Workshop

ADCIRC—STWAVE Steering

7.1 Introduction

This workshop teaches how to use tools in the Steering Module to couple the ADCIRC and STWAVE models. The Steering Module makes it easier for the user to take advantage of optional input/output data sharing between models by automating repetitive user tasks and monitoring model runs. This lesson assumes that the user is familiar with the ADCIRC and STWAVE models.

7.2 Importing the ADCIRC Data

We will be working with an ADCIRC grid of Grays Harbor, Washington and the surrounding coastline. The grid and control files have already been created so they just need to be read into SMS. Note that all of the model files are located in a single directory. When using the Steering Module, it is necessary to place all of the input files and model executables in one directory. To read the ADCIRC files:

1. Select *File* | *Open*.

2. Path to Lesson07 directory and select the file “fort.14”. This is the ADCIRC grid file.

3. Repeat for the “fort.15” file, which is the ADCIRC control file.

7.2.1 Coordinate Conversion

Because the ADCIRC and STWAVE files are in different coordinate systems, it will be necessary to perform a coordinate conversion before continuing. The ADCIRC files that we just read in are in Geographic coordinates while the STWAVE files that we will read in are in State Plane. We will convert ADCIRC to State Plane in SMS but all of the ADCIRC files will remain in Geographic coordinates. This is an important point to understand when using the Steering Module. We want to run the ADCIRC model using Geographic coordinates and the STWAVE model with State Plane coordinates. Since the models will be sharing data, there must be a common coordinate system for data interpolation. To solve this problem, we will convert the ADCIRC mesh to the same coordinate system as the STWAVE files (State Plane.) To perform the conversion:

1. Select *Edit* | *Coordinate Conversions…*
2. Press the *Current Options…* button.
3. In the *Coordinates* dialog, select Geographic NAD 83 (US) for the *Horizontal System*.
4. Select Local for the *Vertical System* and Meters for the *Units*.
5. Press the *OK* button to return to the *Coordinate Conversion* dialog.
6. In the *Convert to* section of the dialog, select State Plane NAD 83 (US) for the *Horizontal System* and Washington South – 4602 for the *St. Plane Zone*.
7. Set the *Units* to Meters.
8. Set the *Vertical System* to Local and the *Units* to Meters.
9. Press the *Convert* button.

The ADICRC mesh is now in State Plane coordinates. Note that the ADCIRC mesh appears in State Plane coordinates inside of SMS but the files that the ADCIRC model will reference remain in Geographic coordinates.

7.3 Importing the STWAVE Data

As with the ADICRC data, all of the STWAVE files that we need for this workshop have already been generated and are located in the WORKSHOP directory. To read the STWAVE data into SMS:

1. Select File | Open.
2. Path to the Lesson07 directory and select the file, “grays_harbor.sim”.

The STWAVE grid may be difficult to see since its domain is much smaller than the ADCIRC mesh. The figure below is a zoomed-in picture of the inlet at Grays Harbor.

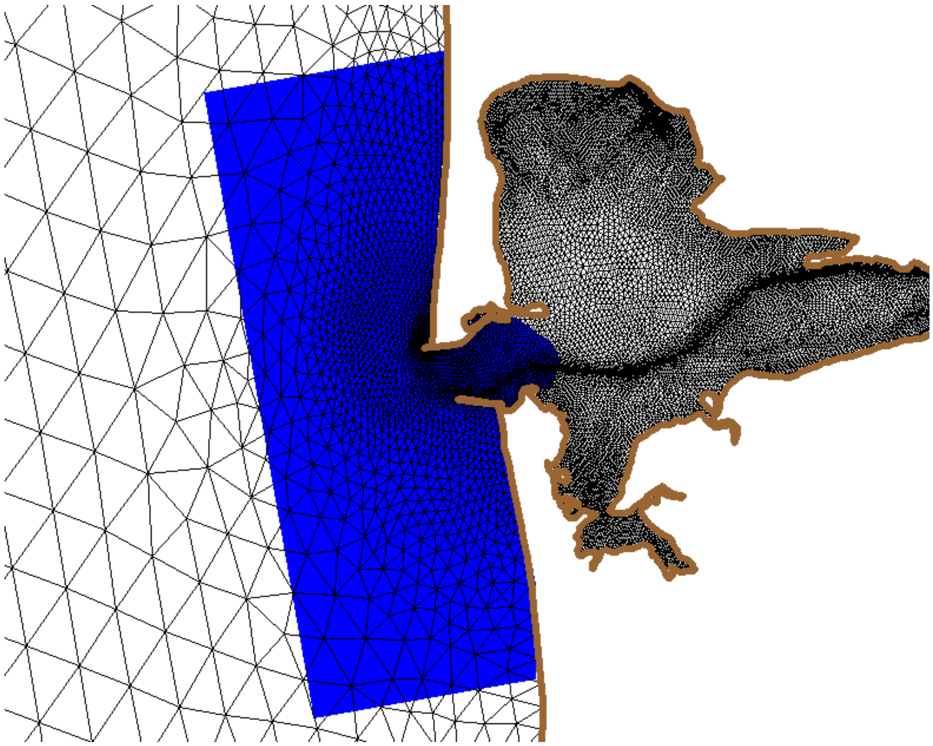

Figure 7-1STWAVE Grid on top of ADCIRC Mesh

7.4 Running the Steering Module

7.4.1 The Main Steering Module Dialog

With the ADCIRC and STWAVE model files read into SMS, we are ready to use the Steering module. For ADCIRC—STWAVE coupling, ADCIRC current field may be passed to STWAVE and STWAVE wave radiation stresses may be passed to ADCIRC. Data sharing and timing options are specified the main Steering Module dialog. To invoke this dialog:

1. Switch to the Mesh Module.

2. Select *Data | Steering Module*.

3. In the *Steering Module* dialog, be sure that the ADCIRC⇔STWAVE option is selected for *Model steering to perform*.

The Steering Module dialog is split into several sections. In each of the sections, there are controls that can be set to tell the Steering Module what data to pass and how often the data will be passed. Before setting up the Steering Module for our

Grays Harbor testcase, we will take some time here to describe more about the steering process.

There are three available steering types for ADCIRC⇔STWAVE steering: one-way ADCIRC→STWAVE, one-way STWAVE→ADCIRC, and two-way coupling. The types are selected by checking the appropriate boxes in the steering dialog. For one-way ADCIRC→STWAVE steering, check the box marked Current Field. For one-way STWAVE→ADCIRC, Wave data would be checked. For two-way steering, both the Current field and Wave data boxes would be checked.

The field Run STWAVE every __ hours, corresponds to how often STWAVE will be run and when radiation stresses will be applied to the ADCIRC simulation and/or how often ADCIRC currents will be applied to the STWAVE simulation. The box labeled, Save Last fort.15, sets the option of whether to keep the last ADCIRC control file or replace it with the original control file. This option allows the user to see the latest control file should ADCIRC crash or end prematurely. The original file can still be retrieved; it is found in the model directory and is named origfort.15. Since the STWAVE domain is smaller than the ADCIRC domain, there are two options for extrapolating the STWAVE solution to the ADCIRC mesh. When Extrapolation is Set to zero, radiation stress values for all elements in the ADCIRC mesh outside of the STWAVE grid domain will be set to zero. For Extrapolation set to Extrapolate out, the steering module utilizes a sigmoidal interpolation routine to set radiation stress values for the ADICRC mesh. For ADCIRC→STWAVE, the steering module linearly interpolates from the ADCIRC mesh to the STWAVE grid. The ADCIRC location and STWAVE location buttons specify location of the model executables.

The Total Simulation time is the simulation time specified in the ADCIRC unit 15 control file. The ADCIRC Time Control… button is a link to the Time Control dialog.

7.4.2 Steering Types Explained

ONE-WAY ADCIRC→STWAVE STEERING: For one-way ADCIRC→ STWAVE steering, current data generated by ADCIRC are saved as input to the STWAVE simulation. The ADCIRC simulation is broken down into smaller simulation times corresponding to the STWAVE run interval. For example, if the total ADCIRC simulation time is 1 day and STWAVE is set to run every 2 hours, ADCIRC will be run 12 times with each run performing calculations on two hours of time steps.

Using the example of a 1-day ADICRC simulation and running STWAVE every 2 hours, the steering process begins with an ADCIRC run from zero to 2 hours. When the ADIRC run is completed, the steering module reads the ADCIRC unit 64 file and interpolates the values from the velocity solution to the STWAVE grid. The steering module then saves the necessary files to run STWAVE. When the STWAVE run is completed, ADCIRC is run again with the simulation time set from zero to 4 hours.

Since a hotstart file was generated at the end of the previous ADCIRC run, the current ADCIRC run performs calculation at timesteps from hr 2 to hr 4. This process of alternately running ADCIRC and STWAVE is continued until 1 day has been reached.

ONE-WAY STWAVE→ADCIRC STEERING: For one-way STWAVE→ ADCIRC steering, wave radiation stresses generated by STWAVE are saved as input to the ADCIRC simulation. Again using the example of a 1-day ADICRC simulation and running STWAVE every 2 hours, the steering process begins with an STWAVE run with input spectra corresponding to time 0. When STWAVE is completed, the model is run again with spectra for time 2 hr. The steering module interpolates the radiation stress solution from STWAVE to the ADCIRC mesh for both time 0 hr and time 2 hr. The ADCIRC model is then run with the wave radiation stress input file (unit 23) containing data for time 0 hr and time 2 hr. When ADCIRC is completed, the STWAVE model is run again with input spectra at time 4 hr. The next ADCIRC run is from time 0 hr to time 4 hr with the unit 23 file containing values for time 2 hr and time 4 hr. This process of alternately running STWAVE and ADCIRC is continued until 1 day has been reached.

TWO-WAY ADCIRC⇔STWAVE STEERING: In two-way ADCIRC⇔STWAVE steering, both ADCIRC current data and STWAVE wave data are shared between the models. The process is the same as one-way STWAVE→ADCIRC steering except that ADCIRC currents are interpolated to the STWAVE grid at the end of each ADCIRC run.

7.4.3 Applying the Steering Module to the Grays Harbor Testcase

For our Grays Harbor testcase, we will be performing one-way STWAVE→ ADCIRC steering and passing wave radiation stresses to ADCIRC every 2 hours. Notice that the total ADCIRC simulation time is 3 days. Since ADCIRC will be run every 2 hours, the STWAVE input files contain 37 spectra (t=0 hrs, t=2 hrs, … t=72 hrs).

1. In the *Steering Module* dialog, enter 2.0 in the edit field to run STWAVE every 2 hours.

2. Check the *Wave data* option.

3. For *Extrapolation*, select Extrapolate out.

4. Enter 1500 for the *Extrapolation distance*.

5. Press the *ADCIRC location* button. In the *Run Model* dialog, select the file “adcirc.exe” in the WORKSHOP directory and press *OK*.

6. Press the *STWAVE location* button. In the *Run Model* dialog, select the file “stwave.exe” in the WORKSHOP directory and press *OK*.

7. Press the *START* button to begin the steering process.

SMS begins steering the models and a dialog similar to the one shown below appears. This dialog gives real-time updates of the progress of the Steering Module. The left plot gives number of iterations vs. timestep for the current ADCIRC run. The plot on the right side of the dialog shows wave height vs. column for the current STWAVE run. The large window contains the current model's output to the screen.

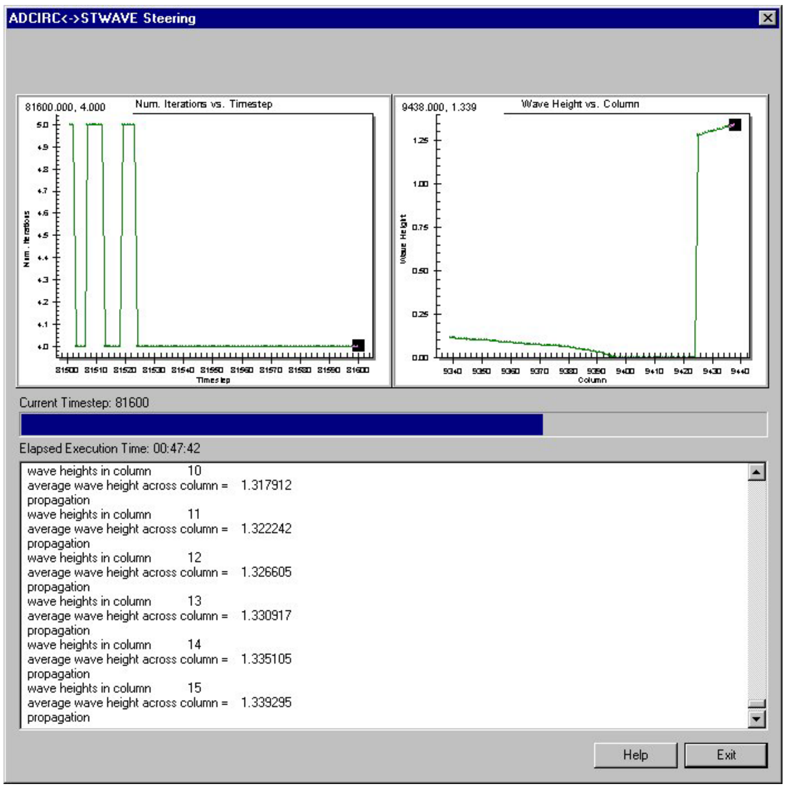

Figure 7-2 Steering Dialog – shows progress of steering run.

7.5 Steering Module Output

While the Steering Module is working, a status file named SteeringStatus.txt is created. This file is located in the same directory as the model input files and contains a log of the Steering Module events. Once the steering process has completed, the original input files for ADCIRC are restored. ADCIRC solution files, such as the unit 63 and unit 64 files, are consolidated and contain data for the entire 3-day simulation. All STWAVE simulation files that were created by the Steering Module are retained and are named 1swsteer.sim for first simulation, 2swsteer.sim for the second simulation, etc. For this workshop, 1swsteer.sim corresponds to time 0 hr, 2swsteer.sim corresponds to time 2 hr, etc.

Because of the enormous computation demand we are placing on the ADCIRC and STWAVE models, the Steering Module will take a long time to finish our testcase of

Grays Harbor. Therefore, we will abort the steering process and view the results of a steering run that has already been completed. For this particular testcase, the Steering Module would run for over 3 days on a Pentium 4 processor. The run time would be about 1.5 days if you are using an AMD processor. To abort the steering process:

1. Press the button labeled *Abort* or *Exit*.

2. Wait several seconds while SMS shuts down the Steering Module.

7.5.1 Setting Up Film Loops

To view the results of our Grays Harbor testcase, we create several filmloops. We will first open the results of the ADCIRC model. To open the ADCIRC output files:

1. Select *File* | *Open*.

2. Path to *workshop/Adcirc_Only* directory and select the file "fort.64". This is the ADCIRC velocity time series solution.

Now that we have the files open, we will manipulate the display so that the filmloops will be easily viewed.

1. Zoom in on the inlet of Grays Harbor. (Grays Harbor is the top of the three inlets).

2. Select *Display* | *Display Options*. Under the 2D Mesh tab, turn off the *Nodes* and *Elements* toggles, and turn on the *Contours* and *Vectors* toggles.

3. Click on the Vectors tab.

4. Under the Vectors Tab, change the shaft length to *Scale length to magnitude*. Make sure the Scaling Ratio is set to 15.0, and click OK.

5. Resize the main SMS window to about one-fourth the size of the screen. We do this so that we may view multiple filmloops side by side.

Once these five steps are completed, the SMS window should look similar to the following:

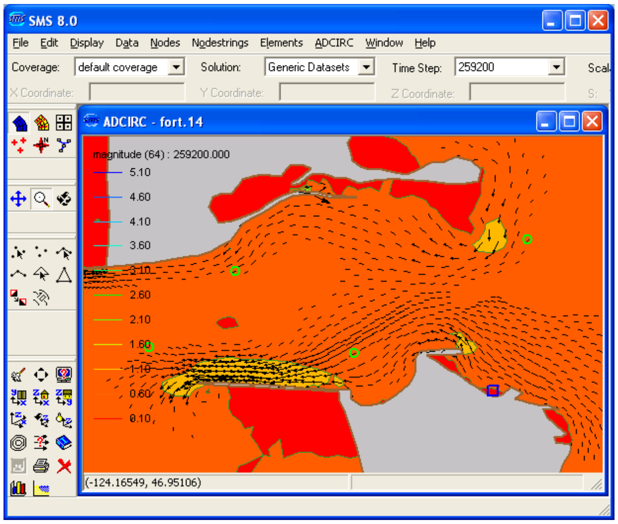

Figure 7-3Solution data in SMS.

We are now ready to create a filmloop to view the results of the ADCIRC model. To create a filmloop:

1. Select *Data* | *Film Loop…*
2. Make sure *Create New Filmloop* is selected and click the file icon.
3. For the filename, type AdcircEbb. Click the Save button.
4. In the *Filmloop* dialog, click the next button.
5. Turn on both the *Scalar Data Set* and the *Vector Data Set* toggles.
6. To set the filmloop to show the timesteps surrounding the peak ebb, set the following timesteps for the simulation: *Run simulation from time: 210,600 to: 228,600.* Set the *Number of Frames* to 25.
7. Click *Next*, and then click *Finish*.

As the filmloop is being created, each frame will be displayed. After all the frames have been made, a separate window will open with the completed filmloop. Once you are done viewing the filmloop, close its window. We will now create a similar filmloop showing the timesteps surrounding a peak flood. To do so:

1. Select *Data* | *Film Loop…*
2. Make sure *Create New Filmloop* is selected and click the file icon.
3. For the filename, type AdcircFlood. Click the Save button.
4. In the filmloop dialog, click the Next button.
5. Make sure the *Scalar Data Set* and the *Vector Data Set* toggles are on.

6. To set the filmloop to show the timesteps surrounding the peak flood, set the following timesteps for the simulation: *Run simulation from time: 240,300 to: 258,300.* Set the *Number of Frames* to 25.

7. Click *Next*, and then click *Finish*.

Again, as the filmloop is being created, each frame will be displayed. After all the frames have been made, a separate window will open with the completed filmloop. Once you are done viewing the filmloop, close its window.

To see the differences in ADCIRC solutions with and without the coupling, we now want to create similar filmloops of the Steering Module results. After these filmloops are created, we will compare them with the filmloops we have already generated.

To open the results calculated with the Steering Module:

1. Select *File* | *Delete All*. When the prompt appears, click Yes to delete the data currently in the window.

2. Select *File* | *Open*.

3. Path to the *workshop/STAD_ONE* directory and select the file "fort.14". This is the grid file.

4. Repeat for the "fort.15" file, which is the control file.

5. Repeat for the "fort.64" file.

6. Now that we have the files open, we will manipulate the display so that the filmloops will be easily viewed.

6. Zoom in on the same part of the inlet of Grays Harbor that you zoomed in on just before making the first filmloop. (Again, Grays Harbor is the top of the three inlets).

7. Select *Display* | *Display Options*. Under the 2D Mesh tab, turn off the *Nodes* and *Elements* toggles, and turn on the *Contours* and *Vectors* toggles.

8. Click on the *Vectors* tab.

9. Under the *Vectors* Tab, change the shaft length to *Scale length to magnitude*. Make sure the *Scaling Ratio* is set to 15.0, and click *OK*.

10. Keep the size of the main SMS window about one-fourth the size of the screen so that we may view multiple filmloops side by side.

We are now ready to create a filmloop to view the results of the Steering Module analysis. To create a filmloop:

1. Select *Data | Film Loop…*

2. Make sure *Create New Filmloop* is selected and click the file icon.

3. For the filename, type StAdEbb. Click the Save button.

4. In the filmloop dialog, click the Next button.

5. Make sure the *Scalar Data Set* and the *Vector Data Set* toggles are on.

6. To set the filmloop to show the timesteps surrounding the peak ebb, set the following timesteps for the simulation: *Run simulation from time: 210,600 to: 228,600.* Set the *Number of Frames* to 25.

7. Click *Next*, and then click *Finish*.

Once you are done viewing the filmloop, close its window. Again, we will create a similar filmloop showing the timesteps surrounding a peak flood. To do so:

1. Select *Data | Film Loop…*

2. Make sure *Create New Filmloop* is selected and click the file icon.

3. For the filename, type StAdFlood. Click the Save button.

4. In the *Filmloop* dialog, click the next button.

5. Make sure the *Scalar Data Set* and the *Vector Data Set* toggles are on.

6. To set the filmloop to show the timesteps surrounding the peak flood, set the following timesteps for the simulation: *Run simulation from time: 240,300 to: 258,300.* Set the *Number of Frames* to 25.

7. Click *Next*, and then click *Finish*.

Once you are done viewing the filmloop, close the filmloop window.

7.5.2 Merging Film Loops

Now that we have filmloops of the peak ebb and the peak flood from both the ADCIRC only and one-way coupling results, we will put these filmloop side by side to compare them. To do this, we will first use a tool that links two filmloops together, and then view the two filmloops side by side. To link the filmloops together:

1. With Windows Explorer, path to the WORKSHOP directory and open the executable “mergeavi.exe”. This is the utility that merges two or more filmloops (.avi files) together.

2. In the Mergeavi window, select *Panel|Setup.*

3. In the *Layout Panels* dialog, set the *Number of panels* to 2 and turn on the *DrawBorder* toggle. To merge the "peak ebb" filmloops from ADCIRC only solution and the one-way coupling, we need to open those files. In the files section of the *Layout Panels* dialog, click the Browse button next to 1. Select *AdcircEbb* and click the OK button. Once you return to the Layout Panels dialog, click the Browse button next to 2. Select *StAdEbb* and click the OK button. Click the OK button to close the Layout Panels dialog.

4. In the Mergeavi window, select *File | Save…*

5. Type *MergedEbb.avi* as the filename and click Save. (You must type the entire file name, including the .avi extension). A new filmloop called MergedEbb now contains both of the peak ebb filmloops. Close the Mergeavi window, and return to the SMS window.

6. In SMS, select *Data | Filmloop…*

7. In the *Filmloop Setup* dialog, select *Open Existing Filmloop* and click on the file button.

8. Select *MergedEbb.avi* and click the *Open* button.

9. In the *Filmloop Setup* dialog, click the *Next* button, and the merged filmloops will be displayed.

The filmloops of the solutions with and without coupling for a peak ebb are played side by side. In the filmloop that is currently playing, the left frame represents solution without model coupling. The right frame shows a visual representation of data calculated using the Steering Module, which takes advantage of input/output data sharing between models. The vectors in both frames represent the velocity of the waves, as calculated by ADCIRC. The coloring and vector length in both frames represent the magnitude of the velocity vectors.

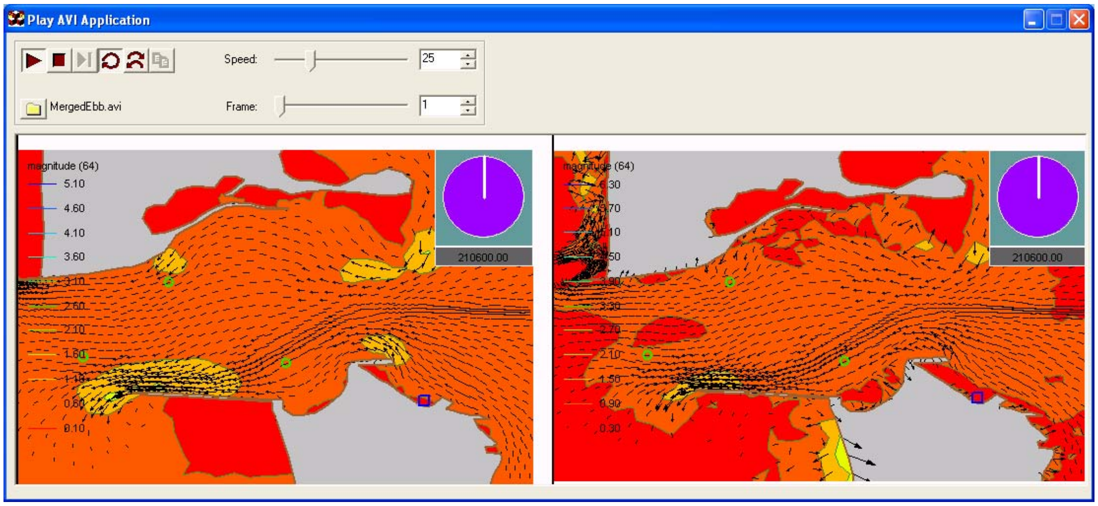

Figure 7-4Merged filmloops of solutions with and without waves around peak ebb.

Now that we have spent some time looking at the visual representation of the results of a peak ebb, we will look at the results of a peak flood. We will follow the same general steps that we followed for the peak ebb:

1. With Windows Explorer, path to the WORKSHOP directory and open the executable "mergeavi.exe".

2. In the Mergeavi window, select *Panel|Setup.*

3. In the *Layout Panels* dialog, set the *Number of panels* to 2 and turn on the *DrawBorder* toggle. To merge the "peak flood" filmloops from adcirc and from the Steering Module, we need to open those files. In the files section of the Layout Panels dialog, click the Browse button next to 1. Select *AdcircFlood* and click the OK button. Once you return to the Layout Panels dialog, click the Browse button next to 2. Select *StAdFlood* and click the OK button. Click the OK button to close the Layout Panels dialog.

4. In the Mergeavi window, select *File|Save…*

5. Type *MergedFlood.avi* as the filename and click Save. (You must type the entire file name, including the .avi extension). A new filmloop called MergedFlood now contains both of the peak flood filmloops. Close the Mergeavi window, and return to the SMS window.

6. In SMS, select *Data|Filmloop…*

7. In the Filmloop Setup dialog, select *Open Existing Filmloop* and click on the file button.

8. Select *MergedFlood.avi* and click the Open button.

9. In the *Film Loop Setup* dialog, click the *Next* button, and the merged filmloops will be displayed.

The two filmloops (with and without coupling) for a peak flood are played side by side.

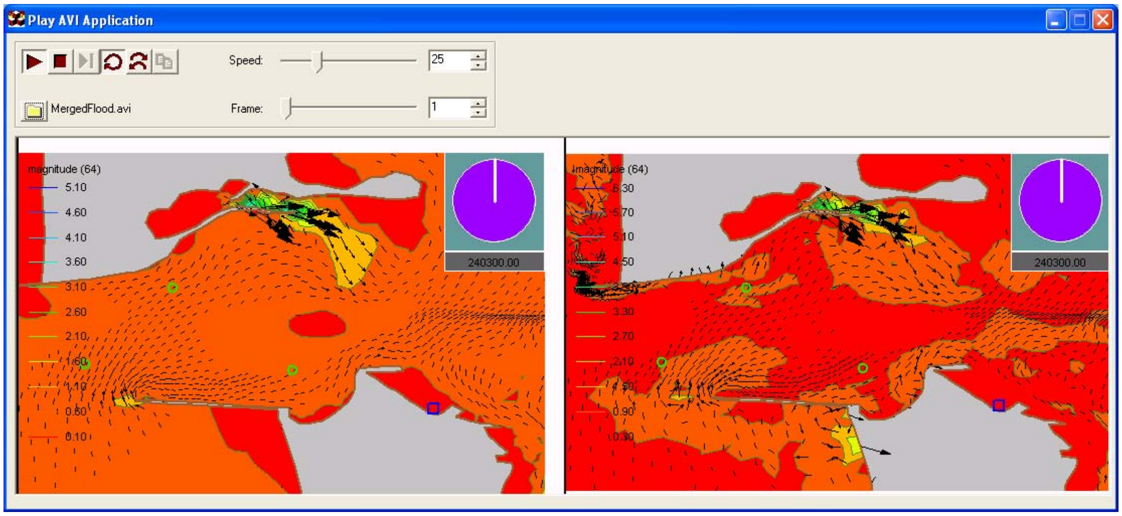

Figure 7-5Merged Filmloops of solutions with and without waves around peak flood.

7.5.3 Organizing Data Sets into Solutions

Another way that the results from the different models may be analyzed is through the use of the Data Calculator. We must first organize the data so that the calculations and results will be more easily understood:

1. Select *File* | *Open*.
2. Path to the *workshop* | *STAD_ONE* directory and select the “fort.63” file, which contains the time series water surface elevation solution from the ADCIRC model.
3. Select *Data* | *Data Browser…*
4. In the *Data Browser* dialog, click the *New* button.
5. Enter StAd_One_Way as the new solution name and click *OK*.
6. Highlight the *Generic Datasets* solution. Select *magnitude (64)* and click the *Move to Solution…* button directly under it.
7. In the *Import Data Set* dialog, select *StAd_One_Way* from the drop-down list and click *OK*.
8. Highlight the *Generic Datasets* solution. Select *water surface elevation (63)* and click the *Move to Solution…* button directly under it.

9. In the *Import Data Set* dialog, select *StAd_One_Way* from the drop-down list and click OK.

10. Repeat steps eight and nine, this time moving the *velocity (64)* Vector Set to the *StAd_One_Way* solution.

The previous steps organized the solution data from the one-way coupling so that the data will be easier to manipulate in the Data Calculator. We will now organize the ADCIRC solution data without coupling.

1. Select *File* | *Open*.

2. Path to the *workshop/ADCIRC_Only* directory and select the “fort.64” file.

3. Repeat step 2, opening the “fort.63” file.

4. Select *Data* | *Data Browser…*

5. In the *Data Browser* dialog, click the *New* button.

6. Enter *Adcirc_Only* as the new solution name and click *OK*.

7. Highlight the *Generic Datasets* solution. Select *magnitude (64)* and click the *Move to Solution…* button directly under it.

8. In the *Import Data Set* dialog, select *Adcirc_Only* from the drop-down list and click *OK*.

9. Highlight the *Generic Datasets* solution again. Select *water surface elevation (63)* and click the *Move to Solution…* button directly under it.

10. In the *Import Data Set* dialog, select *Adcirc_Only* from the drop-down list and click OK.

11. Repeat steps nine and ten, this time moving the *velocity (64)* Vector Set to the *Adcirc_Only* solution.

Now that the data is organized, we need to change the display options so the results from the Data Calculator will be seen more easily. The main change that we will be making to the display is the display of contours. When we use the Data Calculator, we will be calculating the differences in velocity magnitude and water surface elevation between coupled and ADCIRC only solution sets. To easily see the differences, we will be using a custom contour color ramp of red, white, and blue. Red contours will indicate areas where the computed solution with coupling is higher than the solution without coupling. Blue contours will indicate areas where the computed solution with coupling is lower than the solution without coupling. If the solutions are identical, the color of the contour will be white.

1. Select *Display* | *Display Options*

2. Click on the *Contours* tab.

3. Set the *Contour Method* to Color Fill. Click the *Color Options* button.

4. In the *Color Options* dialog, select *User Defined*, and click the *Load Palettes…* button. Path to the Workshop directory. Select rwbpal.pal and click Open. Click the *OK* button to close the *Color* dialog.

5. In the Display Options dialog, under the *Contours* tab, make sure the *Specify a range* toggle is on. For the *Min value* enter –1.0. For the *Max value* enter 1.0. Make sure the *Fill Above* and *Fill Below* toggles are on. Select the 2D Mesh tab. Turn off the *Vectors* toggle and click OK.

7.5.4 Using the Data Calculator

We are now prepared to use the Data Calculator. With the Data Calculator, we will compute the differences in the water surface elevation between the two solutions (with and without coupling). We will also compute the differences in velocity magnitude. To perform these calculations with the Data Calculator:

1. Select *Data* | *Data Calculator…*

2. In the *Data Calculator* dialog, there are several windows which contain the different data sets, the time steps associated with those data sets, and buttons for doing calculations with the datasets. We first want to find the difference between the One-way Steering and the ADCIRC-only analyses. In the Data Sets window, select *b.magnitude (64)*. Make sure the *Solution* is StAd_One_Way and turn on the *Use all time steps* toggle. (If the *Solution* of *b.magnitude (64)* is Adcirc_Only, select the *magnitude (64)* that has the StAd_One_Way first, and then for step 3, select the *magnitude (64)* that is the Adcirc_Only solution).

3. Click the *Add to Expression* button. The Expression edit field now shows b:all. Click the subtract (-) button. Select *d.magnitude (64)* and make sure the *Solution* is Adcirc_Only. Turn on the *Use all time steps toggle* and push *Add to Expression*. The Expression field should now read b:all-d:all. Enter Vel_Mag_Diff in the *Result* field and click *Compute*.

After a few seconds, the dialog will refresh, with a new data set entitled Vel_Mag_Diff. This is the resultant data set from the calculation that we have just completed. We will view the results of that calculation in a couple of minutes. Before viewing the datasets, we want to also create a data set of the differences in water surface elevations for the two solutions. To create this new data set:

1. Delete the text in the *Expression* and *Result* edit fields.

2. In the Data Sets window, select c.*water surface elevation (63)*. Make sure the *Solution* is StAd_One_Way and turn on the *Use all time steps* toggle. . (If the *Solution* of *c.water surface elevation (63)* is Adcirc_Only, select the

water surface elevation (63) that is the StAd_One_Way solution first, and then for step 3, select the *water surface elevation (63)* that is the Adcirc_Only solution).

3. Click the *Add to Expression* button. The Expression edit field now shows c:all. Click the subtract (-) button. Select *e.water surface elevation (63)* and make sure the *Solution* is Adcirc_Only. Make sure the *Use all time steps toggle* is on and push *Add to Expression*. The Expression field should now read c:all-e:all. Enter WSE_Diff in the *Result* field and click *Compute*.

This calculation will take a minute or two to complete. Once it is completed, the dialog will refresh, and there will be a WSE_Diff dataset, which represents the difference in water surface elevation between the two solutions. We are now done using the Data Calculator. Click the Done button to exit the Data Calculator.

7.5.5 Viewing Differences-Velocity Magnitude & Water Surface

Now that we are finished with the data calculations, we are ready to view the results. Zoom in on the inlet of Grays Harbor if you are not able to clearly see it and surrounding shoreline. Resize the main SMS window so that the Scalar and Vector combo boxes may be seen. We will be using the Scalar edit box to switch between the different data sets that we have just created.

We will first look at the difference of velocity magnitudes between the two analyses. Set the *Solution* combo box to Generic Datasets and the *Scalar* combo box to Vel_Mag_Diff. This dataset will allow us to see a visual representation of the differences in the velocity magnitudes. Change the *Time Step* to 219,600. This is the timestep where the peak ebb occurs.

The contours show the difference between the one-way coupling solution and the ADCIRC only solution. Notice the red areas near the shore, which show the significant impact of the waves on the circulation model. Feel free to spend some time viewing the results at other time steps. For your information, the next peak flood is at time step 249,300.

We also want to see the difference that the coupling makes in the water surface elevation. To do this, change the *Scalar* combo box from Vel_Mag_Diff to WSE_Diff. Look at the time steps for both the peak ebb (219,600) and the peak flood (249,300). By looking at both of these time steps, we can see that the water surface elevation solution doesn't change significantly when coupling with a wave model.

7.6 Conlcusion

This concludes the workshop for ADCIRC-STWAVE steering. In addition to ADCIRC-STWAVE steering, there are several other steering types. One of these, M2D-STWAVE steering, will be taught in one of the following workshops.