

***LESSON* Error! Reference source not found.**

CGWAVE Modeling

10.1 Introduction

This workshop will illustrate the functionality of *CGWAVE* and teach you how to use the interface to design and build a grid, and then run the model and interpret results. You will need the following files:

- rectangular.cgi
- breakwater.spr
- jetty.spr
- ideal_inlet.xyz
- ideal_inlet.cst
- morro.xyz
- morro.cst

10.2 Experimenting with the Model

You will start this lesson by working with an existing mesh to experiment with the capabilities of *CGWAVE*. To begin this workshop select *File | Open* and open the file “rectangular.cgi”. This model is of an idealized, flat bottomed rectangular harbor opening on the ocean. *CGWAVE* computes the wave fields in coastal zones, around islands, and in harbors. It includes the capability to simulate how waves reflect off of different surfaces, and changes in bathymetry. In this section we will experiment with some of the principal parameters and examine how they affect model results.

The first parameter we will investigate is the coefficient of reflection for different portions of the coastline. The perimeter of the domain is divided into nodestrings to represent the boundary conditions for *CGWAVE*. Two types of boundary conditions exist including open ocean and coastline. Each section of coastline and the ocean interface are stored in *SMS* as nodestrings. Each coastline section can be assigned its own reflection coefficient to tell *CGWAVE* how waves will reflect from that portion of shoreline. Values for reflection coefficients vary between 0.0 and 1.0 and typical values include:

Reflection Coefficient	Shoreline Condition
1.0	Vertical wall
0.4-0.5	Large Rocks
0.15-0.25	Sandy Beach

10.2.1 Running CGWAVE

To assign initial values for reflection coefficients:

1. Click on the *Select Nodestring*  tool. Four nodestring handles (boxes) will appear over the nodestrings in this model. Three of the nodestrings are colored brown. These are the coastline nodestrings. The blue nodestring represents the ocean.
2. Select all of the coastline nodestrings by dragging a box around them.
3. Select the *CGWave | Assign BC* command. Make sure that *Coastline* is selected and set the *Reflection Coefficient* to 1.0 (wall).
4. Click OK.

With the boundary conditions for the model set, we can set the model controls. To do this:

1. Select *CGWave | Model Control*. The *Model Control Dialog* will appear.
2. Set the *Incident Wave Conditions* as follows: *Direction: 270.0; Period: 6.0; and Amplitude: 0.2*.
3. Make sure the *Bottom Friction* and *Wave Breaking* toggles are off.

4. Click OK.

Now we can save and run the model. To do this:

1. Select the *File | Save As...* command.
2. Make sure that *Save as type* is set to *Project Files (*.spr)*.
3. Enter “*run1.spr*” for the *File Name* and click the *Save* button.
4. Now to run *CGWave*, select the *CGWAVE | Run CGWAVE* command. The Run Model dialog appears.
5. Click the File Browser icon  and select the appropriate executable for the machine you are working on (either *cgw_pcAMD.exe* or *cgw_pcINTEL.exe*).
6. Once an Executable is found, click *OK*. A separate DOS prompt will appear and the model will run in a second or two. When it has finished the window will disappear.

Now the solution file (*run1.cgo*) can be read. Read it in using *File | Open*. As *SMS* reads the output file it creates several data sets. These include steady state data sets representing the wave height, wave phase, and wave direction along with the sea surface and particle velocities and pressures at the water surface, midpoint of the water column and bottom of the ocean. These functions are grouped into a solution set called “*run1.cgo (CGWAVE)*”. In addition *SMS* creates transient data sets in a solution set called “*run1.cgo_anim (CGWAVE)*” to allow animation of the sea surface over a wave period. A scalar data set named *Wave Surface* is created along with a vector data set named *Wave Velocity*. To view the computed wave surface:

1. Click on *Display|Display Options*.
2. In the *2D Mesh* tab turn on the *Vectors* toggle.
3. In the *Contours* tab, change the *Contour Method* to *Color Fill and Linear*.
4. Click on the *Fill color Options* button.
5. In the *Color Options* dialog, change the *Palette Method* to *Hue Ramp* and click *OK*.
6. In the *Vectors* tab, set the *Shaft length* option to *Define Min and Max Length* and enter a *Max Length* of 45 pix.
7. Click *OK* to exit the Display Options Dialog.

The image updates to show the wave surface contoured with vectors showing the wave velocity and direction. An animation can be used to view how these values change through a wave period. To generate this animation:

1. Select *Data | Film Loop...*
2. In the *Film Loop Setup* dialog, make sure that *Create New Filmloop* is on and *Scalar/Vector Animation* is selected.
3. Click on the *File Browser* icon , enter *run1.avi* as the *File name*, and click *Save*.
4. Click *Next*.
5. In the *Time Step Options* page, make sure that *Scalar* and *Vector Data Set* toggles are both on. Leave the time steps at default values.
6. Click *Next*.
7. Click *Finish* to create the film loop.

The film loop will take about a minute to create. Watch the animation to get an idea of how the waves are moving and then close the film loop window.

10.2.2 Varying Run Parameters

Now you have stepped through the process of how to set the parameters, run *CGWAVE* and view the results. The parameters for this run are shown in the *Run 1* row of the table below. Repeat the procedure, changing the conditions to match those specified in each row. Generate an animation for each run as it is made to view the wave conditions. (Changes from one row to the next are marked in bold text.)

Run #	Reflection Coefficients			Wave Conditions			Comment
	Left	Harbor	Right	Direction	Period	Height	
1	1.0	1.0	1.0	270.0	6.0	0.2	Base Run
2	1.0	1.0	0.1	270.0	6.0	0.2	Absorbing beach
3	1.0	0.1	0.1	270.0	6.0	0.2	Absorbing harbor
4	1.0	1.0	1.0	225.0	6.0	0.2	Changing direction
5	1.0	1.0	1.0	270.0	6.0	0.1	Changing height
6	1.0	1.0	1.0	270.0	9.0	0.1	Long Wave
7	1.0	1.0	1.0	270.0	3.0	0.1	Short Wave

Make notes of any observations you find interesting for each run so that we can discuss them later.

10.2.3 Changing the Mesh

Changing run parameters shows how the model responds to different wave conditions. More significant changes are effected by viewing how the model

responds to changes in the geometry. This could include domain shape, internal features, or changes in bathymetry. To illustrate this, we have prepared two variations of the rectangular harbor. The first includes a breakwater in front of the harbor (breakwater.spr) and the second has a jetty on the right side of the harbor (jetty.spr). For each, read in the project, run *CGWAVE* and generate an animation of the results. If time permits at the end of this lesson, you want to come back and modify wave conditions or beach parameters and rerun *CGWAVE*.

10.3 Idealized Inlet

Now that we have examined the effects of some of the model parameters, the next step is to learn how to prepare a mesh for analysis. You will start with the data file *ideal_shin.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 | Delete All*. This clears away all the data in *SMS* from the previous section.
2. Select *File | Open*.
3. Select *ideal_shin.xyz* in the *Lesson10* directory and click the *Open* button.
4. The *File Import Wizard* dialog will appear. The settings are defaulted for opening the file. Push *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 will be created.

10.3.1 Creating a Wavelength Function

The first step in creating a mesh for *CGWAVE* is to create a wavelength function. The wavelength at each point is calculated from the depth values which were read in from the scattered data file. We want to know the wavelength, because we are trying to define the shape of the waves (the sea surface) in the domain. To do that, we need more densely packed nodes where the waves are shorter. To create the wavelength function:

1. Make sure you are in the *Scatter*  module.
2. Select *Data | Create Data Sets*.
3. Turn off everything except the *Transition Wavelength/Celerity* option. (The *Coastal* option must be on to access the *Transition Wavelength/Celerity* option.)
4. Leave the function name as *Transition* and enter a *Period* of 8 seconds.

5. Click the *OK* button.

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

10.3.2 Creating a Size Function

Now that we know the wavelength we can create a size function. The size function is a measure of how long an element edge should be at any location in the domain. As *SMS* creates elements, it will use the size function to guide the process.

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 the previous section contains the length of the waves at each location in the domain. If we used this function as a size function, the resulting mesh would have approximately one element per wavelength. This would give no definition to the sea surface. In this example, we will create a size function with three elements per wavelength. This would give only a rough approximation to the sea surface, but will illustrate the procedure. Normally, you will want to create at least 6 elements per wavelength. The size function will be created by scaling the wavelength function. To do this:

1. Select *Data | Data Calculator*.
2. In the top left section of the *Data Calculator*, highlight the function named *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 middle section of the *Data Calculator*.
4. After the divide symbol, enter the number 3 (three) using the keyboard.
5. In the *Result* field, enter the name *size 3* and then click the *Compute* button.
6. When the computation is completed, *size 3* will appear as a data set.
7. Click the *Done* button to exit the *Data Calculator*.

A new data set named *size 3* is created which has values equal to one third of the *Transition_Wavelength* data set.

10.3.3 Defining the Domain

In *CGWAVE*, the domain is defined by a coastline and an ocean boundary. The ocean boundary can be circular, or semi-circular in shape. Circular domains apply to islands entirely surrounded by ocean. In *SMS*, an arc or string of arcs are used to define the

coastline. Based on the coastline, additional feature arcs are generated to define the ocean boundary.

Opening the Coastline File

SMS can automatically create a coastline at a specific elevation or water depth from a scattered data set. However, for this tutorial, you will open a file that contains a coastline that has already been created. To open the coastline file:

1. Make sure you are in the *Scatterpoint*  module.
2. Change the function in the *Scalar* field in the top *Edit Window* (just beneath the menus) from *size* to *elevation*.
3. While still in the *Scatterpoint*  module, select *Display | Display Options*. Under the *Scatter* tab, at the top left there is a box labeled *Visible*, turn the *Visible* toggle off and click the *OK* button.
4. Select *File | Open*.
5. Select *ideal_shin.cst* in the *Lesson10* directory and click the *Open* button.
6. A dialog will appear asking you what type of coverage should be created. Select *CGWAVE*. This creates a new *CGWAVE* coverage which contains the coastline arc read in from the file.

After a few seconds, the display will refresh with the coastline displayed.

Creating the Ocean Boundary

From the coastline, you can now create the ocean boundary enclosing the domain. This model will use a semi-circular domain that intersects with the coastline. To create the domain:

1. Choose the *Select Feature Node*  tool from the *Toolbox*.
2. Hold the *SHIFT* key and select the two nodes at either end of the coastline arc.
3. Select *Feature Objects | Define Domain*. Select *Semi-circular*, and click *OK*. This creates a semicircular *Ocean* arc.

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*.

This command builds polygons 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 coastline.

10.3.4 Creating a 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 *Scalar Density Paving* type.

Polygon attributes

When using scalar density paving, *SMS* determines element sizes from a *size function* in a scattered data set. The size function was created back in section 10.3.2. To set up the feature polygon for scalar density paving:

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 Density Paving* and press the Scatter Options button.
3. In the bottom left of the *Scatter Options* dialog under Mesh Type, turn on the *Truncate values* option and set the *Min* and *Max* to 10 and 1000, respectively. This sets up a minimum and maximum size to be used when creating elements.
4. Select *size3* in the Data Set window.
5. Click the *OK* button to get back to the Polygon Attributes dialog.
6. In the *Bathymetry Type* section, select the *Scatter Set*.
7. Select the *Scatter Options* dialog under Bathymetry Type, highlight the function named *elevation* in the Data Set window as the scatter function 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 pts file.
8. Click the *OK* button 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. For CGWAVE, the most common reason for extra polygons to exist is the existence of islands in the domain. Islands are assigned a meshing type of *None* since no elements are desired in those polygons.

Map->2D mesh

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:

- Select *Feature Objects* | *Map->2D Mesh*.

This step may take several minutes to complete. Once the mesh is generated, the display is quite cluttered with all the data that has been created. By changing some display settings, the display can become less cluttered. To change the display settings:

1. Switch to the *Map*  module and select *Feature Objects* | *Coverages*. Turn the *Visible* toggle off and click the *OK* button.
2. Switch to the *Mesh*  module and select *Display* | *Display Options*. Under the *2D Mesh* tab turn off everything except the *Elements* and *Contours*.
3. Click the *Contours* tab. Change the *Number of Intervals* to 20. Change the *Contour Method* to *Color fill*. Set the color ramp to a *Hue Ramp*.
4. Click the *OK* button.

The display will refresh and you will see contours of water depth with the elements drawn on top of those. Since the mesh contains over 150,000 elements, the contours can still be seen, but most of the bay is black from the displayed elements. If you zoom in, you can see that as the water depth decreases, so does the element size.

10.3.5 Setting Model Parameters

Wave Conditions

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

1. In the *Mesh*  module, select *CGWAVE* | *Model Control*.
2. Set the *Incident Wave Conditions*: *Direction* = 90.0, *Period* = 8.0, and *Amplitude* = 1.0.
3. Set the *Number of Iterations* to 200,000 and the *Maximum Iterations* to 5,000.
4. *CGWAVE* can use a 1-d wave transformation to bring the wave components to the mesh from deeper water. In this case, *SMS* save a series of depth values to a file. The 1-d parameters can be set in this dialog. By default, the *1-d Spacing* is set to $1.25 \times \text{radius} / 100$, with the number of nodes set to extend the depth values to the edge of the scatter set. In this case, set the # of *1-d nodes* to 2,000, and set the *1-d Spacing* to 2.0.

5. Click the *OK* button to exit the *CGWAVE Model Control* dialog.

Renumbering

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

Select the *Select Nodestring*  tool from the *Toolbox*.

Select the blue ocean nodestring by clicking in the box on the nodestring.

Select Nodestrings | Renumber and push OK.

Running the Model

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 as*, make sure the Save as type is set to Project Files, and enter the name *idealInlet.spr*.
2. Push the *Save* button.

CGWAVE can be run from *SMS*. To run this model on a Pentium 3, 500 MGHZ machine will take approximately 30 minutes. Therefore we will not run it at this time.

1. Select *CGWAVE | Run CGWAVE*.
2. Click on the  select file icon and find the *CGWAVE* executable. Click the *OK* button.

For this simulation, *CGWAVE* should finish in several of minutes. When the simulation is finished, the file *idealInlet.cgo* will contain the *CGWAVE* solution data.

10.4 Morro Bay

You will now read in a real world simulation to see the results. The time involved in generating this mesh and in running *CGWAVE* would be greater than are allow, but bathymetric data (.xyz) and coastline data (.map) are included in your directory for you to experiment with on your own.

For now, read in morro.cgi and morro.cgo and examine the solution.

10.5 Conclusion

You are now familiar with basics of *CGWAVE*. If you have time, you may go back and experiment with other wave conditions and options or you may exit the program. To quit *SMS* at this point:

1. Choose *File | Exit*. If prompted to confirm, click the *Yes* button.