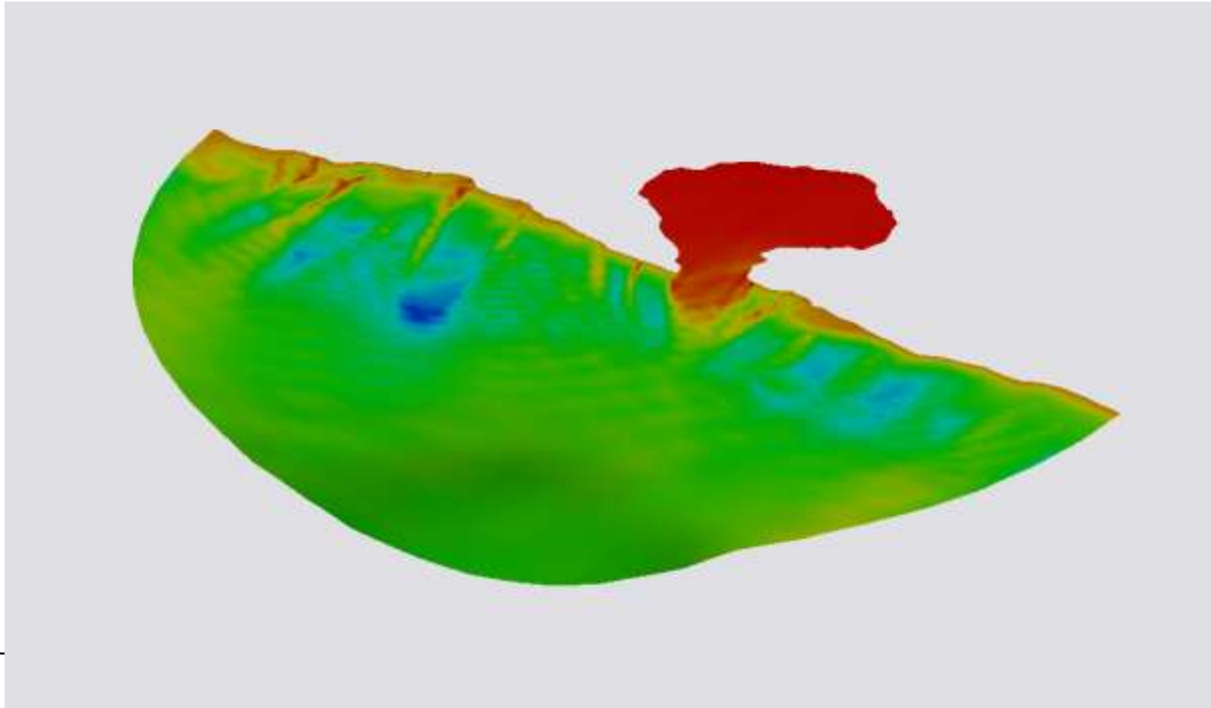
v. 12.1

SMS 12.1 Tutorial

CGWAVE Analysis



Objectives

This tutorial instructs how to prepare a mesh for analysis, and how to run a solution for CGWAVE.

Prerequisites

- Overview Tutorial

Requirements

- CGWAVE
- Scatter Module
- Map Module
- Mesh Module

Time

- 45-60 min

1 Getting Started

First, open the XYZ file containing a set of points with depth data from which a mesh will be created by doing the following:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Select “indiana.xyz” in the *data files* folder for this tutorial and click the **Open** button to open the *File Import Wizard – Step 1 of 2* dialog.
3. In the *File import options* section, toggle on *Space*.
4. In the section between *File import options* and *File preview*, set *Start import at row* to “2”.
5. Toggle off *Heading row*.
6. Click **Next >** to open the *File Import Wizard – Step 2 of 2* dialog.
7. Click **Finish** to open the file and close the *File Import Wizard – Step 2 of 2* dialog.

A scatter set named “indiana” will be created in the Project Explorer. This data is referenced to a UTM coordinate frame and is in meters. To give this information to SMS:

1. Select *Display* | **Projection…** to bring up the *Display Projection* dialog.
2. In the *Horizontal* section, select the *No projection* radio button.
3. Select “Meters” from the *Units* drop-down in both the *Horizontal* and *Vertical* sections.
4. Click **OK** to close the *Display Projection* dialog.

 © Aquaveo 2015

2 Creating a Wavelength Function

The first step in creating a mesh for CGWAVE is to create a wavelength function. The wavelength function is an intermediate step to creating a size function, which is covered in section 3, "Creating a Size Function".

The z value of each point in the "indiana.xyz" data is actually a water depth value. The wavelength at each point is calculated from this depth value using a complicated equation. It is sufficient to say that a larger wavelength is calculated from a larger water depth value.

To create the wavelength function:

1. Select the "indiana" dataset in the Project Explorer make it active.
2. Select *Data* | **Dataset Toolbox…** to bring up the *Dataset Toolbox* dialog.
3. In the *Tools* section, select *Coastal* | **Wave Length and Celerity**.
4. In the *Wave Length and Celerity* section, enter "Transition" for *Output base name*.
5. Enter "20.0" for the *Period*.
6. Click **Compute**.
7. Click **Done** to close the *Dataset Toolbox* dialog.

Two new datasets, "Transition_Wavelength" and "Transition_Celerity", will be visible in the Project Explorer.

3 Creating a Size Function

The size function is created from the wavelength function. The size function determines the element size that will be created by SMS. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

The wavelength function created above contains values that are twice as large as the desired size values. The wavelength function will be scaled by one half to create the size function.

To do this:

1. Select *Data* | **Data Calculator** to bring up the *Dataset Toolbox* dialog.
2. In the Datasets sub-section of the Data Calculator section, double-click the dataset with "Transition_Wavelength" in the title to add it to the expression in the Calculator section.
3. Click **/** (division) on the calculator button pad.
4. Press the "5" on the keyboard.

This represents the number of elements generated per wavelength. It is usually more appropriate to use a larger number of elements per wavelength (e.g., 10). The smaller number is used here to allow faster execution of the model.

 © Aquaveo 2015

5. Enter “size” in the *Output dataset name* field and click **Compute**.
6. Click **Done** to close the *Dataset Toolbox* dialog.

A new dataset, “size”, based on the Transition_Wavelength dataset, will appear in the Project Explorer.

3.1 Smooth Size Function

The final step in creating a size function is to smooth the size function. This modifies the size function so the its values do not change too quickly. Size functions that change too quickly can create poor transitions in element size.

1. Select *Data* | **Dataset Toolbox** to bring up the *Dataset Toolbox* dialog.
2. In the *Tools* section, select *Spatial* | **Smooth datasets**.
3. In the *Datasets* subsection of the *Smooth datasets* section, select the “size” dataset.
4. In the Smoothing Options section, selection the Element area change radio button.
5. Enter “0.5” in the *Area change limit* field. This modifies the size function so the elements created by it are—at most—twice as big or half as small as their adjacent elements.
6. Enter “size smoothed 0.5” in the *Output dataset name* field and click **Compute**.
7. Click **Done** to close the *Dataset Toolbox* dialog.

If desired, the differences between the dataset “size” and “size smoothed 0.5” can be visualized by using the data calculator to subtract “size” from “size smoothed 0.5” and contouring the resulting dataset.

4 Defining the Domain

A domain represents the region that is offshore. In CGWAVE, the domain can be a circular, semi-circular, or rectangular region. In SMS, a feature arc is used to define the coastline. After the coastline is defined, feature arcs and feature polygons are used to define the domain region.

4.1 Creating the Coastline

SMS can automatically create a coastline at a specific elevation or water depth from a scattered dataset. The active function of the active scattered dataset will be used for this operation. The project should currently have only one scattered dataset. To make the elevation function active:

1. Select the “Z” dataset in the Project Explorer to make it active.
2. Right-click on the “Area Property” coverage and select *Type* | *Models* | **CGWAVE**.
3. Right-click on “Area Property” and select **Rename**.

 © Aquaveo 2015

4. Enter "CGWAVE" and press the *Enter* key to set the new name.
5. Select "CGWAVE" to make it active.

With the coverage type set and the active scattered dataset defined, create the coastline by doing the following:

1. Select *Feature Objects* | **Create Coastline…** to bring up the *Create Contour Arcs* dialog.
2. In the *Contour options* section, enter "1.0" for the *Elevation*.
3. Enter "10.0" for the *Spacing*.
4. Click **OK** to close the *Create Contour Arcs* dialog.

The display will refresh with an arc representing the 1.0 water depth line, as shown in Figure 1.

Figure 1 The arc representing the coastline

4.2 Creating the Domain

SMS can create a domain from the coastline. This model will use a semi-circular domain that intersects with the coastline.

To create the domain:

1. Using the **Select Feature Vertex** tool, hold down the *Shift* key and select one vertex near each end of the coastline arc.

© Aquaveo 2015

2. Select *Feature Objects* | **Define Domain** to bring up the *Domain Options* dialog.
3. Select *Semi-circular* and click **OK** to close the *Domain Options* dialog. This creates a semicircular *Ocean* arc as shown in Figure 2.
4. Select *Feature Objects* | **Build Polygons**. This creates a feature polygon must be created from the feature arcs.

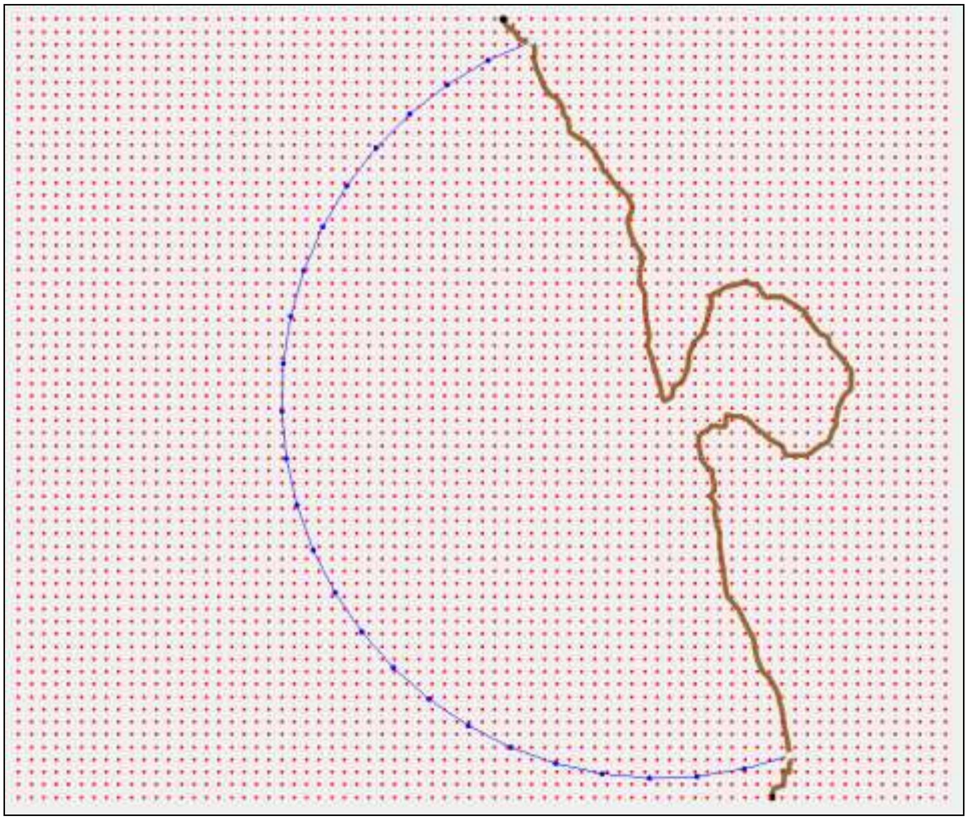


Figure 2 The ocean domain arc

The polygon is formed from the set of arcs that form a closed loop (in this case, from the ocean domain arc and the part of the coastline arc between the ends of the ocean domain arc). The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created.

If the domain was created on the wrong side of the coastline, it indicates that the coastline is oriented in the wrong direction. If this happens, use these steps to correct it:

1. Using the **Select Feature Arc** tool, select the semi-circular arc and delete it.
2. Select the coastline arc, then right-click and select **Reverse Arc Direction**.
3. Select the two nodes remaining from the semi-circular arc using the **Select Feature Point** tool and repeat the steps at the beginning of this section to create the domain.

Make sure that the domain does not extend outside the extent of the scatter set. If it does, delete the semi-circular arc and recreate it with points further within the interior of scatter set.

 © Aquaveo 2015

5 Creating the Finite Element Mesh

There are various automatic mesh generation techniques that can be used to create elements inside a specified boundary. One of these is applied to each polygon, after which a finite element mesh can be generated. For this tutorial, there is only one polygon, which will be assigned the density mesh type.

5.1 Setting up the Polygon

When using density meshing, SMS determines element sizes from a *size function* in a scattered dataset. The size function to be used in this example was created in a previous section.

To set up the feature polygon for density meshing:

1. Using the **Select Feature Polygons** tool, double-click inside the ocean domain polygon to bring up the *2D Mesh Polygon Attributes* dialog.
2. In the *Mesh Type* section, select *Scalar Paving Density* from the drop-down.
3. Click the **Scatter Options…** button to bring up the *Interpolation* dialog.
4. In the *Interpolation Options* section, enter "10.0" in the *Single Value* field. The value in this field and the *Min* field, below, must be the same. If they are not when closing the dialog, a dialog will advise they have been changed to match when exiting the *Interpolation* dialog.
5. Toggle on *Truncate values* in the *Other Options* section.
6. Enter "10.0" in the *Min* field.
7. Enter "10000.0" in the *Max* field. This sets up a minimum and maximum size to be used when creating elements.
8. In the *Scatter Set to Interpolate From* section, select the "size smoothed 0.5" dataset.
9. Click **OK** to close the *Interpolation* dialog.
10. Select "Scatter Set" from the *Bathymetry Type* drop-down.
11. In the *Bathymetry Type* section, select *Scatter set*.
12. Click on the Scatter Options… button to bring up the *Interpolation* dialog.
13. In the *Scatter Set to Interpolate From* section, select "Z" from the list of datasets.
14. Toggle off *Truncate Values* in the Other Options section. As mesh nodes are created, their elevation values will be assigned from the original water depth values that were imported from the original XYZ file.
15. Click **OK** to close the *Interpolation* dialog.
16. Click ***OK*** to close the *2D Mesh Polygon Properties* dialog.

The polygon is now set up to generate finite elements inside the boundary. When more than one polygon exists, the meshing attributes need to be set up for each of the polygons.

© Aquaveo 2015

5.2 Generating the Elements

Since there is only one polygon in this tutorial, it's time to have SMS generate the finite element mesh from the defined domain.

To create the mesh:

1. Select *Feature Objects* | **Map→2D Mesh** *to bring up the 2D Mesh Options dialog*.
2. *Toggle off Copy coverage before meshing*.
3. *Click* **OK** *to close the 2D Mesh Options dialog and bring up the Mesh Name dialog.*
4. *Accept the default name by clicking* **OK** *to close the Mesh Name dialog.*
5. *Toggle off Scatter Data and Map Data in the Project Explorer*.

The Main Graphics Display should now appear as in Figure 3.

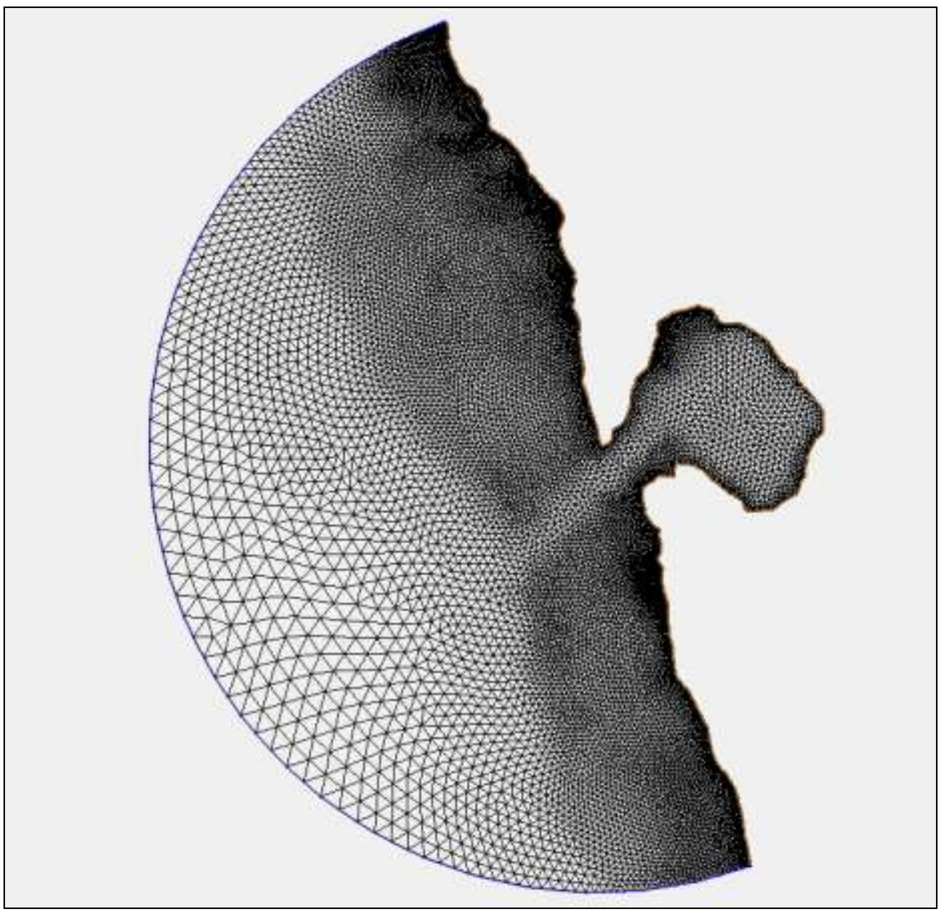


Figure 3 The mesh after turning off scatter and map data

To further declutter the project display, do the following:

1. Select the ***Display Options*** macro to bring up the Display Options dialog.
2. Select "2D Mesh" from the list on the left.

 © Aquaveo 2015

3. On the *2D Mesh* tab, click the All Off button.
4. Toggle on *Elements*, *Contours*, and *Nodestrings*.
5. On the *Contour Options* tab in the *Contour method* section, select “Color Fill” from the drop-down.
6. In the *Contour interval* section, select “Number” from the drop-down and enter “20” in the field to the right of the drop-down.
7. Click **OK** to close the *Display Options* dialog. The display should appear as in Figure 4.

After the display is refreshed, notice the contours of water depth with the elements drawn on top of those. As the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.

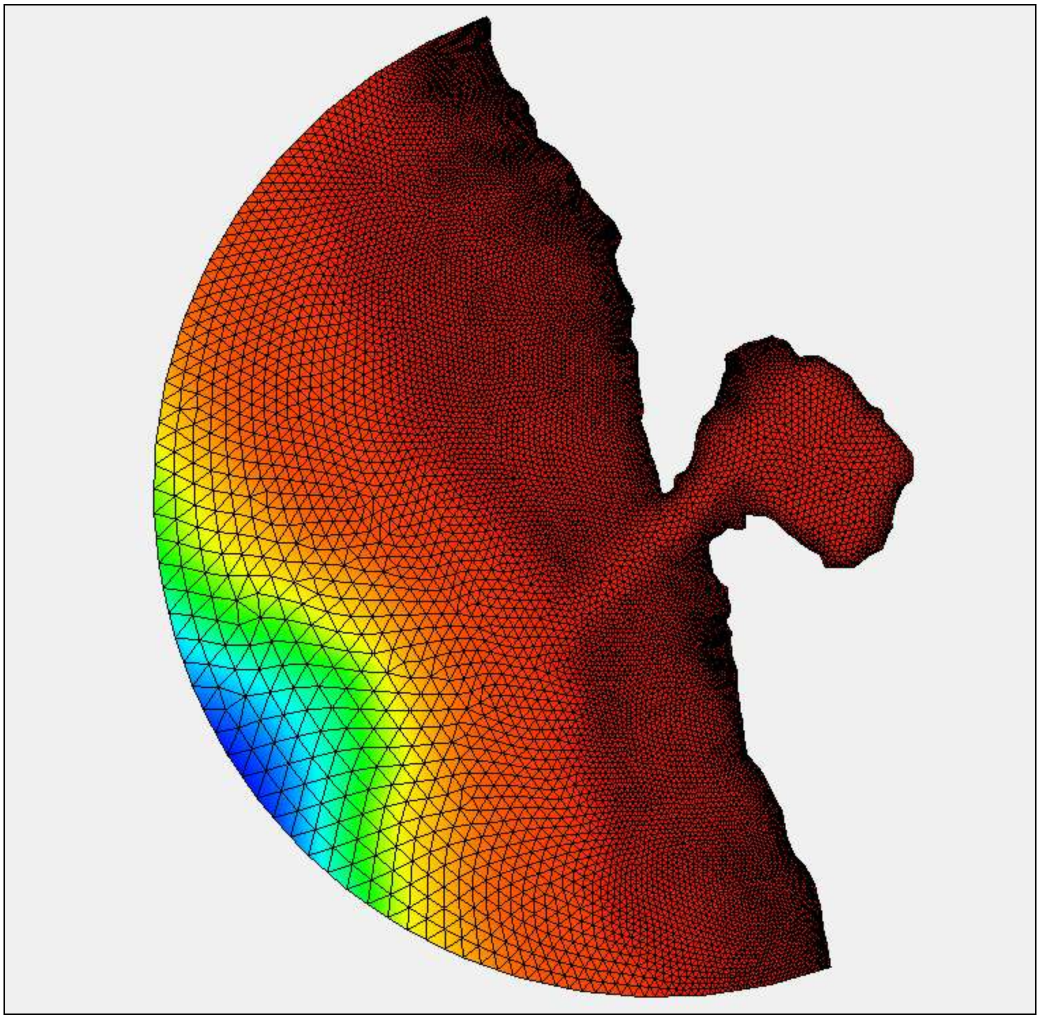


Figure 4 The contour color fill shows the increasing depth

 © Aquaveo 2015

6 Model Control

When creating a CGWAVE model, the boundary conditions are wave amplitude, phase, and direction.

To define these incident wave conditions:

1. Select CGWAVE Mesh in the Project Explorer to make it active.
2. Select *CGWAVE* | **Model Control…** *to bring up the CGWAVE Model Control dialog*.
3. In the *Incident Wave Conditions* section, set:
    - *Direction (˚)* to “30.0”.
    - *Period (sec)* to “20.0”.
    - *Amplitude (m)* to “1.0”.
4. In the *Solver Options* section, set:
    - *Output Echo Frequency* to “1”.
    - *Maximum Iterations* to “500,000”.

CGWAVE uses a 1D file. The 1D parameters must be set in this dialog and the 1D depths extracted. By default, the ideal spacing is computed and the number of 1D nodes is set to run to 1.5˚ radius away from the coastline. Leave the settings in the *1D Domain Extension Parameters* section at the defaults.

5. In the *1D Domain Extension Parameters* section, click on the **Extract Depths** button to bring up the *Select Dataset* dialog.
6. Select “Z” from the list in the *Select* section, then click **Select** to close the *Select Dataset* dialog.
7. Click **OK** button to close *the CGWAVE Model Control dialog*.

7 Renumbering the Mesh

The mesh needs renumbering before being saved using the following steps:

1. Using the **Select Nodestring** tool,
2. Select the ocean domain nodestring by clicking in the selection box at the center of the nodestring.
3. Right-click and select **Renumber Nodestrings**.

8 Saving the CGWAVE Data

CGWAVE uses a geometry file and the 1D file mentioned above to run an analysis. This file consists of two lines that run perpendicular from the coastline to the extents of the domain. The 1D file is generated automatically by SMS using the active scatter set. The file contains depth information on both sides of the domain. To save these files:

 © Aquaveo 2015

1. Select *File* | **Save New Project…** to bring up the *Save* dialog.
2. Enter “indianaout.sms” in the *File name* field.
3. Select “Project Files (*.sms)” from the *Save as type* drop-down.
4. Click **Save** to save the project and close the *Save* dialog.

9 Running CGWAVE

CGWAVE can be run from SMS by doing the following:

1. Click on CGWAVE Mesh in the Project Explorer to make it active.
2. Select *CGWAVE* | **Run CGWAVE** to bring up the *CGWAVE* dialog.
3. When the model is finished running, toggle on *Load solution* and click **Exit** to close the *CGWAVE* dialog and bring up the *CGWAVE Solutions Options* dialog.
4. Click **OK** to accept the default settings, close the *CGWAVE Solution Options*, and open the *CGWAVE – Trans* dialog.
5. When the process is finished, click **Exit** to close the *CGWAVE – Trans* dialog and bring up the *Dataset Time Information* dialog.
6. Click **OK** to accept the defaults and close the *Dataset Time Information* dialog. The Main Graphics Window should appear as in Figure 5.

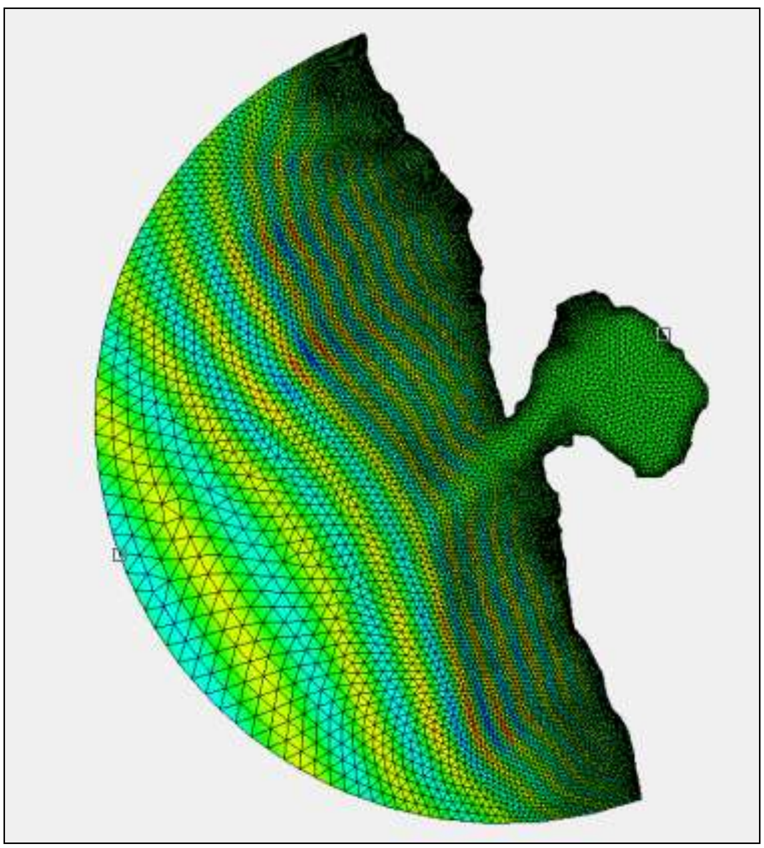


Figure 5 After running CGWAVE

© Aquaveo 2015

SMS saves the location of the CGWAVE executable as a preference. If this preference is defined, the model will launch and run per step 3, above. If the preference is undefined, SMS shows a message that the CGWAVE executable is not found.

1. If the CGWAVE executable is not found, click the **File Browser** button bring up the *Open* dialog.
2. Browse to the location of the CGWAVE executable and select it.
3. Click **Open** to close the *Open* dialog.
4. Click **OK** button to run the model.

One of the model parameters for CGWAVE is wave breaking. If this option is on, the model will compute how the waves break. If not, users can still approximate the breaking by selecting the option to break the waves as reading the solution file.

When opening the file, SMS will translate the wave output into datasets that can be visualized. These include phase, wave height, wave direction, sea surface, pressure and particle velocity at three locations in the water column, and a time series of wave surface and wave velocity over a wave cycle.

If CGWAVE does not run, the user may have an older version of CGWAVE. If this is the case, do the following:

1. Open the *data files\indianaout.cgi* file in a text editor.
2. Remove the following line from near the beginning of the file:

   ```
   %maximum iterations for nonlinear mechanisms &
   ```

3. Change the following line from this:

   ```
   12        35        1    500000        1000      8
   ```

   to this:

   ```
   12        35        1    500000        8
   ```

10 Post Processing

Now that CGWAVE has finished running and the solutions have been added to the Project Explorer, the different solutions that were generated can be viewed. Solutions can be viewed directly in SMS by selecting the different meshes that were generated during CGWAVE run. Film loops can also be generated.

10.1 Functional Surface

To make it easier to view the wave transitions, do the following:

1. Select *Display* | **Display Options**.
2. Select *2D Mesh* from the list on the left.
3. Toggle everything off except for *Functional Surface*.
4. Click on the **Options** button next to *Functional Surface* to bring up the *Functional Surface Options* dialog.

 © Aquaveo 2015

5. In the *Z Offset* section, select “Display surface above geometry” from the drop-down.
6. In the *Z Magnification* section, toggle on *Override global value* and enter “50.0” in the *Magnification value* field.
7. In the *Display Attributes* section, select the *Contour surface* radio button and click on the **Options…** button to bring up the *Dataset Contour Options* dialog.
8. In the *Color method* section, click on the **Color Ramp** button to open the *Color Options* dialog.
9. In the *Palette Method* section, select *Intensity Ramp*.
10. Click on the large *Color* button (not the drop-down arrow next to it) to bring up the *Color* dialog.
11. Select blue and click **OK** to close the *Color* dialog.
12. In the *Current Palette* section, move the black arrows toward the center slightly so too much white or black is not included in the palette.
13. Click **OK** to close the *Dataset Contour Options* dialog.
14. Click **OK** to close the *Functional Surface Options* dialog.
15. Click **OK** to close the *Display Options* dialog.
16. Toggle off Map Data in the Project Explorer so the mesh is the only data visible.
17. Select the “Max Velocity - Bed” vector dataset and the “Sea Surface Elevation” scalar dataset to make them active.
18. Using the **Rotate** tool, rotate the mesh so it looks like Figure 6.

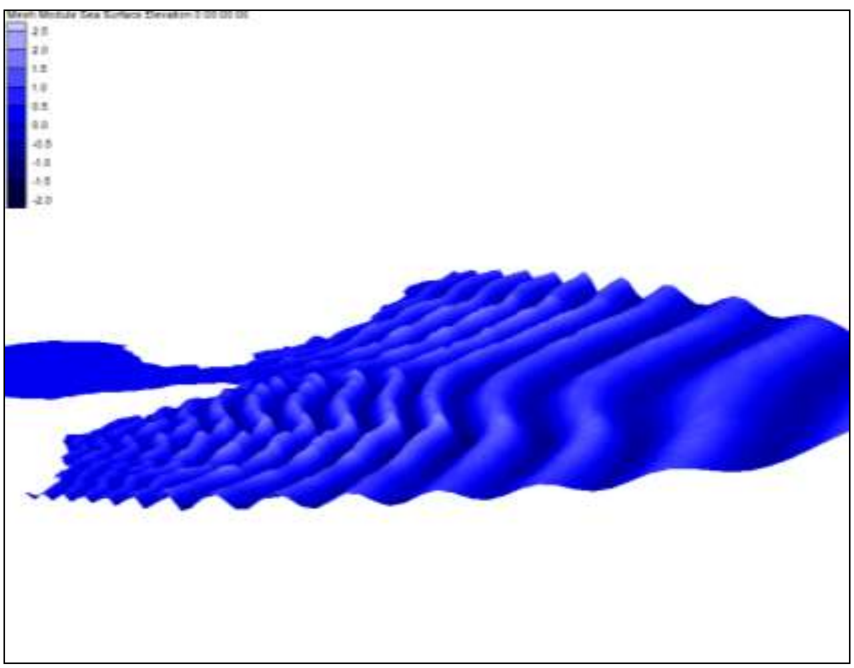


Figure 6 Functional surface showing waves

 © Aquaveo 2015

10.2 Film Loops

Film loops can be very useful when showing different solutions that were generated. Film loops can be embedded (e.g., in websites or documents) which can be a useful and quick way to show how SMS and the different modules worked.

To create a film loop, do the following:

1. Select *Data* | **Film Loop** to bring up the *Film Loop Setup – General Options* dialog.
2. Toggle on *Create AVI File* and click on the **Folder Selector** icon to bring up the *Save* dialog.
3. Browse to the *data files* folder for this tutorial.
4. Enter “CGWAVE.avi” in the *File name* field.
5. Select “AVI File (*.avi)” from the *Save as type* drop-down.
6. Click **Save** to close the *Save* dialog.
7. Click **Next** to bring up the *Film Loop Setup – Time Options* dialog.
8. Select the *Specify Number of Frames* radio button and enter “50” in the field to the right.
9. Click **Next** to bring up the *Film Loop Setup – Display Options* dialog.
10. Enter “90” in the *Quality* field. This will create less pixilated frames and a smoother film loop.
11. Click **Finish** to close the *Film Loop Setup – Display Options* dialog.
12. SMS will create the film loop and then launch the Play AVI Application. Once finished viewing the AVI, click the **Close** button at the top right corner of the window.

If the user wishes to embed the film loop in a document or share it online, the file is in the *data files* folder for this tutorial. The AVI file may be viewed in any video viewing software such as Windows Media Player.

11 Conclusion

This concludes the *CGWAVE Analysis* tutorial. The user may continue to make more videos and look through the other meshes created, or exit SMS at this point.

© Aquaveo 2015