CE 5362 Surface Water Modeling
Project 3

Introduction and Purpose

SToRM is a component of the USGS Multi-Dimensional Surface Water Modeling System (McDonald and others, 2012) SToRM is one of a generation of recently available 2–D hydrodynamic models available without charge or for low cost. In the next several exercises you will apply SToRM to several test situations where actual measurements are available in part to test the tool itself at physical scales that differ by about an order of magnitude, and to develop the skill set necessary to use the tool for an arbitrary situation.

Problem Statement

1. Replicate the Green River example presented in class. As output produce a vector plot of the velocity field and a streamline plot. The purpose is to convince yourself you can operate the software.
2. Simulate a rectangular channel (research flume) as depicted in Figure 1, The flume elevation information is contained in the file `rf-plain.tpo`.

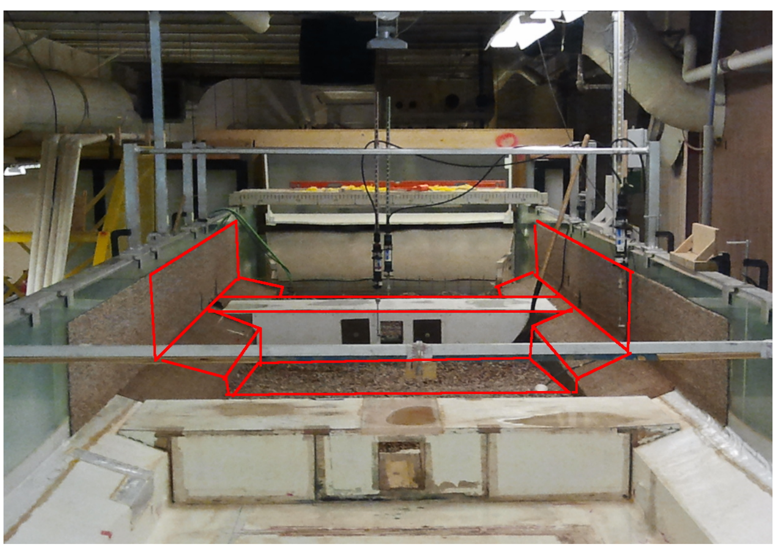

Figure 1: Research flume looking upstream. Red outline is approximate geometry of model application. Ignore the culvert models, assume flume is a simple rectangle

Use the following additional values to parameterize the model

- Initial flow depth $h = 0.30\ m$
- Upstream inflow boundary condition $Q = 0.1\ \frac{m^3}{s}$
- Downstream outflow boundary condition, $H = 0.3\ m$
- Initial X-component of velocity, $U = 0.2\ m/s$
- Initial Y-component of velocity, $V = 0.0\ m/s$
- Manning's roughness coefficient $n = 0.04$

Produce simulation outputs that show the velocity distribution (vector plots), and the streamline plot, similar to Figure 2 and Figure 3. Discuss any steps you had to take to reflect the geometry correctly for the numerical model to function correctly.

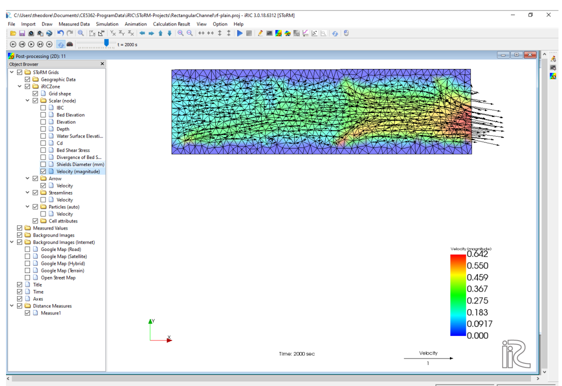

Figure 2: Vector plot at simulation time $t = 2000\ sec$, research flume, no obstruction

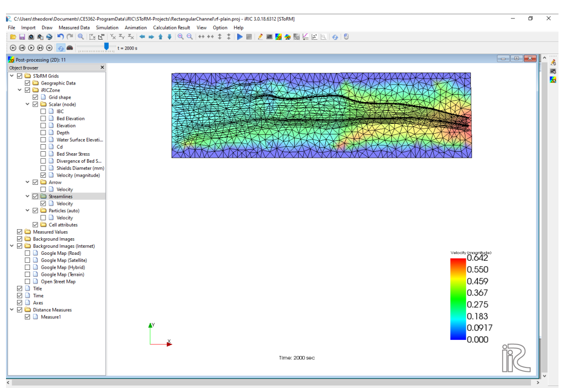

Figure 3: Streamline plot at simulation time $t = 2000\ sec$, research flume, no obstruction

3. Simulate a rectangular channel (research flume) as depicted in Figure 4, which includes the step with a cut in the middle to represent the small culverts

   The flume elevation information is contained in the file `rf-cutstep.tpo`.

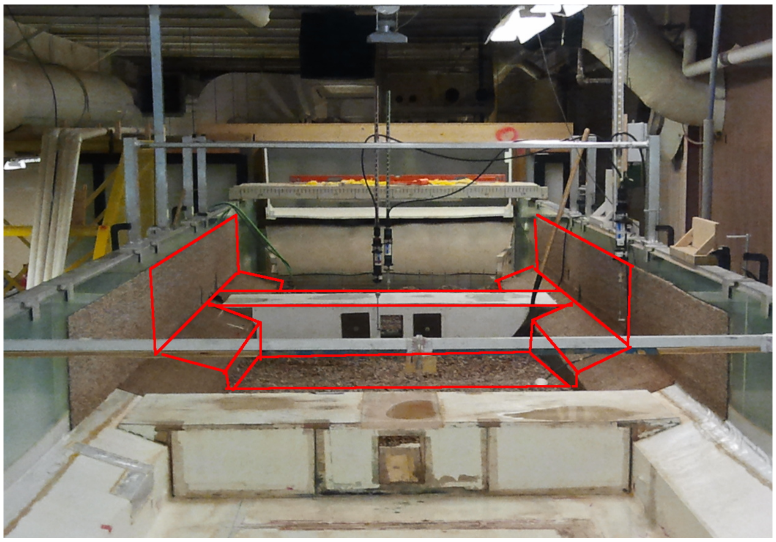

Figure 4: Research flume looking upstream. Red outline is approximate geometry of model application.

   Use the following additional values to parameterize the model

   - Initial flow depth $h = 0.30\ m$
   - Upstream inflow boundary condition $Q = 0.1\ \frac{m^3}{s}$
   - Downstream outflow boundary condition, $H = 0.3\ m$
   - Initial X-component of velocity, $U = 0.2\ m/s$
   - Initial Y-component of velocity, $V = 0.0\ m/s$
   - Manning's roughness coefficient $n = 0.04$

   Produce simulation outputs that show the velocity distribution (vector plots), and the streamline plot. Discuss any steps you had to take to reflect the geometry correctly for the numerical model to function correctly.

   Compare the results to the previous model, is the effect of the step apparent in the velocity or streamline diagram?

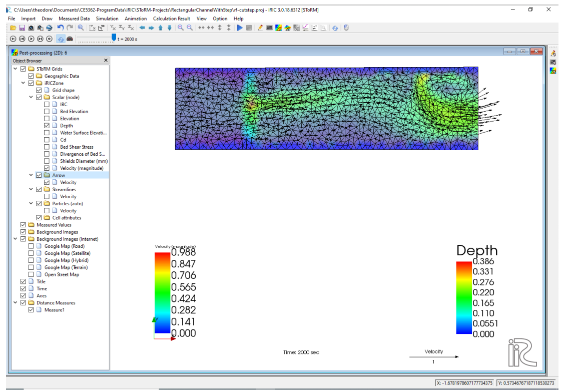

Figure 5: Vector plot at simulation time $t = 2000$ sec, research flume, with obstruction

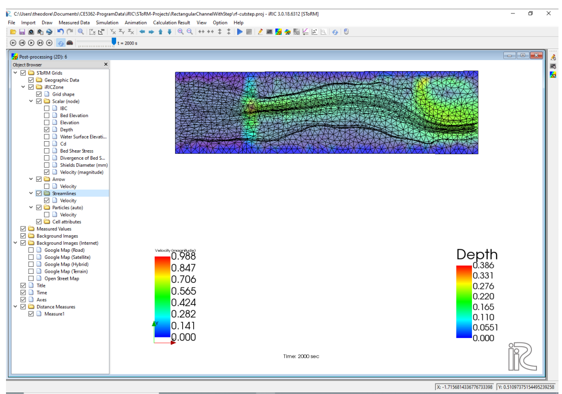

Figure 6: Streamline plot at simulation time $t = 2000$ sec, research flume, with obstruction

4. (Advanced) Edit the `.tpo` file to move the cut to one side of the flume. Does this change the velocity pattern. Produce output plots.

References

USGS Geomorphology Laboratory (2011). System for Transport and River Modeling. `http://wwwbrr.cr.usgs.gov/projects/GEOMORPH_Lab/project-SToRM.html` Webpage last accessed, 12 Jan 2012.

McDonald, R.R., Nelson, J.M., and Bennett, J.P., (2012). *in press*. Multi-dimensional surface-water modeling system user's guide: U.S. Geological Survey Techniques and Methods, 6-B2, 136 p.