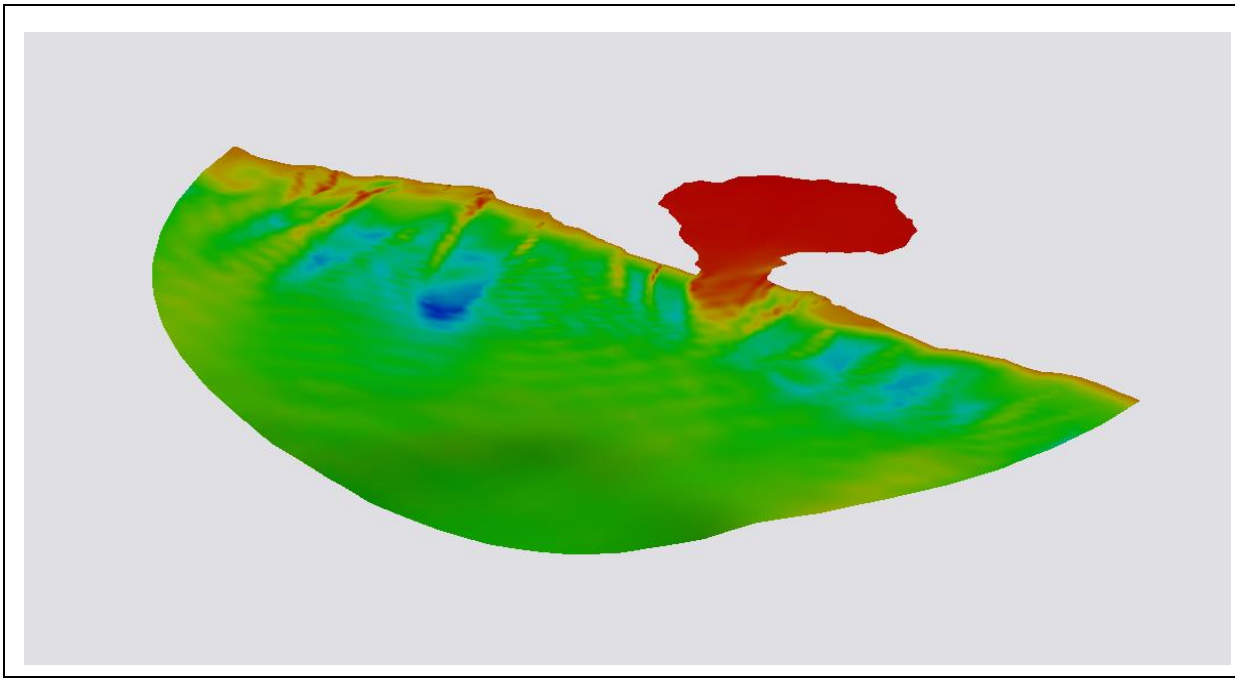
SMS

v. 13.0

*SMS 13.0 Tutorial*

# *CGWAVE Analysis*



## Objectives

Learn how to prepare a mesh for analysis, and how to run a solution for CGWAVE.

## Prerequisites

- Overview
- Map Module
- Wave Based Mesh

## Requirements

- CGWAVE
- Mesh Module

## Time

- 15–25 min

AQUAVEO™

# 1 Getting Started

Start the tutorial by opening the project file containing a mesh generated from a CGWAVE map coverage using a wave based function. For more information on how this was done, see the "Creating A Wave Based Mesh" tutorial.

To open the project:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files*\ folder for this tutorial and select "CGWAVE.sms".
3. Click **Open** to exit the *Open* dialog.

The mesh in the project will appear as in Figure 1. The CGWAVE model parameters will be applied to this mesh before running the model.

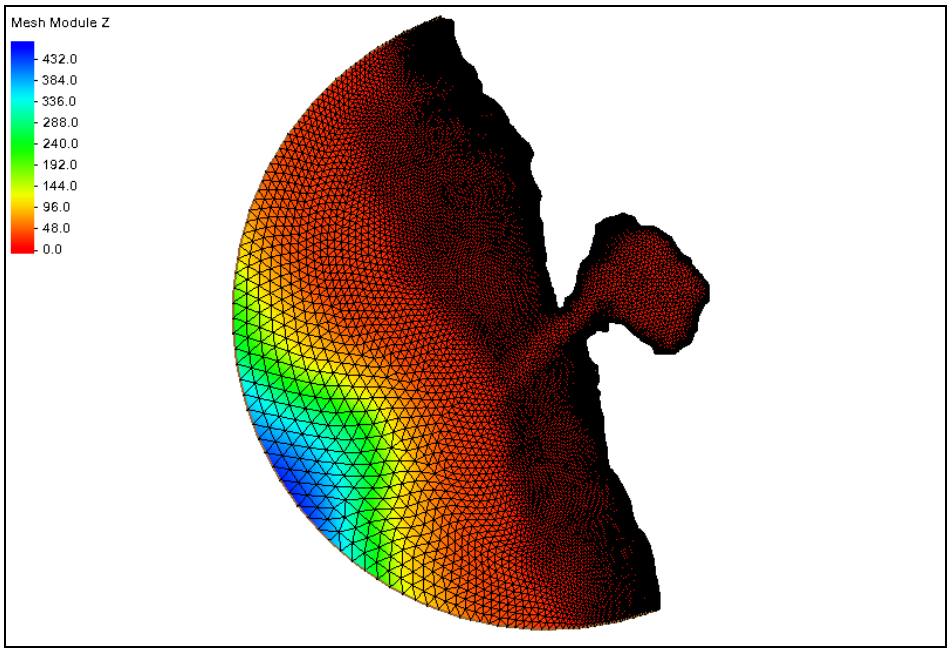



*Figure 1 Wave based mesh*

## 2 Saving a New Project

Before continuing with the tutorial, save the project under a new name so the original files can be used again later.

1. Select *File* | **Save As…** to bring up the *Save As* 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 As* dialog.

## 3 Set Model Parameters

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. The period specified should be at least as large as the period used to generate the size function that was used in the mesh generation.
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.

## 4 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:

5. Click **Save**.

# 5 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.

Now to view the results.

7. Select "Pressure – Surface" under "Time Varying" in the Project Explorer to make it active.

The project should appear similar to Figure 2.

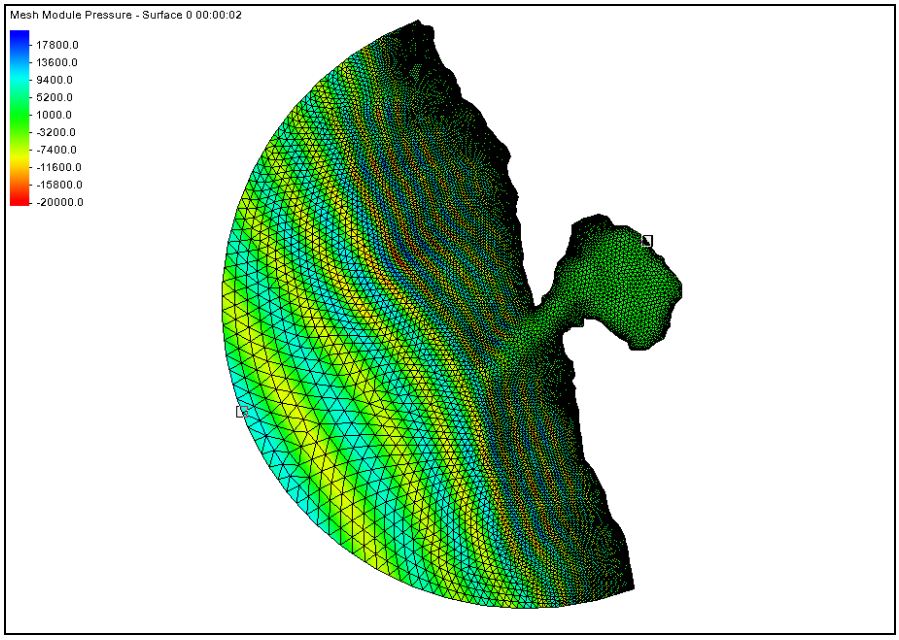



*Figure 2 After running CGWAVE*

## 5.1 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.

1. If the CGWAVE executable is not found, click the **File Browser** button to 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** 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
```

# 6 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.

## 6.1 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 3.

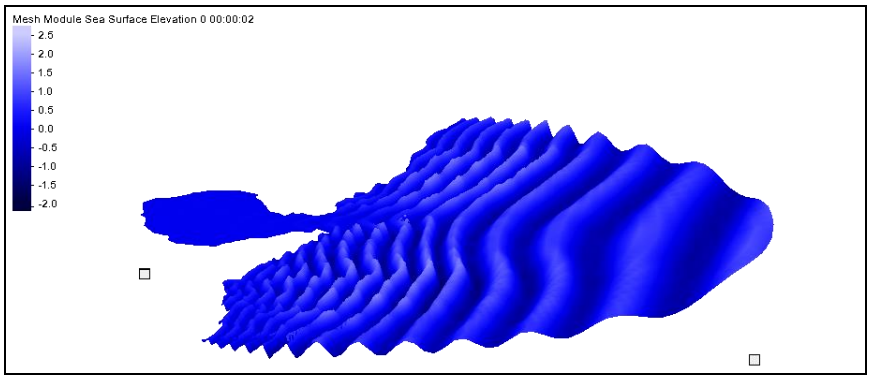



*Figure 3 Functional surface showing waves*

## 6.2 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 viewer software which can display AVI files.

# 7 Conclusion

This concludes the "CGWAVE Analysis" tutorial. Continue to explore the CGWAVE model options, make more videos, or exit SMS at this point.