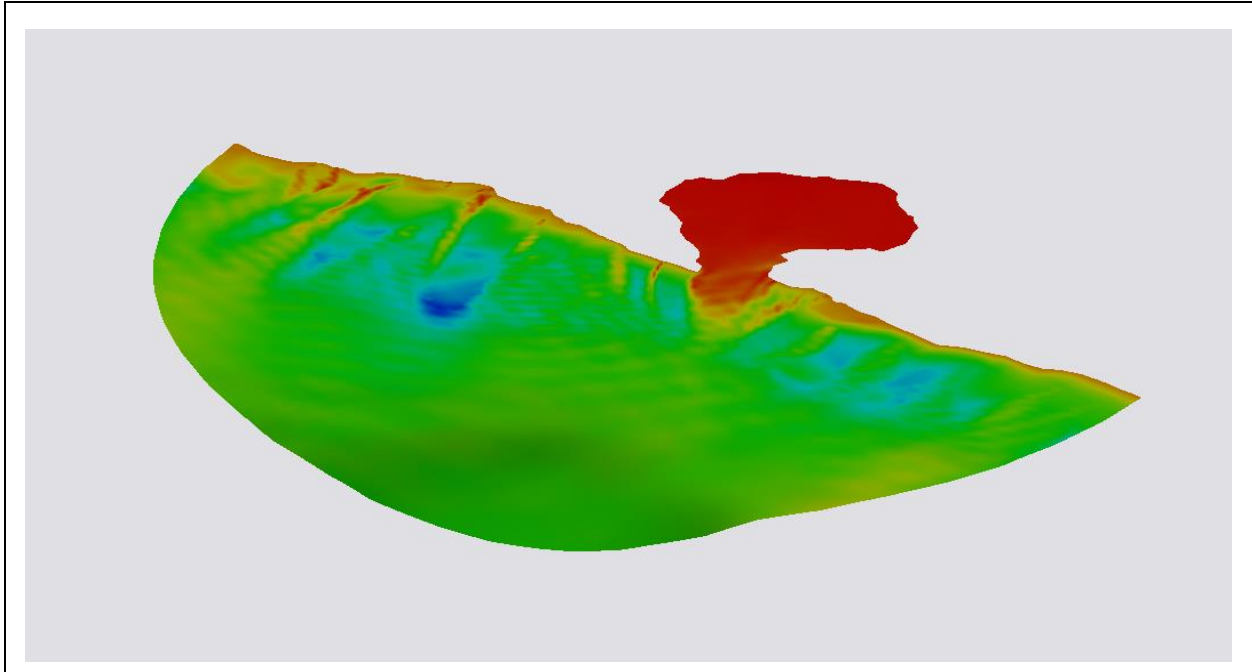


SMS 12.2 Tutorial

CGWAVE Analysis



Objectives

Learn 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	2
2	Creating a Wavelength Function	3
3	Creating a Size Function	3
3.1	Smooth Size Function	4
4	Defining the Domain	5
4.1	Creating the Coastline	5
4.2	Creating the Domain	6
5	Creating the Finite Element Mesh.....	7
5.1	Setting up the Polygon	7
5.2	Generating the Elements	8
6	Model Control	10
7	Renumbering the Mesh	11
8	Saving the CGWAVE Data	11
9	Running CGWAVE	11
10	Post Processing.....	13
10.1	Functional Surface.....	13
10.2	Film Loops	14
11	Conclusion	15

1 Getting Started

First, open the XYZ file containing a set of points with depth data. This will be used to create a mesh.

1. Select *File* | **Open...** to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select “indiana.xyz”.
3. Click **Open** to exit the *Open* dialog and open the *Step 1 of 2* page of the *File Import Wizard* dialog.
4. In the *File import options* section, turn on *Space*.
5. Below the *File import options* section, enter “2” as the *Start import at row*.
6. Turn off *Heading row*.
7. Click **Next** to go to the *Step 2 of 2* page of the *File Import Wizard* dialog.
8. Click **Finish** to import the file and close the *File Import Wizard* dialog.

A scatter set named “indiana” will be created in the Project Explorer, and the project should appear similar to Figure 1. 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.

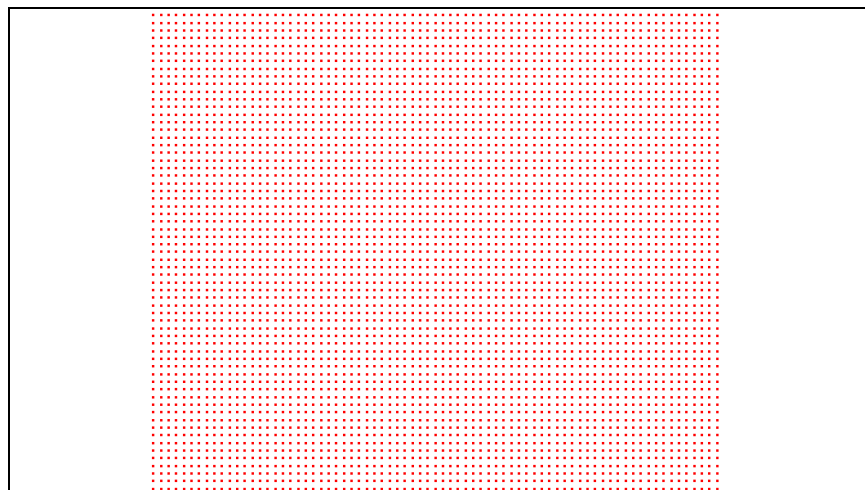



Figure 1 Scatter set from the imported XYZ file



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 the next section.

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. A larger wavelength is calculated from a larger water depth value.

To create the wavelength function:

1. Select “ indiana” 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 “Wave Length and Celerity” from the tree list.
4. In the *Wave Length and Celerity* section, enter “Transition” as the *Output base name*.
5. Enter “20.0” as 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 / Dataset Toolbox...* to bring up the *Dataset Toolbox* dialog.
2. In the *Tools* section, select “Data Calculator” from the tree list.
3. In the *Datasets* section of the *Data Calculator* section, double-click “d2. Transition_Wavelength” to add it to the expression in the *Calculator* section.
4. Click / (division) on the calculator button pad.
5. 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.

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

A new dataset—“ size”—based on the “ Transition_Wavelength” dataset will appear in the Project Explorer. The project should appear similar to Figure 2.

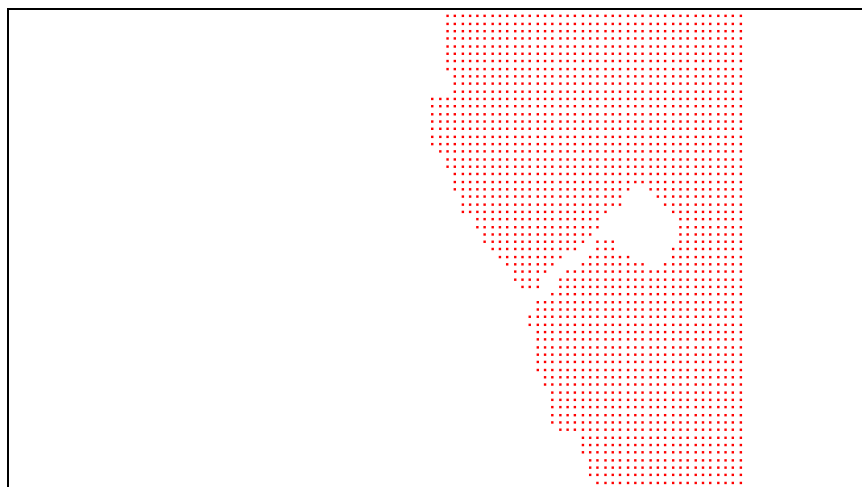


Figure 2 Scatter set after creating a size function

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 that 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 *Smooth datasets*.
3. In the *Datasets* subsection of the *Smooth datasets* section, select the “size” dataset.

4. In the *Smoothing Options* section, select *Element area change*.
5. Enter “0.5” as the *Area change limit*.

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” as the *Output dataset name* 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 “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**.
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” as the *Elevation*.
3. Enter “10.0” as the *Spacing along contour*.
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 3.

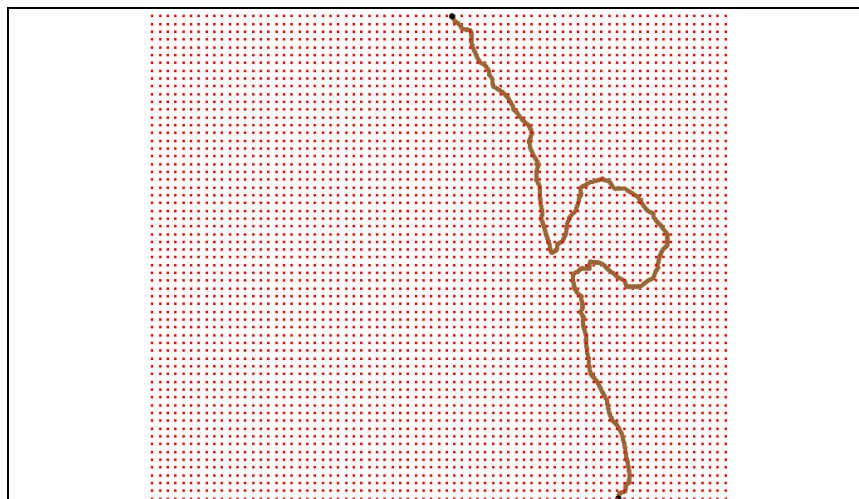



Figure 3 The arc representing the coastline

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.
2. Select *Feature Objects* | **Define Domain...** to open the *Domain Options* dialog.
3. Select *Semi-circular* and click **OK** to close the *Domain Options* dialog.
4. Select *Feature Objects* | **Build Polygons**.

This creates a semicircular *Ocean* arc and a feature polygon (Figure 4). The feature polygon must be created from the feature arcs. 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.

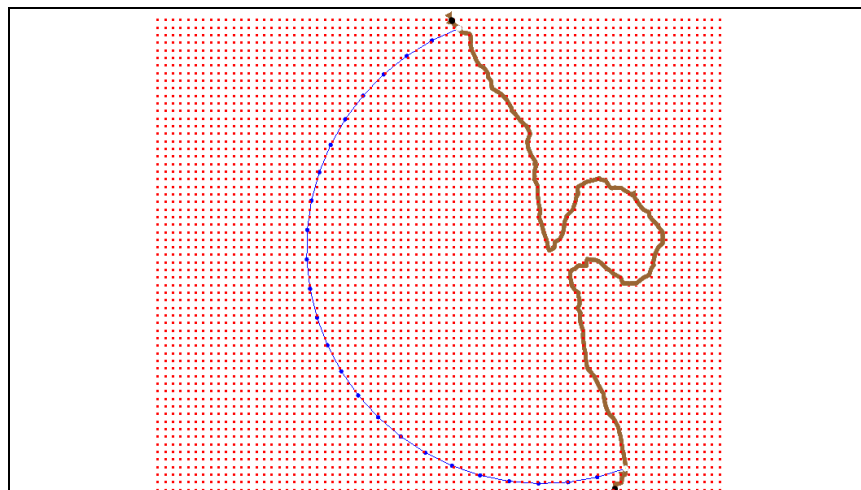




Figure 4 The ocean domain arc

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, this 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 steps 2–4 from the beginning of this section to create the domain.


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 Section 3.

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 Properties* dialog.
2. In the *Mesh Type* section, select *Scalar Paving Density* from the drop-down.
3. Click **Scatter Options...** 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 in the *Min* field—in step 6 below—must be the same. If they do not match when closing the *Interpolation* dialog, a warning will advise they have been changed to match.

5. In the *Other Options* section, turn on *Truncate values*.
6. Enter “10.0” as the *Min*.
7. Enter “10000.0” as the *Max*.

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. In the *Bathymetry Type* section, select “Scatter Set” from the drop-down.
11. Click **Scatter Options...** to bring up the *Interpolation* dialog.
12. In the *Scatter Set to Interpolate From* section, select “Z” from the list of datasets.
13. In the *Other Options* section, turn off *Truncate Values*.

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.



14. Click **OK** to close the *Interpolation* dialog.
15. 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. As only one polygon exists in this tutorial, this process is complete.

5.2 Generating the Elements

Now 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. Turn 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. Turn off “ Scatter Data” and “ Map Data” in the Project Explorer.

The Main Graphics Display should now appear as in Figure 5.

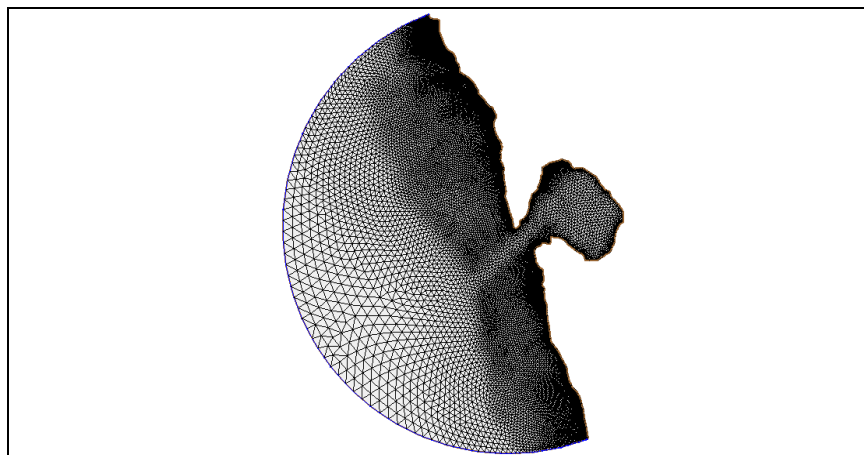



Figure 5 The mesh after turning off scatter and map data

To further declutter the project display, do the following:

1. Click **Display Options**  to bring up the *Display Options* dialog.
2. Select “2D Mesh” from the list on the left.
3. On the *2D Mesh* tab, click **All Off**.
4. Turn 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.

After the display is refreshed, notice the contours of water depth with the elements drawn on top of those (Figure 6). As the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.

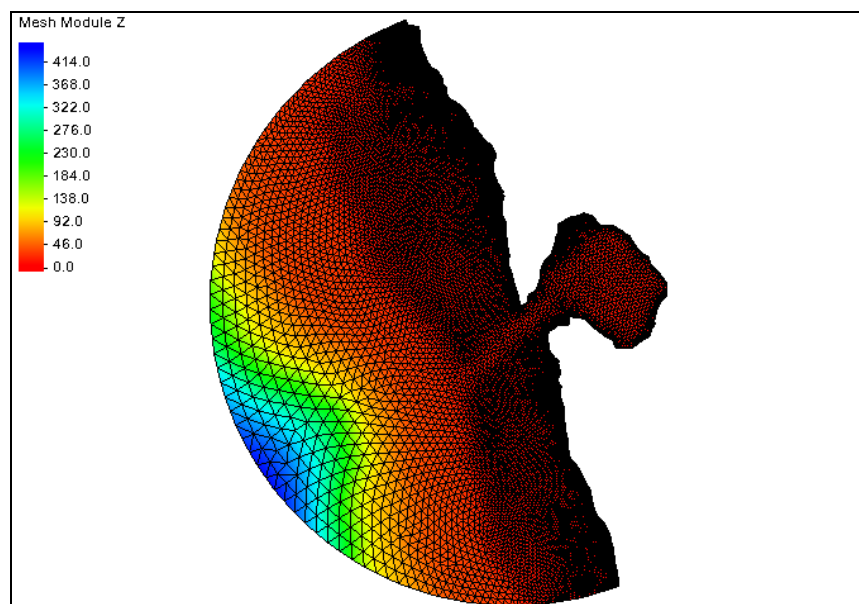



Figure 6 The contour color fill shows the increasing depth

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 open the *CGWAVE Model Control* dialog.
3. In the *Incident Wave Conditions* section, enter “30.0” in the *Direction (°)* column.
4. Enter “20.0” in the *Period (sec)* column.
5. Enter “1.0” in the *Amplitude (m)* column.
6. In the *Solver Options* section, enter “1” as the *Output Echo Frequency*.
7. Enter “500,000” as the *Maximum Iterations*.

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.

8. In the *1D Domain Extension Parameters* section, click **Extract Depths** to bring up the *Select Dataset* dialog.
9. In the *Select* section, select “Z” and click **Select** to close the *Select Dataset* dialog.
10. Leave the remaining settings at the defaults and click **OK** 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, select the ocean domain nodestring by clicking in the selection box at the center of the nodestring.
2. 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:

1. Select *File / Save New Project...* to bring up the *Save* dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.
3. Enter “indianaout.sms” in the *File name* field.
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. Select “ CGWAVE Mesh” in the Project Explorer to make it active.
2. Select *CGWAVE / Run CGWAVE* to bring up the *CGWAVE* model wrapper dialog.
3. When the model is finished running, turn 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.
7. Select “ Pressure – Surface” under “ Time Varying” in the Project Explorer to make it active.

The project should appear similar to Figure 7.

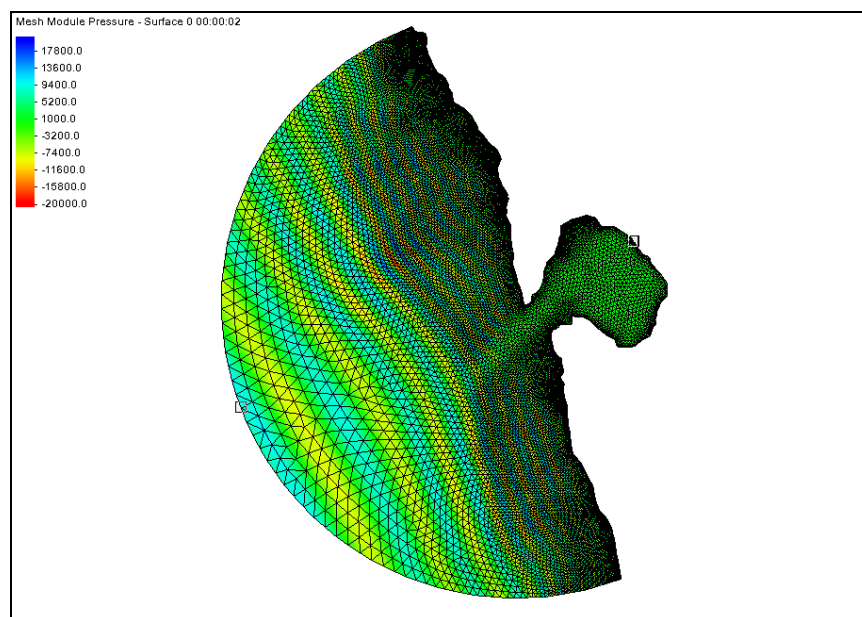



Figure 7 After running CGWAVE

9.1 Troubleshooting CGWAVE

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.

8. If the CGWAVE executable is not found, click the **File Browser**  button to bring up the *Open* dialog.
9. Browse to the location of the CGWAVE executable and select it.
10. Click **Open** to close the *Open* dialog.
11. 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. Click **Display Options**  to open the *Display Options* dialog.
2. Select “2D Mesh” from the list on the left.
3. Click **All Off**, then turn on *Functional Surface*.
4. Click **Options...** button next to *Functional Surface* to bring up the *Functional Surface Options* dialog.
5. In the *Z Offset* section, select “Display surface above geometry” from the drop-down.
6. In the *Z Magnification* section, turn on *Override global value* and enter “50.0” as the *Magnification value*.
7. In the *Display Attributes* section, select *Contour surface* and click **Options...** to bring up the *Dataset Contour Options – Pressure – Surface* dialog.
8. In the *Color method* section, click **Color Ramp...** 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 *Color Options* dialog.
14. Click **OK** to close the *Dataset Contour Options* dialog.
15. Click **OK** to close the *Functional Surface Options* dialog.
16. Click **OK** to close the *Display Options* dialog.
17. Hide “ Map Data” in the Project Explorer so the mesh is the only data visible.
18. Select the “ Max Velocity - Bed” vector dataset and the “ Sea Surface Elevation” scalar dataset to make them active.

19. Using the **Rotate**  tool, rotate the mesh so it looks like Figure 8.

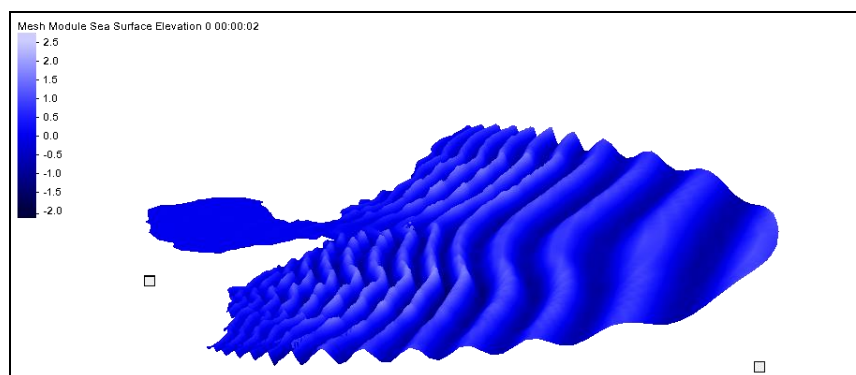



Figure 8 Functional surface showing waves

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 General Options page of the *Film Loop Setup* dialog.
2. Turn on *Create AVI File* and click **Folder Selector**  to bring up the *Save* dialog.
3. Browse to the *data files*\ folder for this tutorial.
4. Select “AVI File (*.avi)” from the *Save as type* drop-down.
5. Enter “CGWAVE.avi” in the *File name* field.
6. Click **Save** to close the *Save* dialog.
7. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.
8. Select *Specify Number of Frames* and enter “50” in the field to the right.
9. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.
10. Enter “90” as the *Quality*.

This creates less pixilated frames and a smoother film loop.

11. Click **Finish** to close the *Film Loop Setup* dialog.

SMS will create the film loop and then launch the Play AVI Application.

12. Once finished viewing the AVI, click the **Close**  button at the top right corner of the window.

If wanting 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 using any video viewing software which can display AVI files.

11 Conclusion

This concludes the “CGWAVE Analysis” tutorial. Continue to make more videos and look through the other meshes created, or exit SMS at this point.