond to the above list of steps required to create a simple mesh in BlueKenue.



Figure A106: Create closed line for the exterior boundary of the mesh.



Figure A107: Create New T3 Mesh



Figure A108: Drag newClosedLine into newT3Mesh Outline



Figure A109: Select edge length for elements and run.



Figure A110: The new T3 mesh!



Figure A111: Create New 2D Interpolator.



Figure A112: Drag subset of xyz data into newInterpalator2D.



Figure A113: Choose Tools>Map Object and select newInterpolator2D.



Figure A114: Give a new name and choose put M for units.



Figure A115: The new interpolated mesh.

Appendix B Self-installation of TELEMAC hydrodynamics suite

TELEMAC-MACARET Python Installation Notes Link

B.1 Pre-requisite programs for TELEMAC installation

These brief notes detail requirements for source code compilation using a Python/gfortran/windows configuration. As stated above in Section 1, the required programs to be installed include:

- - Python 2.7.3 or 2.7.5
- - gfortran (gcc) Compiler
- - SVN Tortoise subversion program

B.2 TELEMAC source code checkout using Tortoise SVN

Checking out the source code of TELEMAC is easily done using Tortoise SVN software. Create a new folder named 'opentelemac' in the desired directory (e.g. c:\... for this example), right-click the folder and select 'SVN Checkout...'. Next, you'll enter the SVN address below and then enter the username and password when prompted. Note: if there is a new version of TELEMAC available, simply change the SVN address per www.opentelemac.org.

```
URL of Repository (as of November 1st, 2013):
http://svn.opentelemac.org/svn/opentelemac/tags/v6p3r1/
username: ot-svn-public
password: telemac1*
```

In order for TELEMAC to communicate with the required programs from the DOS command prompt, the user/system environmental variables must direct the system to each application/required file. You can get to the environmental variable dialogue through the Control- Panel, typing 'environmental variables' into the search bar, or from the 'Start' menu. Once there, append the environmental variables to include the ones stated below. As time goes on, each system will be slightly different, however the general idea is as follows:

- 1. grant access to Python, scripts¿python27 folder, and gcc bin folder within PATH
- 2. include EQ_LIBRARY_PATH for gfortran (gcc) to reference
- 3. include SYSTELCFG path to the TELEMAC configuration file

B.3 Environment variables for running/compiling TELEMAC

```
PATH:
c:\gcc\bin;c:\Python27\Lib;c\Python27; ...
c:\opentelemac\v6p3\scripts\python27;
EQ_LIBRARY_PATH:
c:\gcc\x86_64-w64-mingw32\lib
SYSTELCFG:
c:\opentelemac\v6p3\configs\systel.cfg
```

See www.opentelemac.org for additional installation instructions if this isn't clear: http://www.opentelemac.org/... search for Python installation instructions.

B.4 Testing your PATH variables

To test that your path variables are working, open a DOS command prompt session and type the following:

- 'gfortran -v'
- 'python -v'

If your path variables are set-up correctly, the version information will print to your screen. (note: to exit python-mode type 'exit()' or 'CTRL+Z+Enter')

B.5 Compiling TELEMAC

Once the path variables are working and the 'systel.cfg' file is appropriately set-up (or copied from this installation repository) for your system specifics, open the command prompt. Change directory to the pytel folder directory (**not necessary**),

```
(e.g. cd c:\opentelemac\v6p3\scripts\python27) and execute the command:
```

```
'compileTELEMAC.py'
```

If all is set-up correctly, then the TELEMAC system will soon be installed, compiled, and ready for use on your machine.

As mentioned before, if you're having any trouble installing TELEMAC with these instructions as well as the provided guide- lines of opentelemac.org, feel free to contact me.

Cheers!

Christopher Gifford-Miears cgiffordmiears@gmail.com