

SURFACE WATER MODELING SYSTEM

CGWAVE Analysis

1 Introduction

This lesson will teach you how to prepare a mesh for analysis and run a solution for *CGWAVE*. You will start with the data file *indiana.xyz* which contains a set of points that contain depth data from which a mesh will be created. To open the data:


1. Select File | Open.
2. Select *indiana.xyz* in the in the Data Files Folder for this tutorial and click the Open button.
3. The File Import Wizard dialog will appear. Click Next to proceed to Step 2 of the File Import Wizard.
4. Click Finish to open the file. (This wizard allows you to open data that may not have data in 3 columns of x, y, and z. Data in any number of columns in any order can be opened through the wizard).

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

- Select *Edit | Projection*, set the *Horizontal Local projection* and *Vertical Projection* units to *Meters*, and click *OK*.

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 explained in section 3. 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* data set in the *Project Explorer* (this makes the scatter module active).
2. Select *Data / Data Set Toolbox*.
3. Select *Wave Length and Celerity* under the *Coastal* menu on the left hand side
4. Change the *Output base name* to *Transition* and the *Period* at 20 seconds.
5. Click *Compute* and then *Done*.

Two new data sets will be created, one named *Transition_Wavelength* and the other named *Transition_Celerity*. These can be seen in the *Project Explorer*.

3 Creating a Size Function

The size function is created from the wavelength function. The size function is the function that 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 that was created in section 2 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*.
2. In the top middle section of the *Data Calculator*, highlight the function named *d2.Transition_Wavelength* and click the *Add to Expression* button. The letter that represents this function will appear in the *Expression* field.
3. Click the */* (divide symbol) in the bottom right section of the *Data Calculator*.
4. After the divide symbol, enter the number 5 (five) using the keyboard. (Note: This “5” represents that we will generate approximately five elements per wavelength. It is usually more appropriate to use a larger number of elements per wavelength (10). The smaller number is used here to allow faster execution of the model.)

-
5. In the *Output data set name* field, enter the name *size* and then click the *Compute* button.
 6. When the computation is completed, *size* will appear as a data set.
 7. Click the *Done* button to exit the *Data Calculator*.

A new data set named *size* that was created based on the *Transition_Wavelength* data set should appear in the *Project Explorer*.

3.1 Smooth Size Function

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

5. Select *Data | Data Set Toolbox*.
6. Switch to the *Spatial | Smooth data sets* option.
7. Select the scatter data set named “size”.
8. Change the *Element area change limit* to 0.5. This will modify the size function so the elements created by the size function are at most twice as big or half as small as their adjacent elements.
9. Enter the name “size smoothed 0.5” in the *Output data set name* and click the *Compute* button.
10. Click *Done* to exit the Smooth Size Data Set dialog.
11. If desired, the differences between the data set “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 data set.

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 data set. The active function of the active scattered data set will be used for this




operation. You should currently have only one scattered data set. To make the elevation function active:

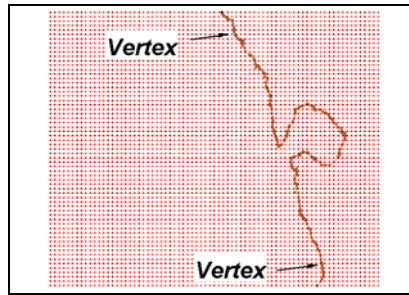
1. Select the elevation (Z) function in the *Project Explorer*. (This makes the elevation function current in the scatter module.)
2. Right click on the “default coverage” item and select “Type” then select “Models”. In the menu that appears select the type as “CGWAVE”.
3. Right click on the “default coverage” item and select “Rename”. Name this coverage “CGWAVE” and press enter.
4. Select the “CGWAVE” (left click). This will make it the active object.
5. With the coverage type set and the active scattered data set defined, you are ready to create the coastline. Select *Feature Objects | Create Coastline*. Enter 1.0 for the *Elevation*, leave the *Spacing* at 10.0, and click the *OK* button.

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

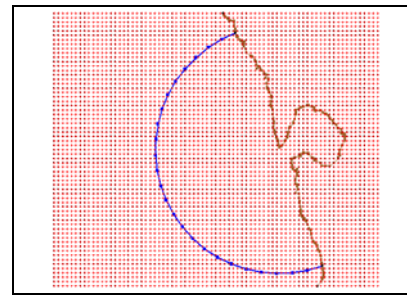
4.2 Creating the Domain

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. Choose the *Select Feature Vertex*  tool from the *Toolbox*.
2. Hold the *SHIFT* key and click on two vertices, one near each end of the coastline arc, as shown in Figure 1a.
3. Select *Feature Objects | Define Domain*. Select *Semi-circular*, and click *OK*. This creates a semicircular *Ocean* arc as shown in Figure 1b. If the domain is created on the wrong side of the coastline, it indicates that the coastline is oriented in the wrong direction. If this happens:
 - Choose the *Select Feature Arc*  tool from the *Toolbox*
 - Select the semi-circular arc and delete it.
 - Select the coastline arc and reverse its direction using the *Feature Objects/Reverse Arc Direction* command.
 - Select the two nodes remaining from the semi-circular arc using the *Select Feature Point*  tool and repeat the command in step 3 to create the domain. *Note: Make sure that the domain does not extend outside the extent of the scatterset. If it does delete the semi-circular arc and recreate it with points further within the interior of scatterset.*



(a). The coastline feature arc.



(b). The domain feature arc.

Figure 1 The indiana scatterpoint and feature data.

Now that feature arcs define the domain, a feature polygon must be created from the feature arcs. To create the polygon:

- Select *Feature Objects* | *Build Polygons*.


After this command is executed, polygons are formed from any set of arcs that form a closed loop. The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created. For this example, there should now be a single polygon made from the semi-circular ocean arc and the part of the coastline arc with which it intersects.

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 data set. The size function to be used in this example was created back in section 3. To set up the feature polygon for density meshing:

1. Choose the *Select Feature Polygons*  tool from the *Toolbox*. With this tool selected, double-click inside the polygon that defines the domain.
2. In the *Polygon Attributes* dialog, change the *Mesh Type* to *Scalar Paving Density* and press the *Scatter Options* button.
3. In the bottom left of the *Interpolation* dialog, turn on the *Truncate values* option and set the *Min* and *Max* to 10 and 10,000, respectively. This sets up a minimum and maximum size to be used when creating elements.

-
4. Select the data set “size smoothed 0.5” as the *Scatter Set to Interpolate From*.
 5. Click the *OK* button to get back to the *Polygon Attributes* dialog. (If a warning appears about the extrapolation value, click *OK*.)
 6. In the *Bathymetry Type* section, select *Scatter set*.
 7. Select the *Scatter Options* option under *Bathymetry Type*, make sure the function named *elevation (Z)* is highlighted in the *Scatter Set to Interpolate From* section and make sure the *Truncate Values* option is turned off. As mesh nodes are created, their elevation value will be assigned from the original water depth values that were read from the xyz file.
 8. Click the *OK* button twice to close both dialogs.

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.

5.2 Generating the Elements

Since there is only one polygon in this example, you are ready to have *SMS* generate the finite element mesh from the defined domain. To create the mesh:

1. Select *Feature Objects | Map->2D Mesh*.
2. Make sure that *Copy coverage before meshing* is toggled off.

After a few moments, the mesh will be created to look something like the finite element mesh in Figure 2.

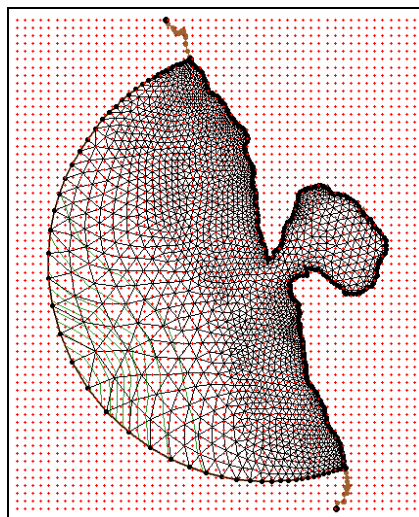



Figure 2 The completed finite element mesh.

At this point, the display is quite cluttered with all the data that has been created. Some of the visible objects can be hidden using the *Project Explorer*. To hide the scatter data and map data, uncheck the toggle boxes next to those objects in the *Project Explorer*.

To change the display settings use the *Display Options*  dialog by selecting the macro or right clicking on one of the model folders in the *Project Explorer* and choosing *Display Options*.

1. Switch to the *2D Mesh* tab and turn off everything except the *Elements*, *Contours*, and *Nodestrings*.
2. Click the *Contour Options* tab. Change the *Number of Contours* to 20. Change the *Contour Method* to *Color Fill*.
3. Click the *OK* button.

After the display is refreshed, you will see contours of water depth with the elements drawn on top of those. You can clearly see that as the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.


6 Model Control

When creating a *CGWAVE* model, the boundary conditions are wave amplitude, phase, and direction. To define these *incident wave conditions*:

1. Click on the *Mesh* object in the *Project Explorer* to make it active and select *CGWAVE / Model Control*.
2. Set the *Incident Wave Conditions: Direction = 30.0, Period = 20.0, and Amplitude = 1.0*.
3. In the *Solver Options* section, make sure the *Output Echo Frequency* is set to 1 and the *Maximum Iterations* is set to 500,000.
4. *CGWAVE* uses a 1-d file. The 1-d parameters must be set in this dialog and the 1-d depths extracted. By default, the ideal spacing is computed and the *# of 1-d nodes* is set to run to 1.5*radius away from the coastline. We can leave these defaults. Click on the *Extract Depths* button to extract the values.
5. Choose the elevation (*Z*) function to extract from and click *Select*.
6. Click the *OK* button to exit the *CGWAVE Model Control* dialog.

7 Renumbering

The mesh needs renumbering before being saved. To do this:

-
1. Select the *Select Nodestring*  tool from the *Toolbox*.
 2. Select the blue ocean nodestring by clicking in the box on the nodestring.
 3. Select *Nodestrings / Renumber*.


8 Saving the CGWAVE Data

CGWAVE uses a geometry file and the 1-d 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 1-d 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...* and enter the name *indianaout*.
2. Push the Save button.

9 Running CGWAVE

CGWAVE can be run from *SMS*. To run *CGWAVE*:

1. Click on the *Mesh* object in the *Project Explorer* to make it active.
2. Select *CGWAVE / Run CGWAVE*.
3. *SMS* saves the location of the *CGWAVE* executable as a preference. If this preference is defined, the model will launch. If the preference is undefined, *SMS* shows a message that the *cgwave* executable is not found. In this case click the *File Browser*  button to find the *CGWAVE* executable and Click the *OK* button to run the model.

For this simulation, *CGWAVE* should finish in a couple of minutes. When the simulation is finished, the file *indianaout.cgo* will contain the *CGWAVE* solution data. This file will automatically open when you click *Exit* as long as you have the *Load Solution* box checked in the *CGWAVE Model Wrapper*. 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, you can still approximate the breaking by selecting the option to break the waves as you read the solution file. As you open the file, *SMS* will translate the wave output into data sets 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.

NOTE: If *CGWAVE* does not run, you may have an older version of *CGWAVE*. Open the *indianaout.cgi* file in a text editor and change the first few lines from:

```
%number of characters in title &  
%number of terms in the series &  
%number of iterations for checking convergence &
```

```

%maximum iterations &
%maximum iterations for nonlinear mechanisms &
%maximum connectivity &
      12      35      1      500000      1000      8
to:

%number of characters in title &
%number of terms in the series &
%number of iterations for checking convergence &
%maximum iterations &
%maximum connectivity &
      12      35      1      500000      8

```

10 Post Processing

Now that CGWAVE has run properly and the solution files loaded into SMS, we can look at the different solutions that were generated. Data 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

It might also be very helpful to change the display options so that it might be easier to view the waves transitions. To do so,

1. Select *Display / Display Options*. Select 2D Mesh.
2. Toggle everything off except for *Functional Surface*.
3. Click on the *Options* button next to Functional Surface.
4. Set the Z Offset to Display surface above geometry
5. Under Display Attributes, select *Contour surface* and click on the Options button.
6. Click on the *Color Ramp* button to open the color options dialog.
7. Change the *Palette Method* to Intensity Ramp and change the color to blue. You might also want to reduce the spectrum of the current palette so that the colors available do not include too much white or black.
8. Click OK four times to change the display.
9. Select the 'Particle Velocity @ Bed' vector mesh and the 'Sea Surface Elevation' scalar mesh. The display screen should now look like Figure 3.

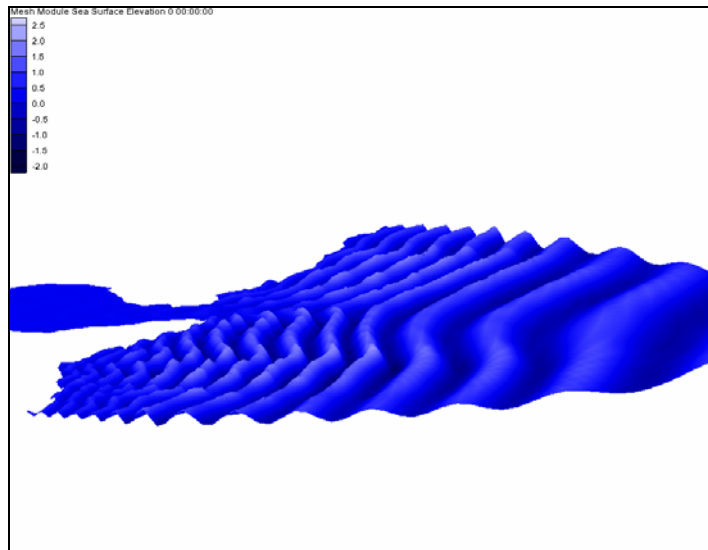


Figure 3 Functional surface.

10.2 Film Loops

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

To create film loops,

1. Select *Data / Film Loop*. The Film loop setup dialog will open.
2. Toggle on *Create AVI File* and click on the *Folder* icon. Make sure that the film loop is saved in the same folder where all the files are and rename the AVI file as “CGWAVE”.
3. Click Save.
4. Click Next and *Specify the number of frames* as 50.
5. Click Next and change the *Quality* to 90. That will allow for less pixilated frames and smoother film loop.
6. Click Finish to create the film loop. The AVI dialog should open shortly. If you wish to embed the film loop, the file will be in the folder where the SMS project files are saved. The file will have an ‘.avi’ extension. You may open that file in any video viewing software such as Windows Media Player.

11 Conclusion

This concludes the CGWAVE tutorial. You may continue to make more videos and look through the other meshes created or you can exit SMS at this point.