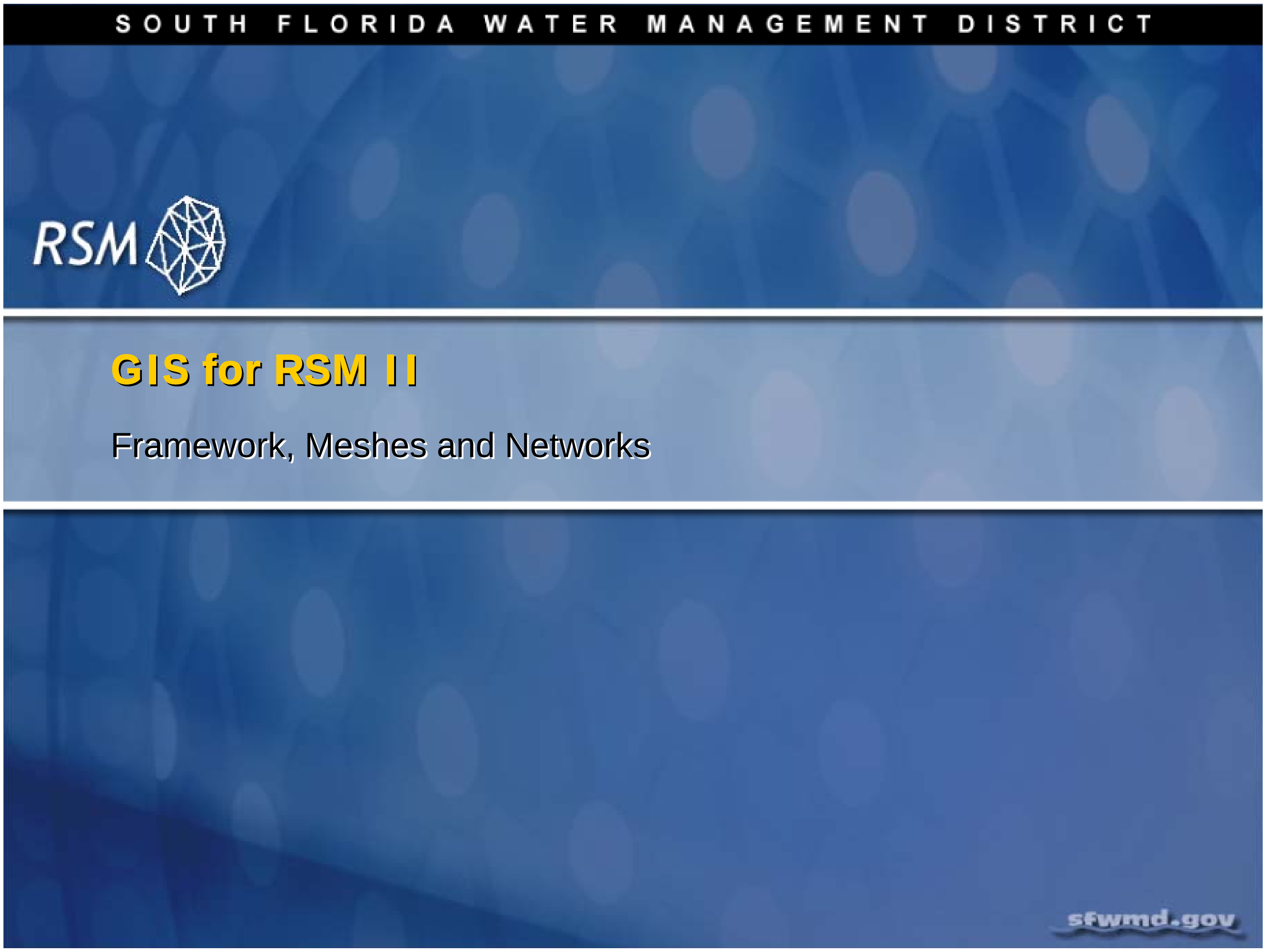


Lecture 6: GIS for RSM II – Framework, Meshes and Networks

This session discusses building and refining a mesh and a network using the Global Information System (GIS) and the Groundwater Modeling System (GMS) software for the Regional Simulation Model.

NOTE:

Additional Resources

Lal, A.M.W., 2000. Numerical errors in groundwater and overland flow models. Water Resources Research, 36(5), pp. 1237-1247. (slides 23, 27, 30)

Lal, A.M.W., 2006. Determination of multiple aquifer parameters using generated water level disturbances. Water Resources Research, 42, WO3429, doi:10.1029/2005WR004218. (slide 31)

Lal,.A.M.W., 1998a. "Performance comparison of overland flow algorithms", ASCE J. of Hydraulic Eng., 124(4), April 1998, pp. 342-349. (slide 32)

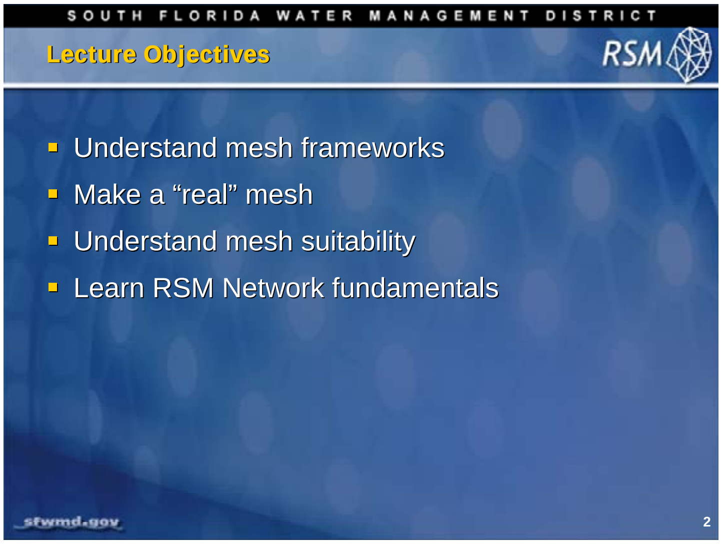
These three references can be found in the labs/lab6_GMS directory:

`WRR_ERR.pdf`

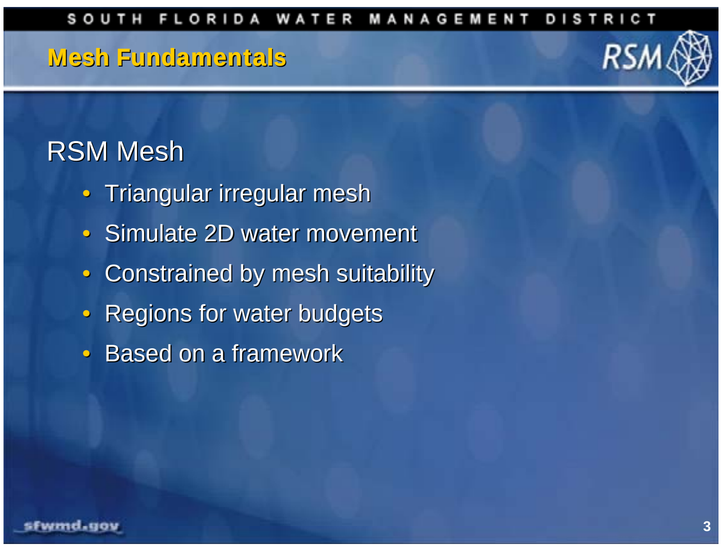
`AQUIFER_PARAMETER_0.pdf`

`PERFORMANCE-1998.pdf`

A video file of lab exercise 6.2.1, `lab6.wmv`**, has been proved to allow students with no access to ArcMap the ability to see how that software would be used.**

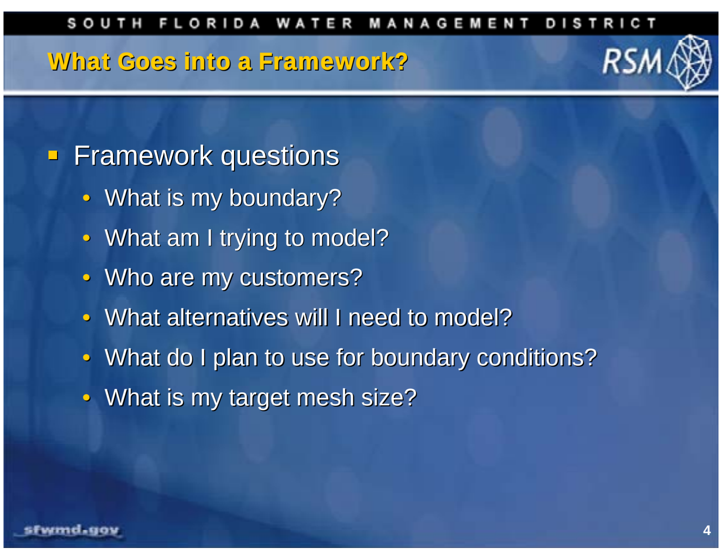


The irregular triangular mesh is the basic component of the Regional Simulation Model (RSM) for numerical 2D simulation of water movement. An example was provided in Figure 5.4 of Lab 5 that shows how the size and shape of the triangles can be specified to best represent features in the model.



To accurately simulate water movement using the diffusive wave equation:

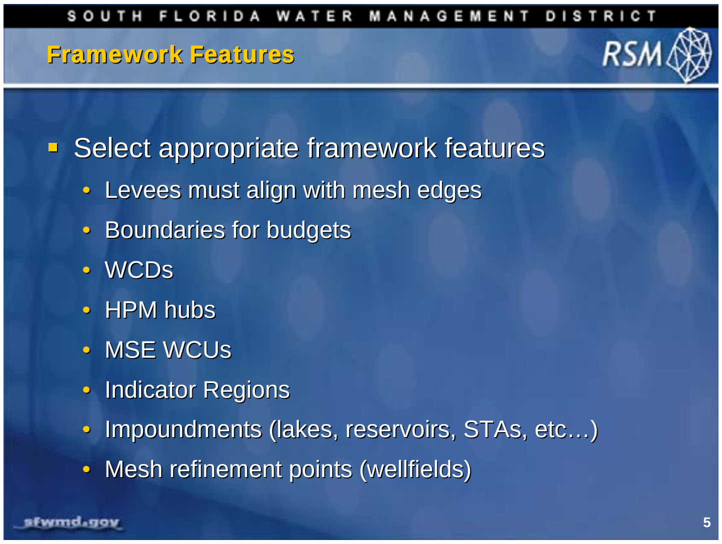
- The size and shape of the cells are constrained by the characteristics of the numerical method
- The cells should be acute (size depends on the model timestep, topography, groundwater slope and flow resistance)
- The cells form the boundaries for determining water budgets by basin, service area or county
- The shape of the mesh is constrained by no-flow boundaries such as levees, major roads and catchment boundaries



The mesh framework determines the shape of the mesh. The size and shape of the cells follow the location of the framework lines and the density of the vertices on the framework lines. And, the framework lines depend on the objectives of the modeling project:

- Is it a regional or subregional model?
- Where are the necessary planning boundaries and the location of features for alternative plan formulations?

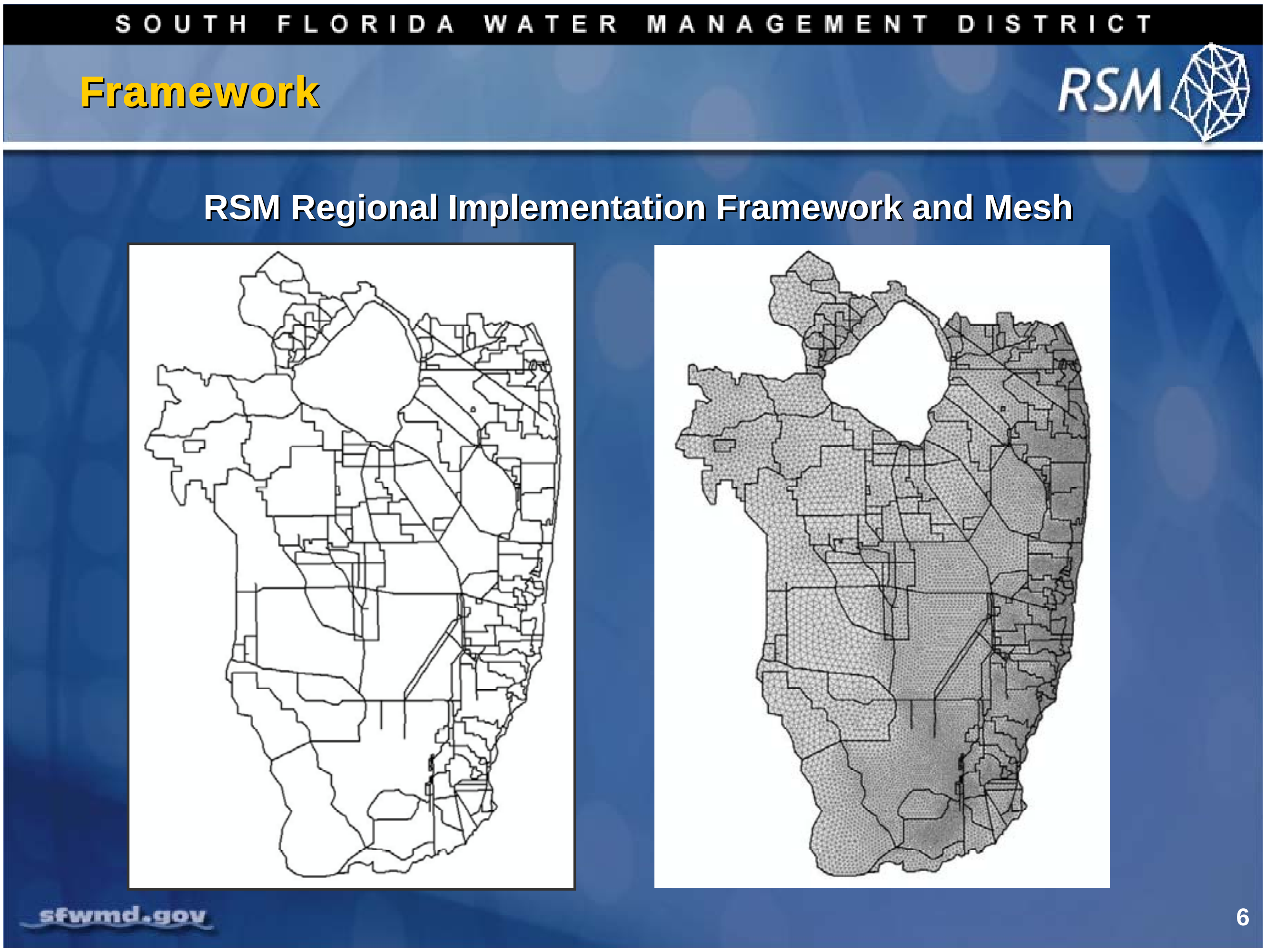
Additionally, since run time is a major consideration, selection of the appropriate



mesh size is important.

Specifically, the mesh framework must conform to the features listed on the accompanying slide, "Framework Features".

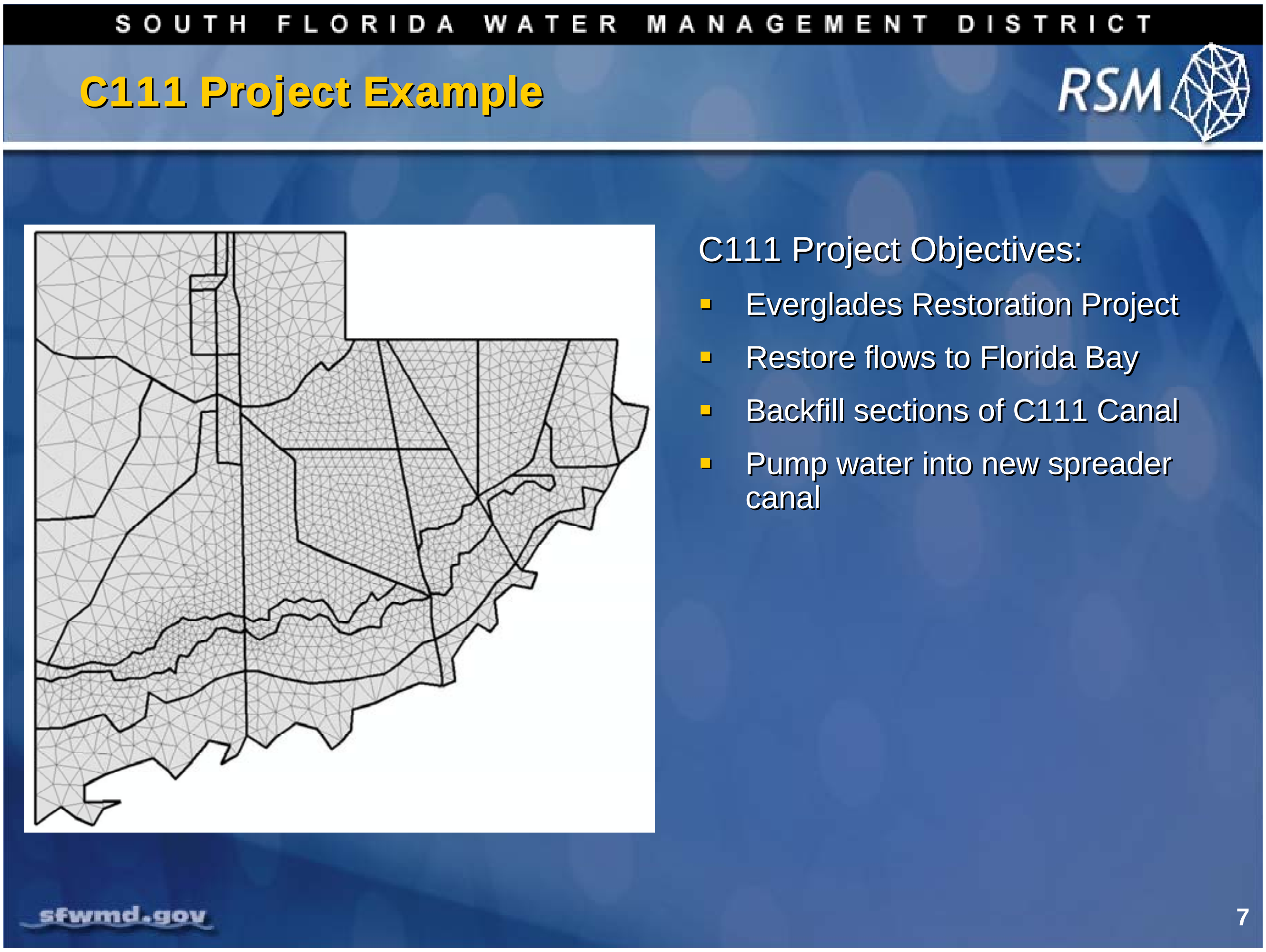
Transecting flow gages and modifying the mesh with additional features will provide cell and node alignments with the best configuration for evaluating the model output using water budget analysis.



A standard framework has been developed for the RSM subregional model implementations. This framework contains basin boundaries and the major levees.

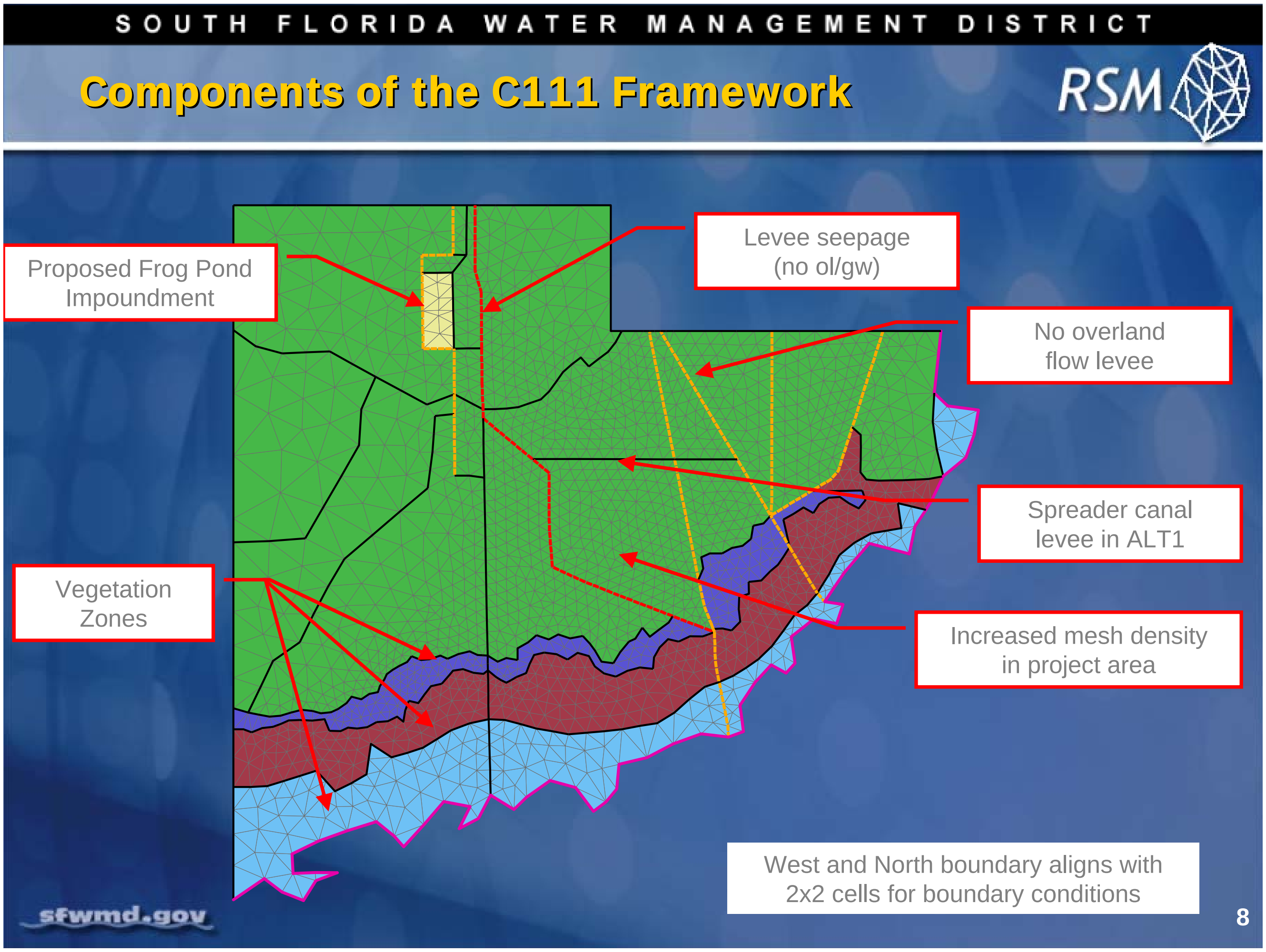
It does not contain all of the features that may be appropriate for any specific RSM implementation, but it provides a good starting point.

Furthermore, the framework does not follow the basin boundaries exactly. In some areas, there are many fine details which are too small for mesh discretization.



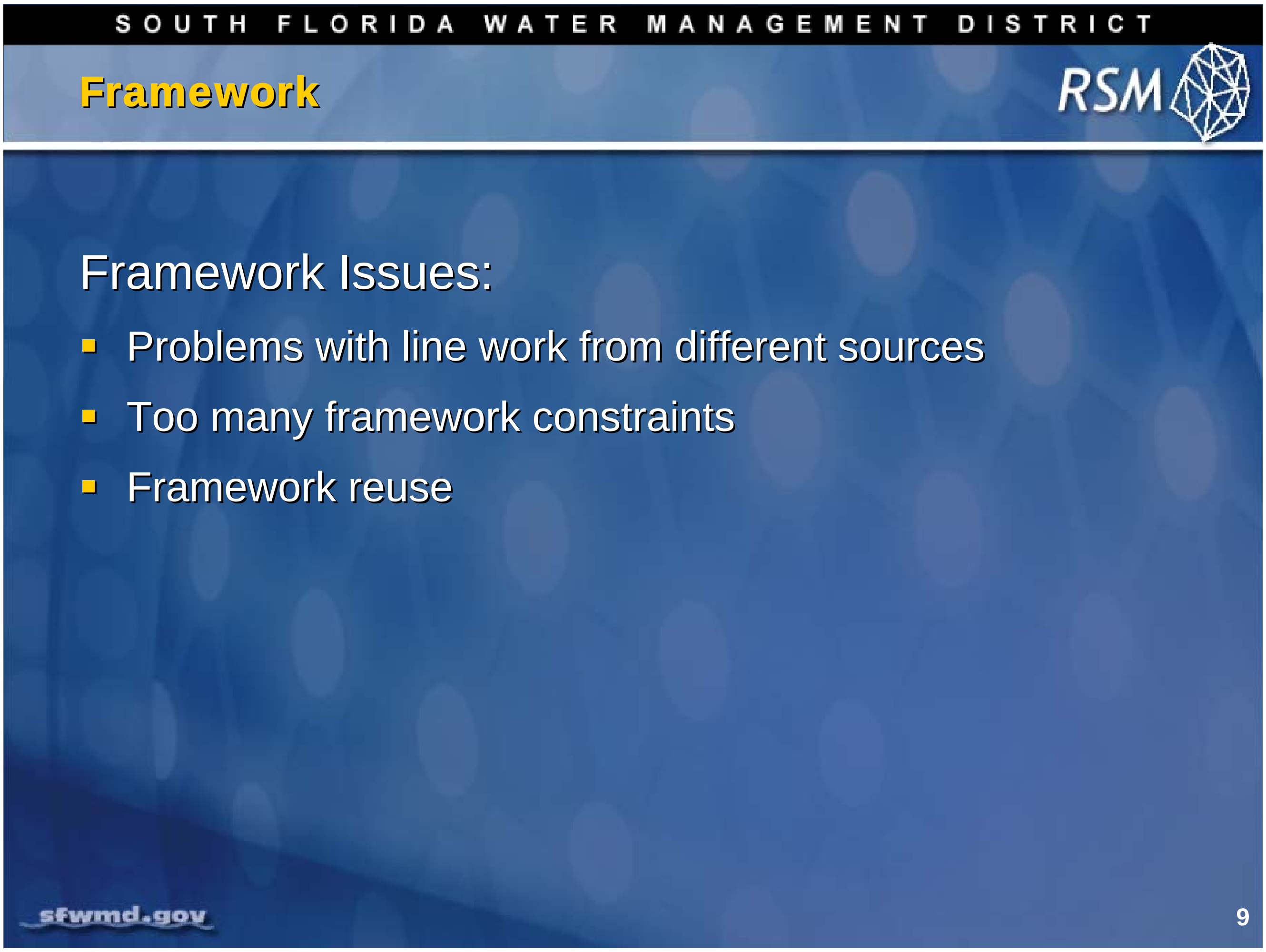
The C111 Spreader Canal RSM is an example of a modeling project created to evaluate the impacts of creating a new spreader canal on overland flow and groundwater flow to Florida Bay.

In addition to levees and basin boundaries, the mesh framework includes boundaries of various vegetation communities along the coast. The density of the cells reflects the need for hydrologic details as well as the limits caused by high gradients or lack of gradients.



The framework for the C111 RSM Project consists of the following components:

- Vegetation zones
- Levees
- Locations of the project features (new impoundments and spreader canal)

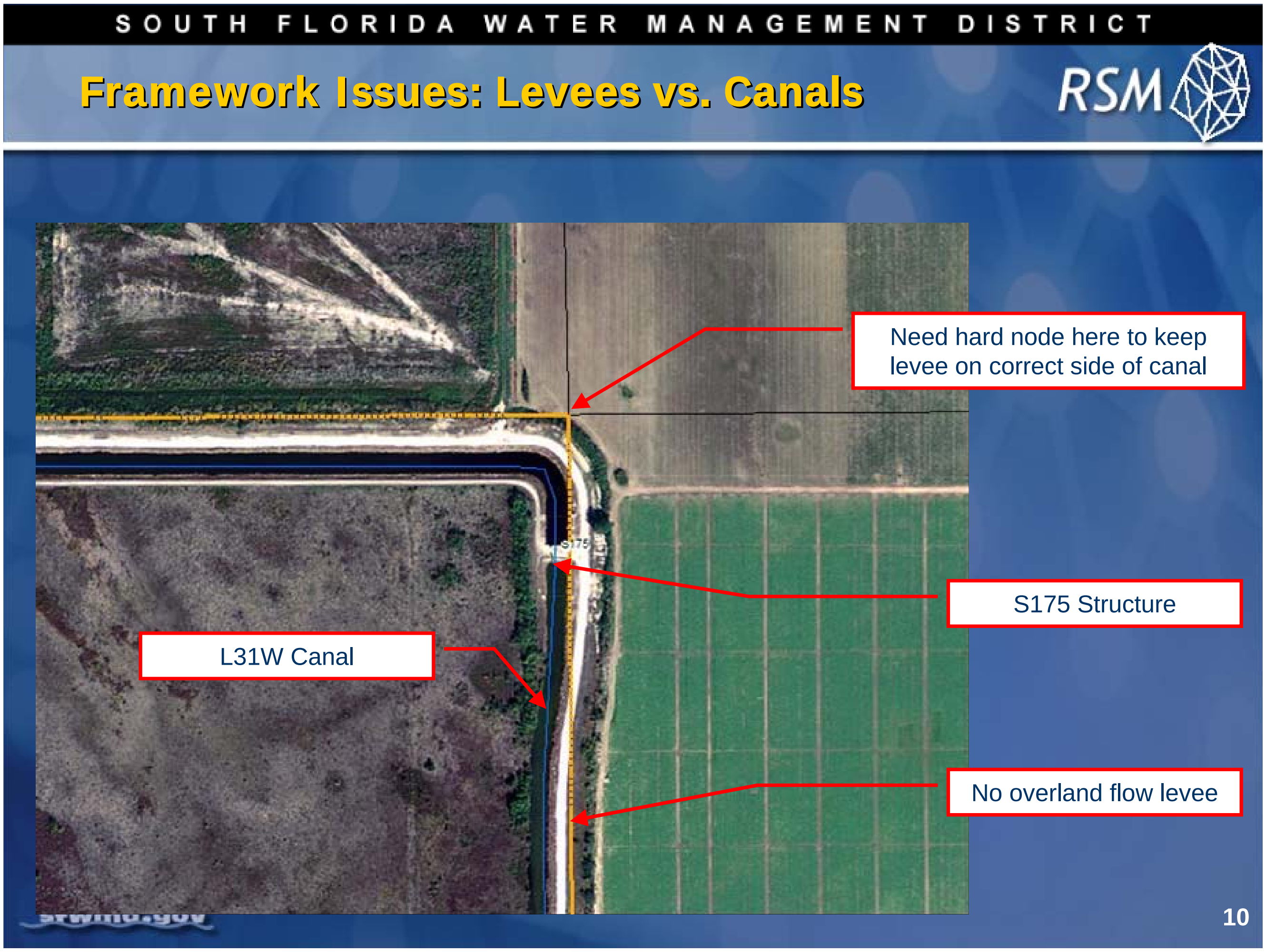


Issues concerning the use of the framework for creating a suitable mesh exist. The potential for a substantial problem occurs when using additional line work from local sources with the standard SFRSM framework.

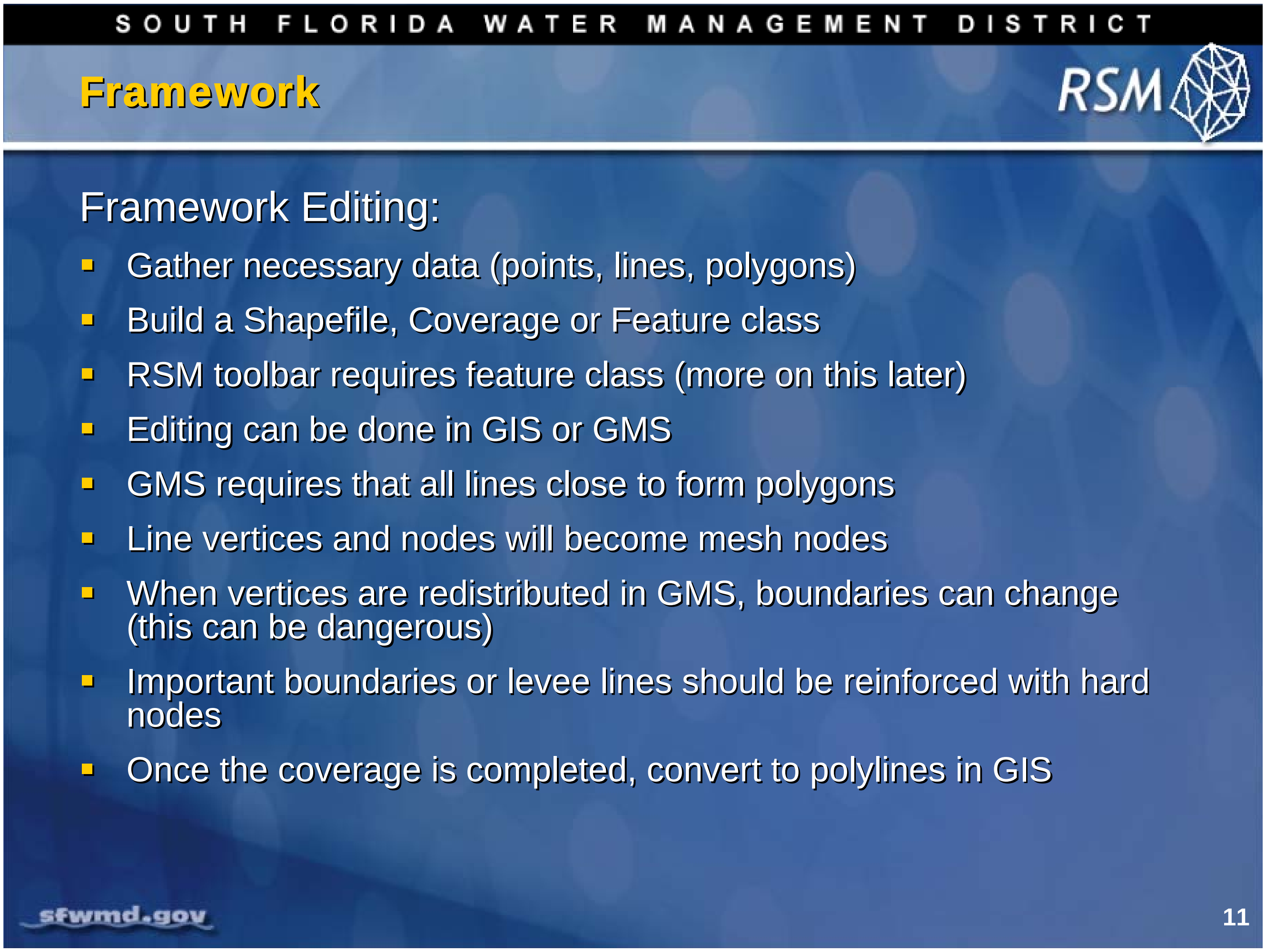
Local line work is commonly generated by local stakeholders for defining regions of interest in the model domain. And, line work from different organizations can have significant inconsistencies.

There may also be a problem using too many framework constraints. In this case, poorly dimensioned triangles could result in the mesh, requiring considerable work to fix.

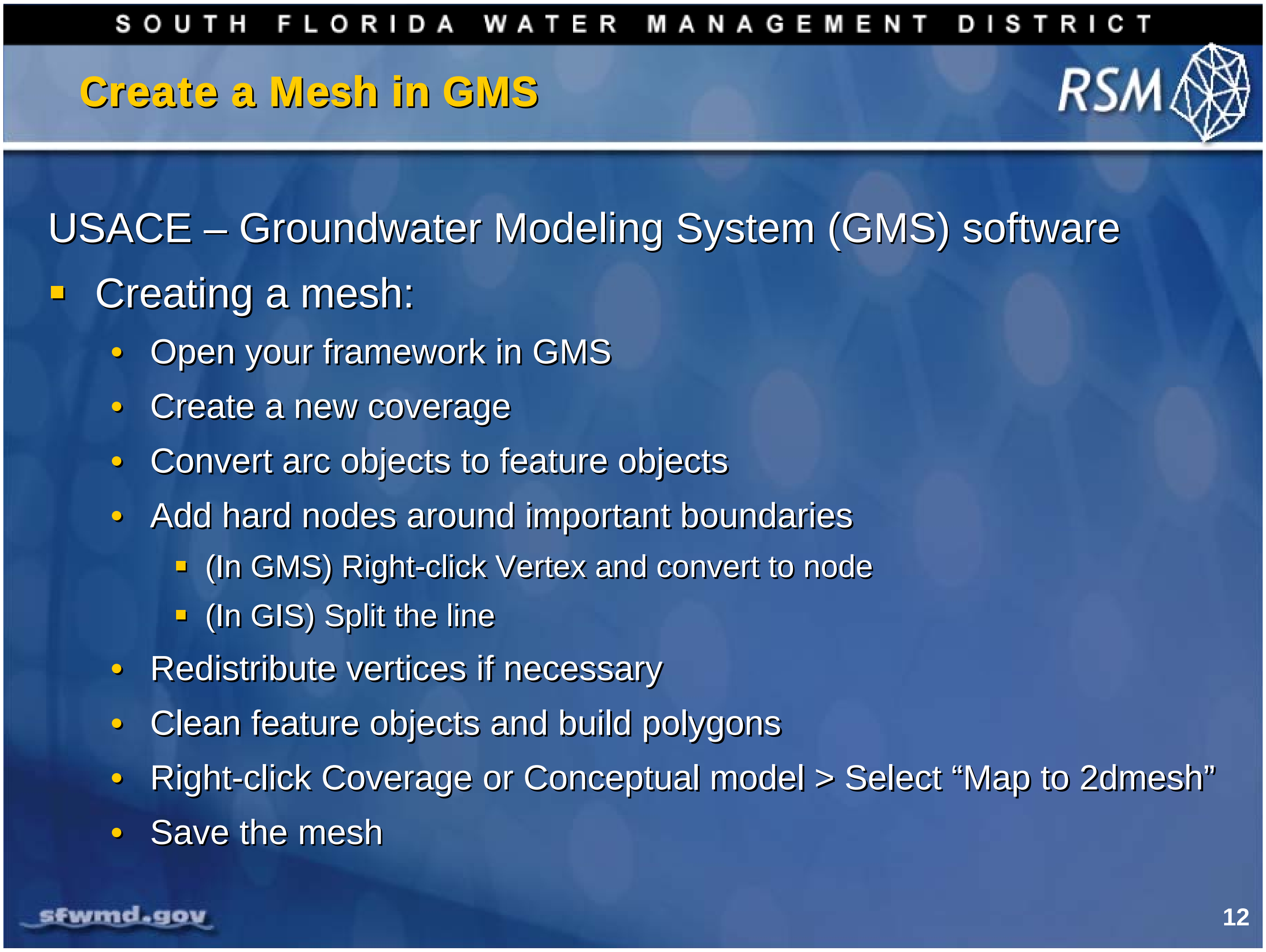
Each framework should be developed and archived so it can become an addendum to the standard SFRSM framework for reuse.



Line work in the mesh framework is frequently close and must be carefully checked. This slide presents an example of the location of a canal in close proximity to a levee. With high quality aerial photography it is easy to check the location of the line work. Since the features are close to one another, it is important to exercise care to ensure the levee framework line does not cross the canal line work. If the lines cross, leakage of water from the canal into the adjacent basin will occur.

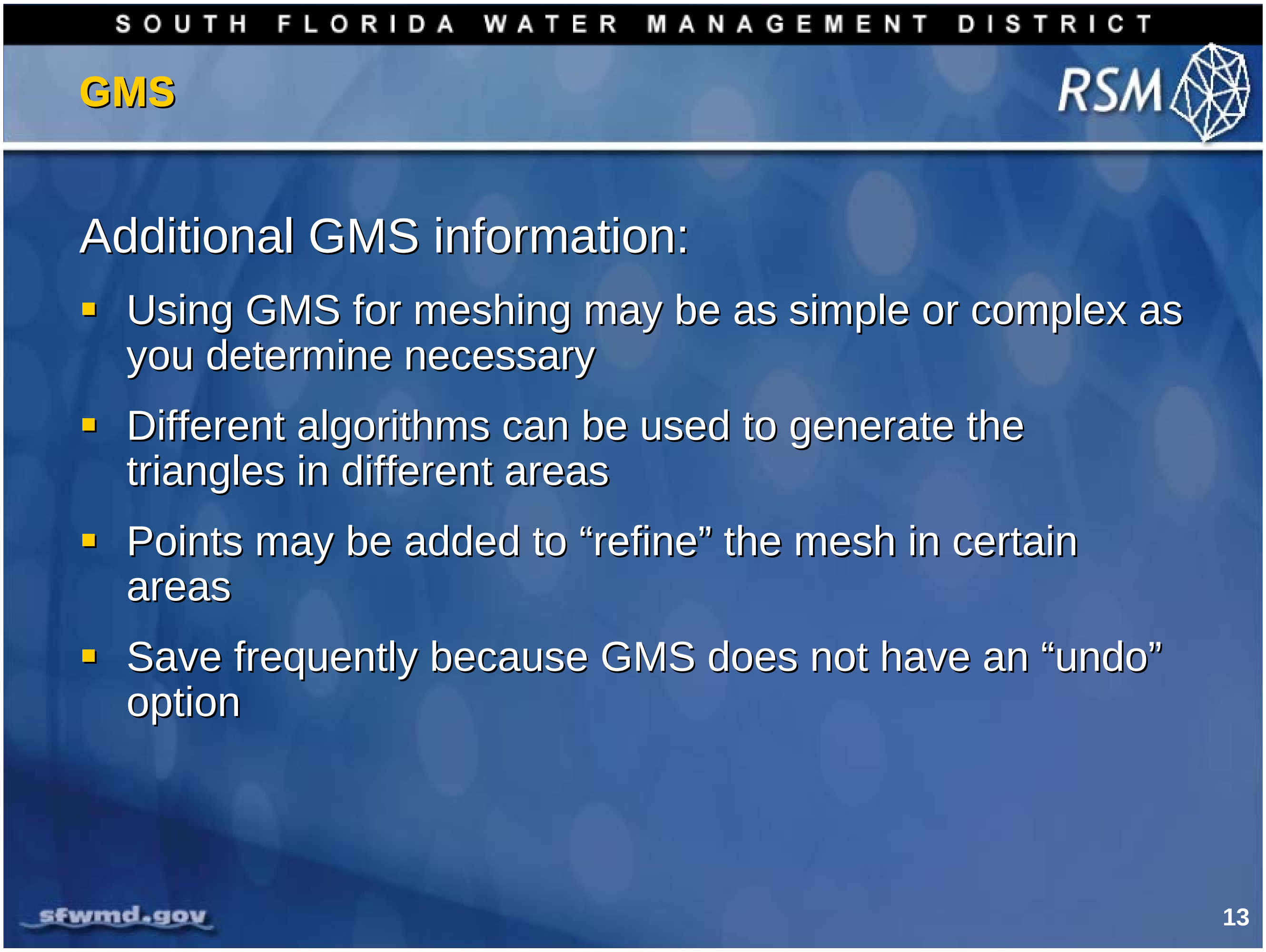


To create the appropriate framework, it is necessary to follow the steps outlined in the slide above. The final framework feature class created in GIS will contain polylines and not polygons.



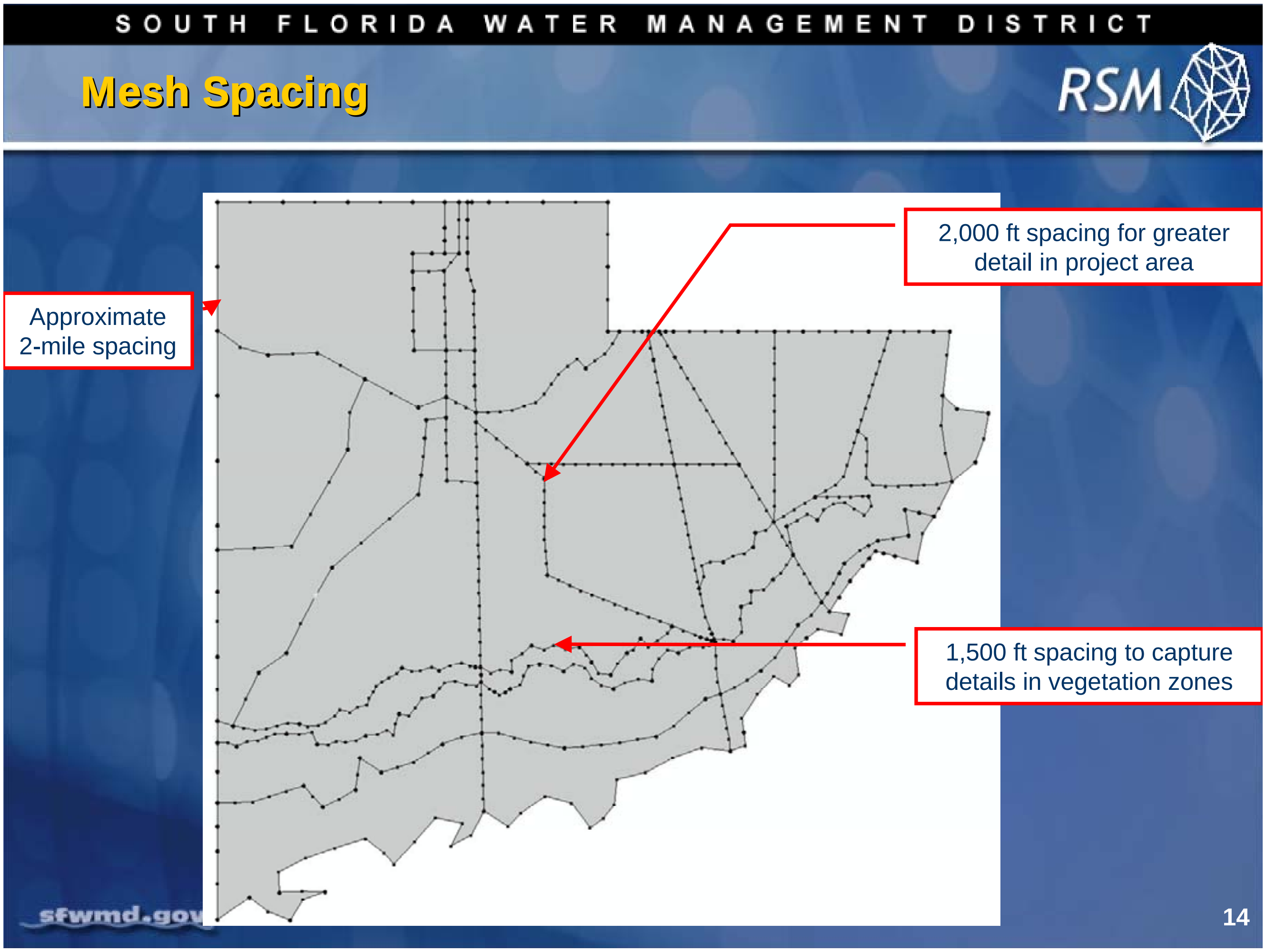
The process outlined in this slide is used in Hydrologic and Environmental Systems Modeling to create a mesh.

The modelers at the South Florida Water Management District use the mesh creation capabilities available through the Groundwater Modeling System (GMS) software developed by the U.S. Army Corps of Engineers (USACE).

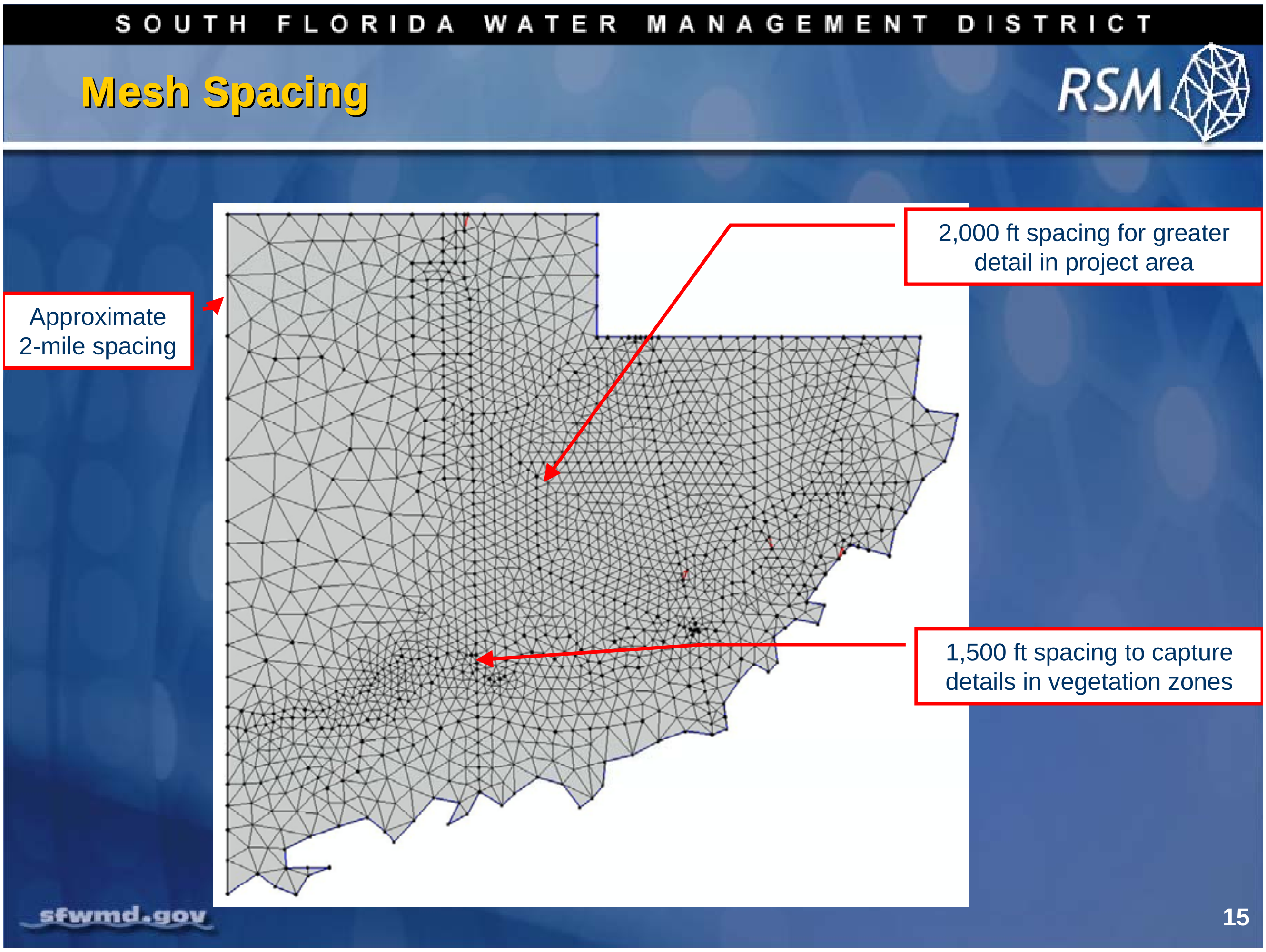


The GMS, and other mesh creation programs, can be simple or complex; you determine the degree of complexity required for modeling your project.

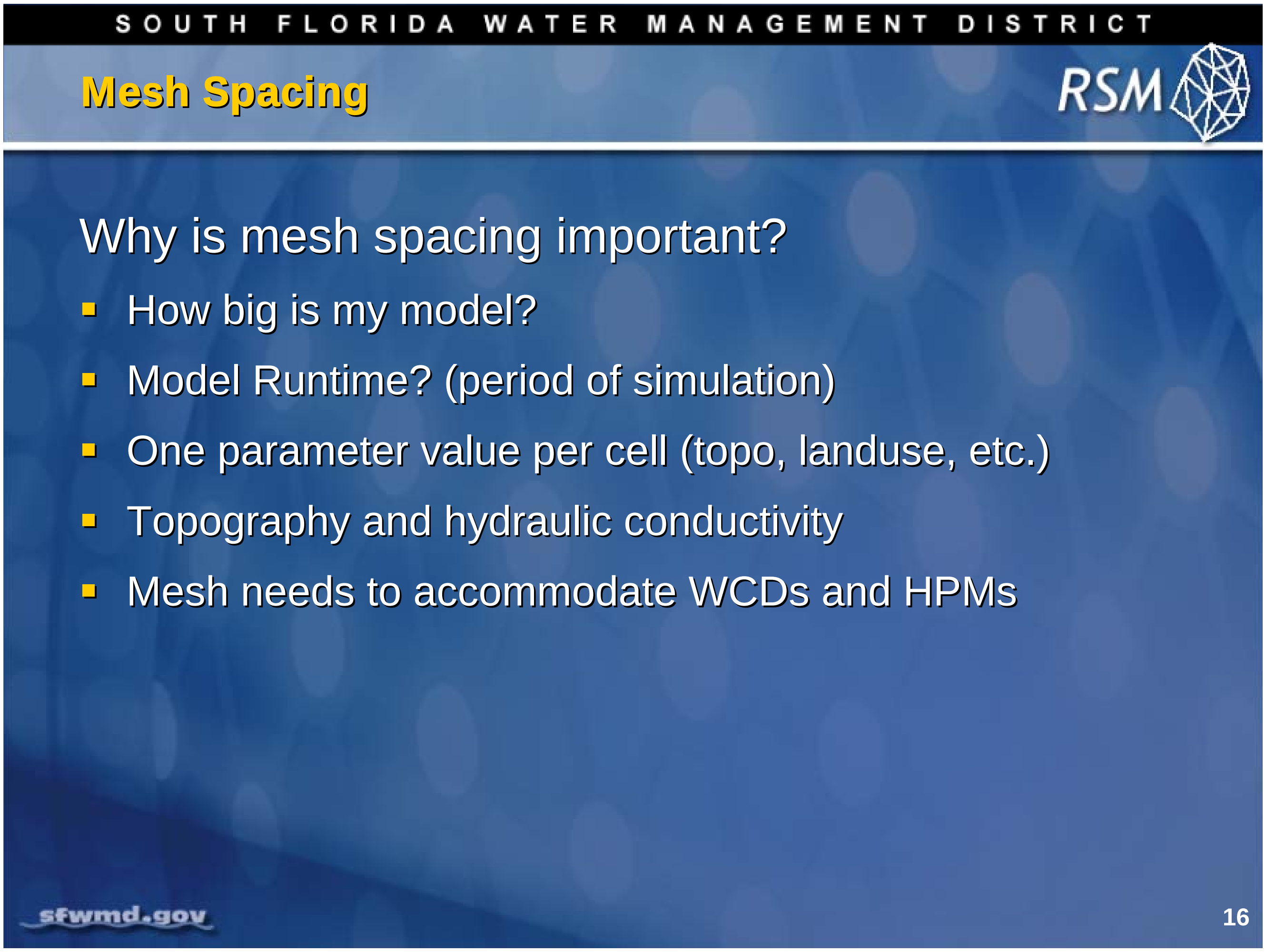
The District HESM team has implemented a simple method for using GMS to create meshes for the Regional Simulation Model.



Mesh spacing is determined by the distribution of the vertices along each framework line. The vertices can be redistributed along each framework line within the GMS. Spacing of the vertices depends on the required and appropriate cell dimensions for that area.

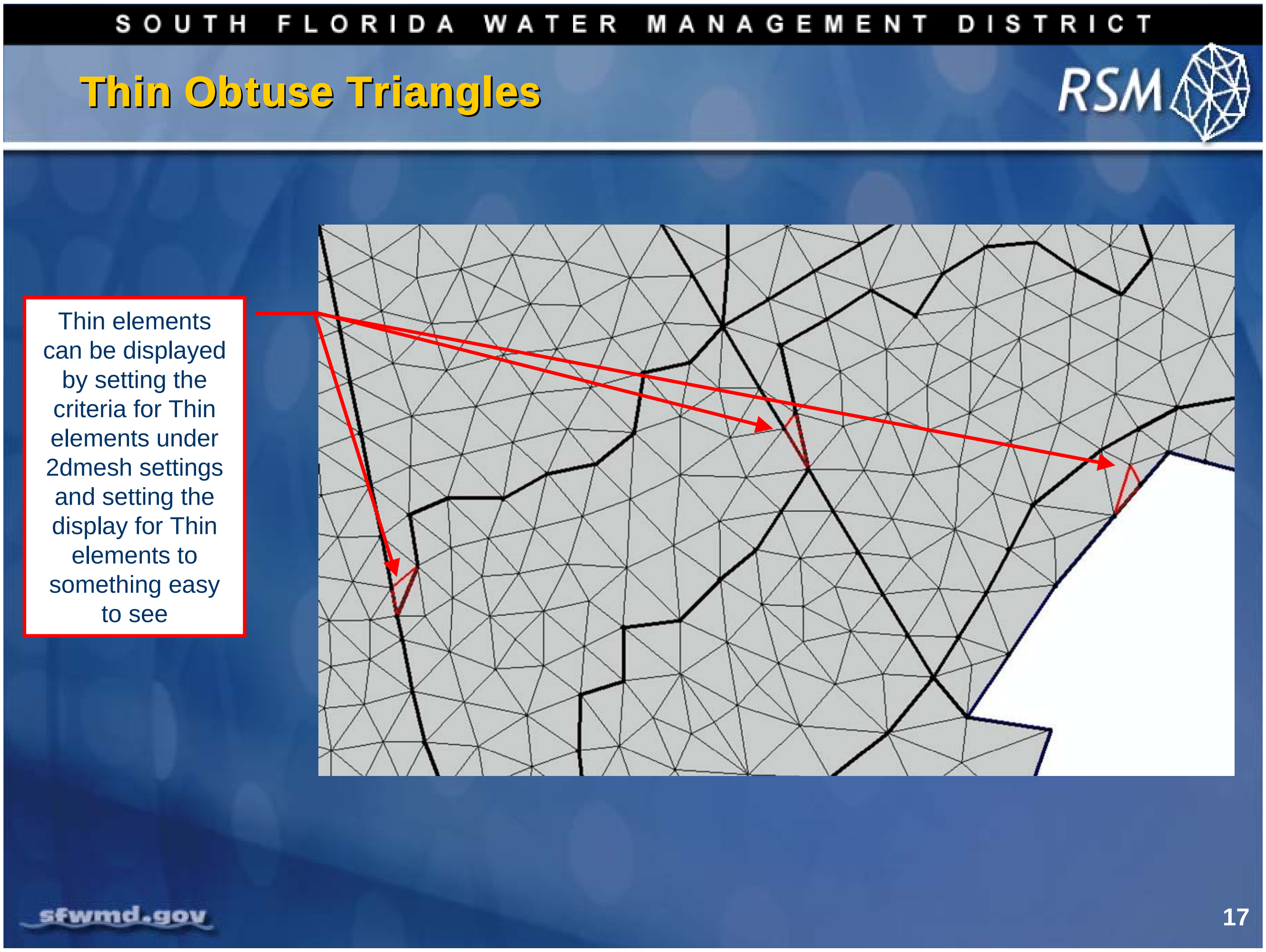


In the C111 example, the spacing along the western edge is 2 miles while the spacing in the vegetation indicator zones is 1,500 feet. The resulting mesh meets the needs of the C111 project.

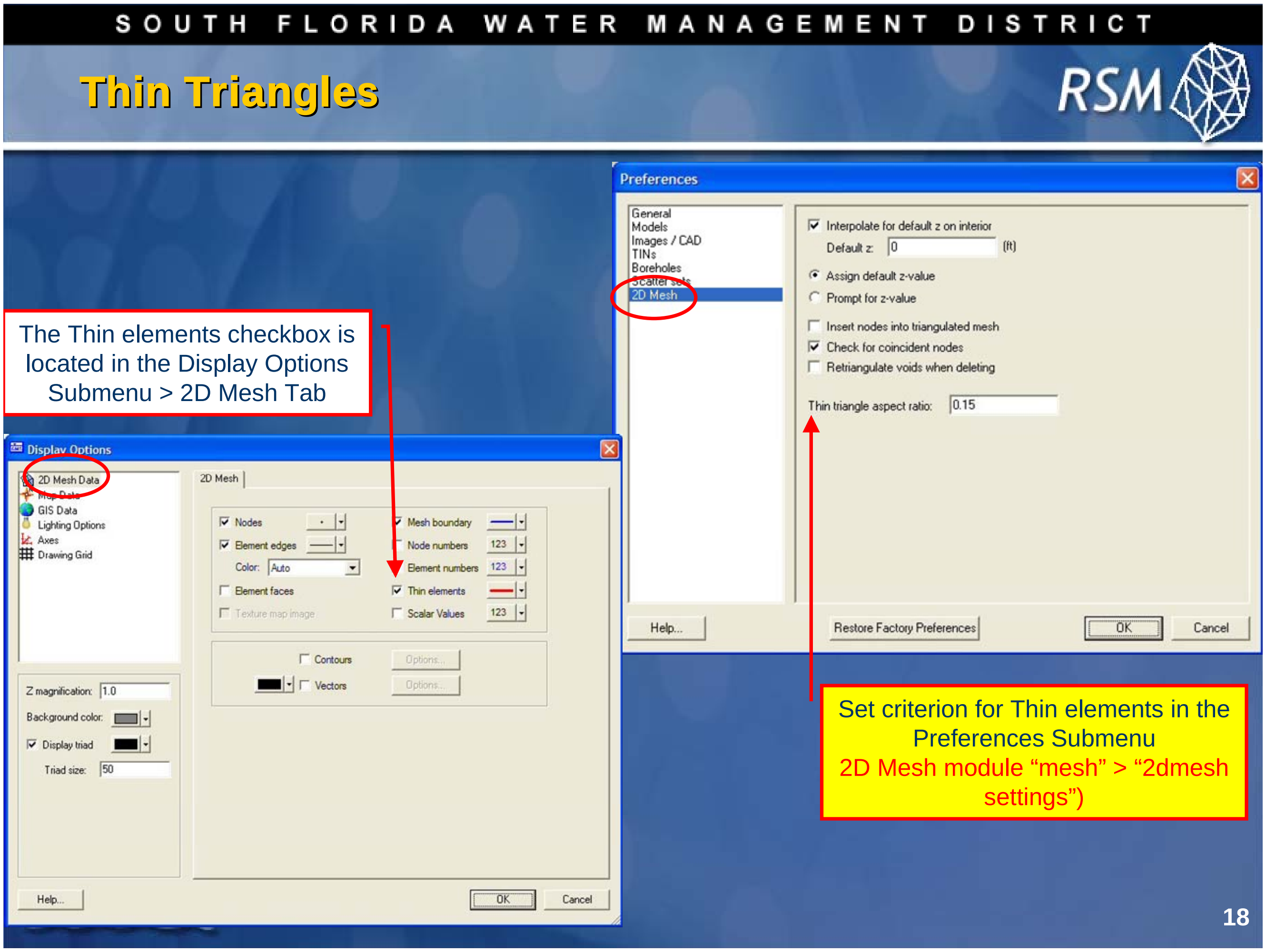


Although mesh spacing is a critical concern in the development of the model, it is constrained by run time and computer storage limitations, and risk of numerical error.

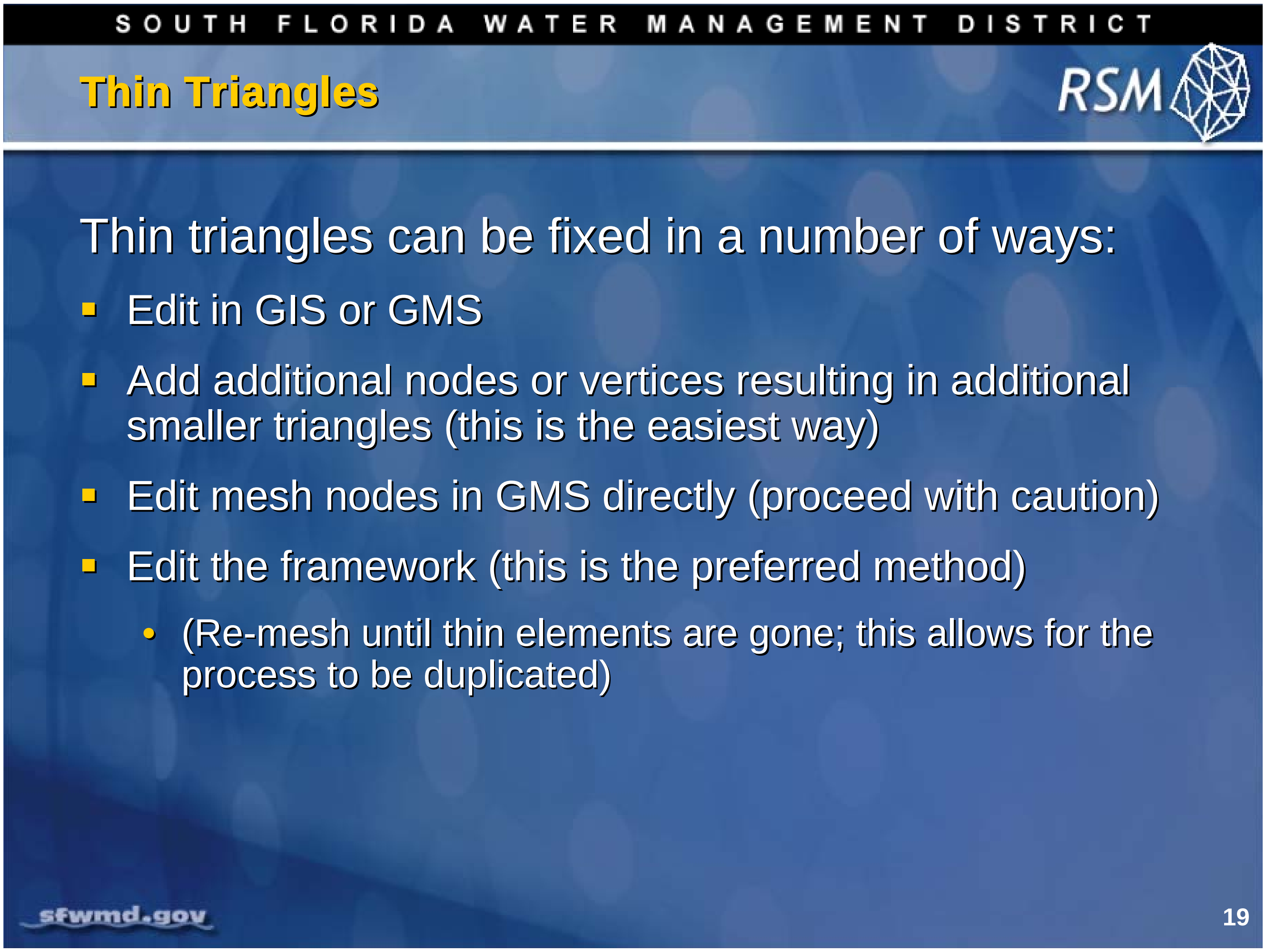
A large model can require 12-hour runs and several gigabytes of storage for output. Additionally, numerical errors occur when cells are too large or too small. For complex landscapes, it is desirable to use small cells that best represent the spatial discretization of the landscape.



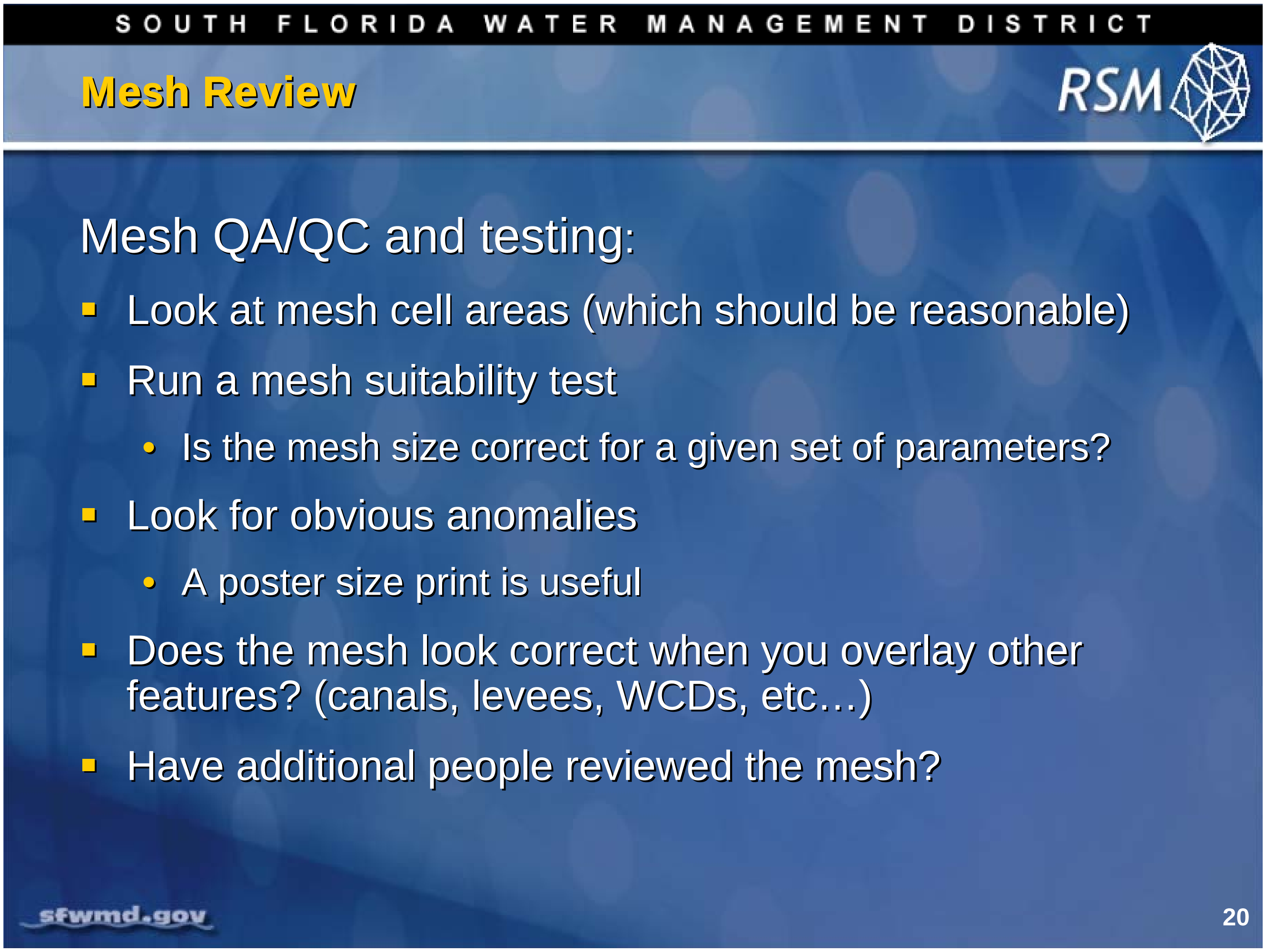
Another critical concern is the creation of inappropriate triangles in your mesh. Obtuse triangles introduce numerical error in the implicit finite-volume method.



Thin triangles can be highlighted in GMS for correction. The HESM team has defined a criterion for creating the mesh to avoid thin triangles with an aspect ratio less than 0.15.

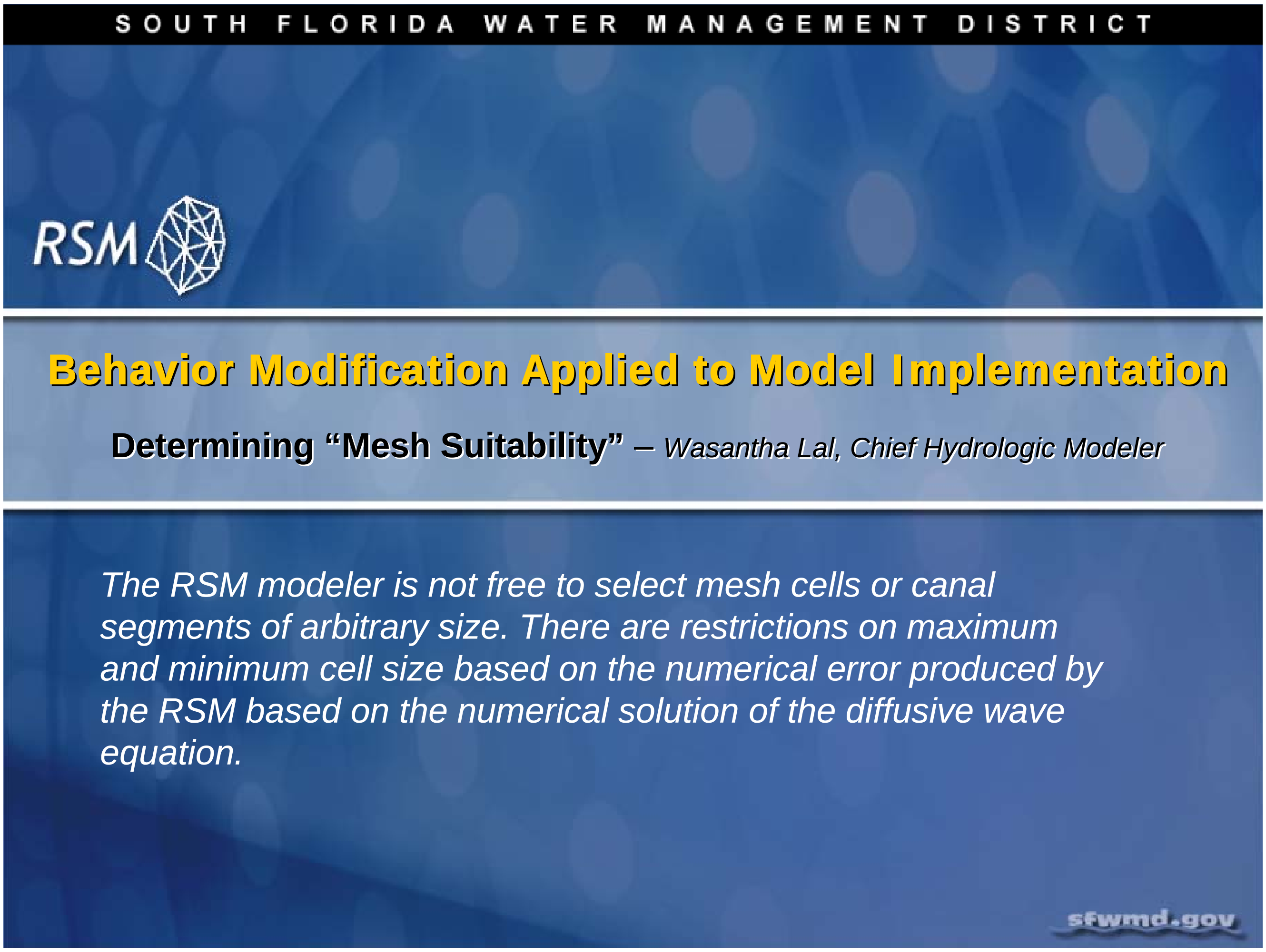


The best way to correct for thin triangles is to edit the Framework feature class and modify the locations of fixed nodes. Your revised framework can then become part of the standard framework.

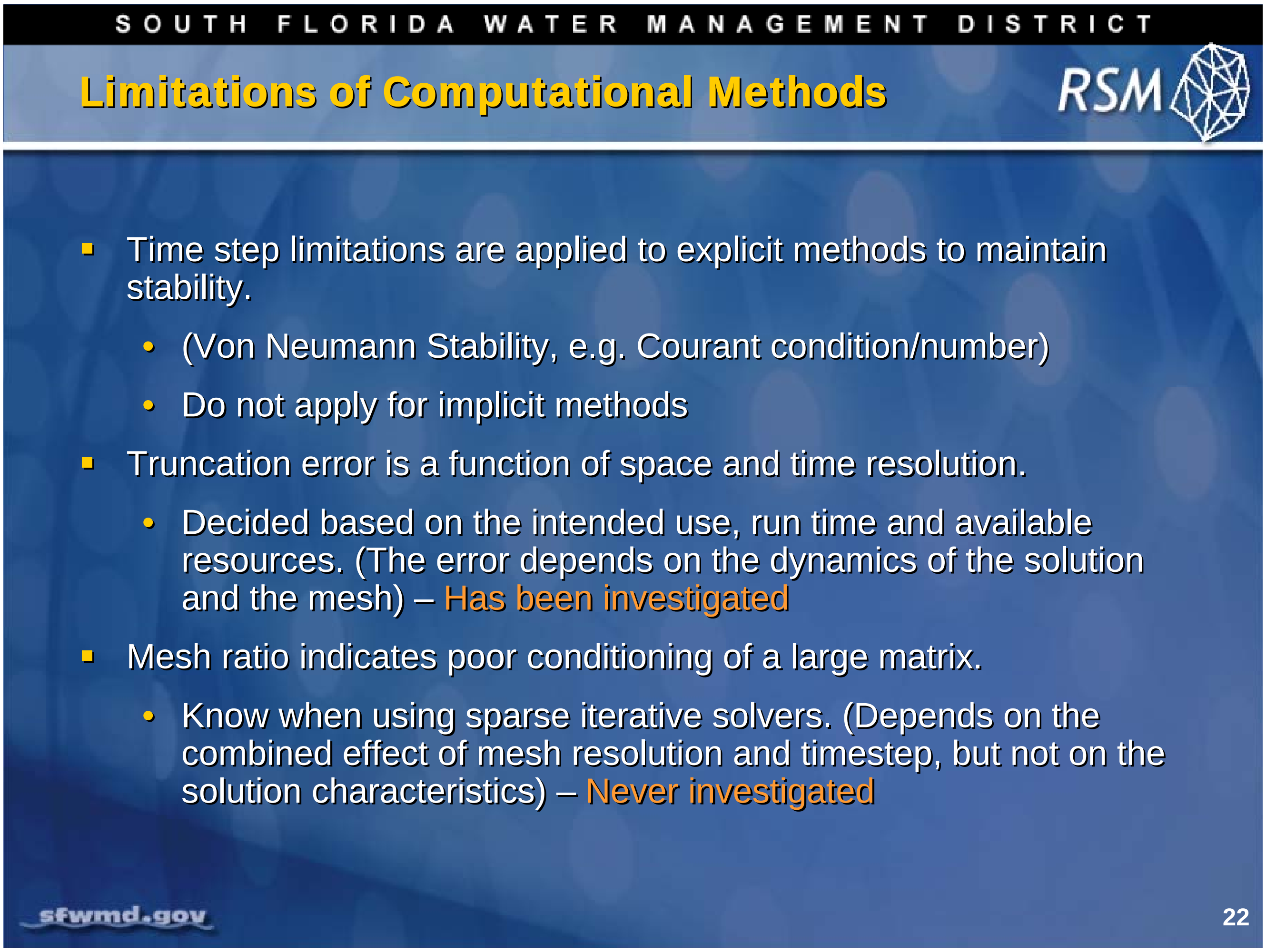


Review the mesh to ensure it meets the needs of the modeler and the stakeholders.

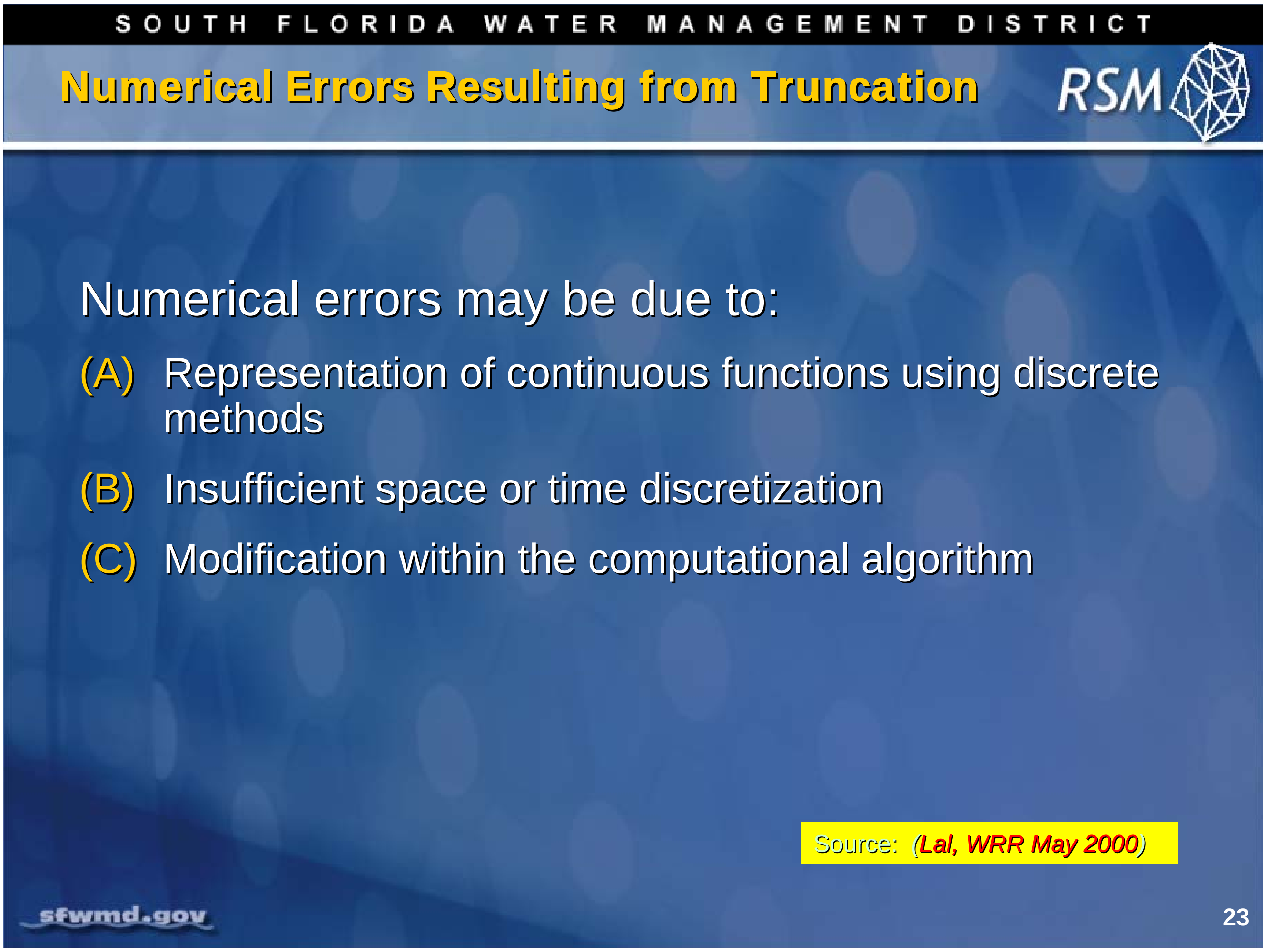
Although automated mesh generation tools provide a sound mesh, the necessary requirements defined in the Framework feature class can lead to creation of meshes with inferior characteristics. Unfortunately, such poor characteristics often only present themselves as excessive run times as the sparse solver iterates to solve the equations of flow in a poor portion of the mesh.



The topic of mesh generation requires a discussion about the limits of large and small cells. In one sense it is tempting to create large cells to reduce run time and model complexity. Sometimes, with small domains, it is tempting to create small cells to obtain better resolution of local hydrology. However, there are limits to cell size due to the numerical error that increases if limits are exceeded.

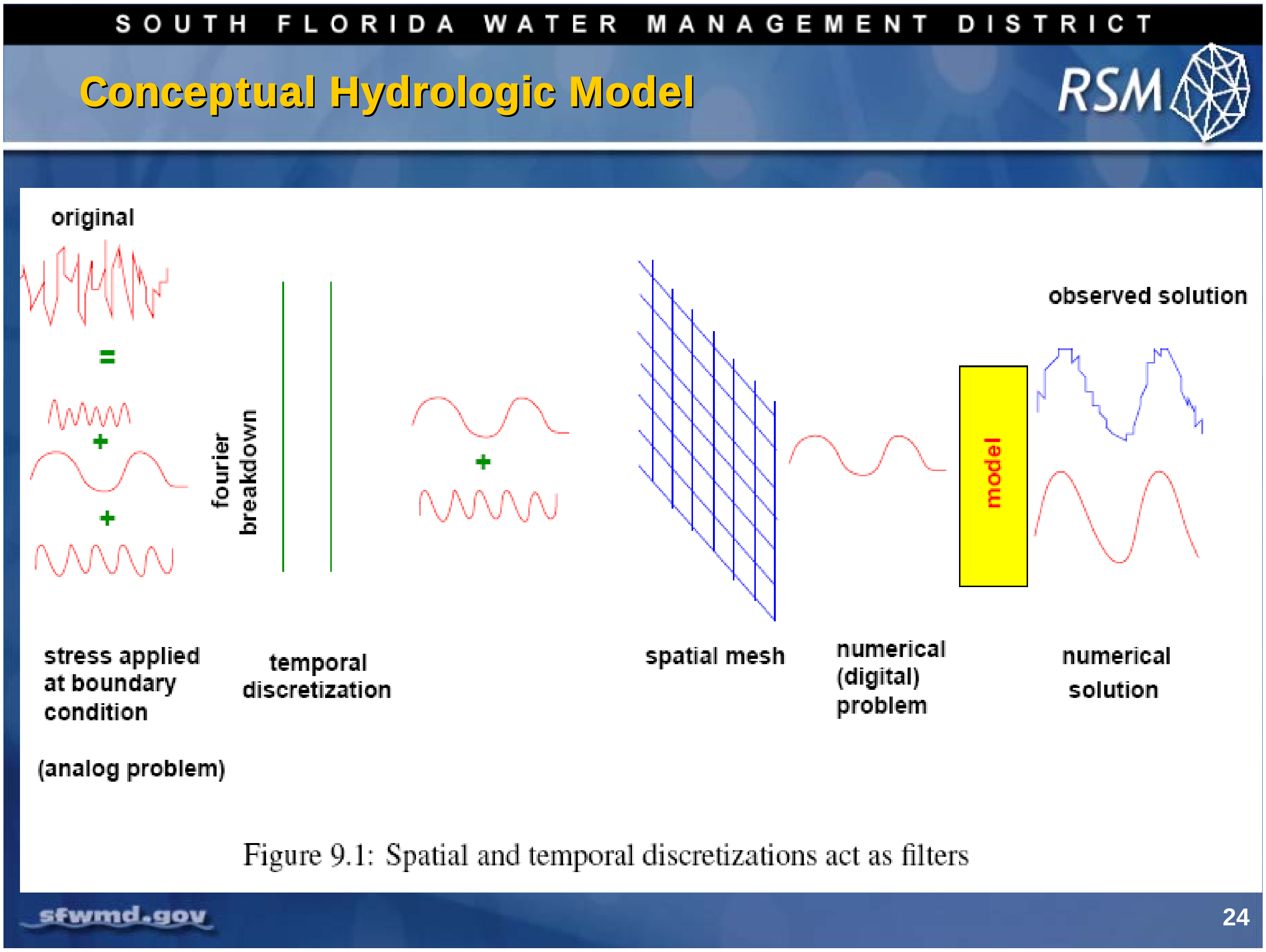


Investigating the effects of Δx and Δt on numerical stability and truncation error has required considerable effort. Additional work, as mentioned in the slide above, concerns the effects of mesh resolution and hydrologic properties on the numerical error incurred in the numerical solution.



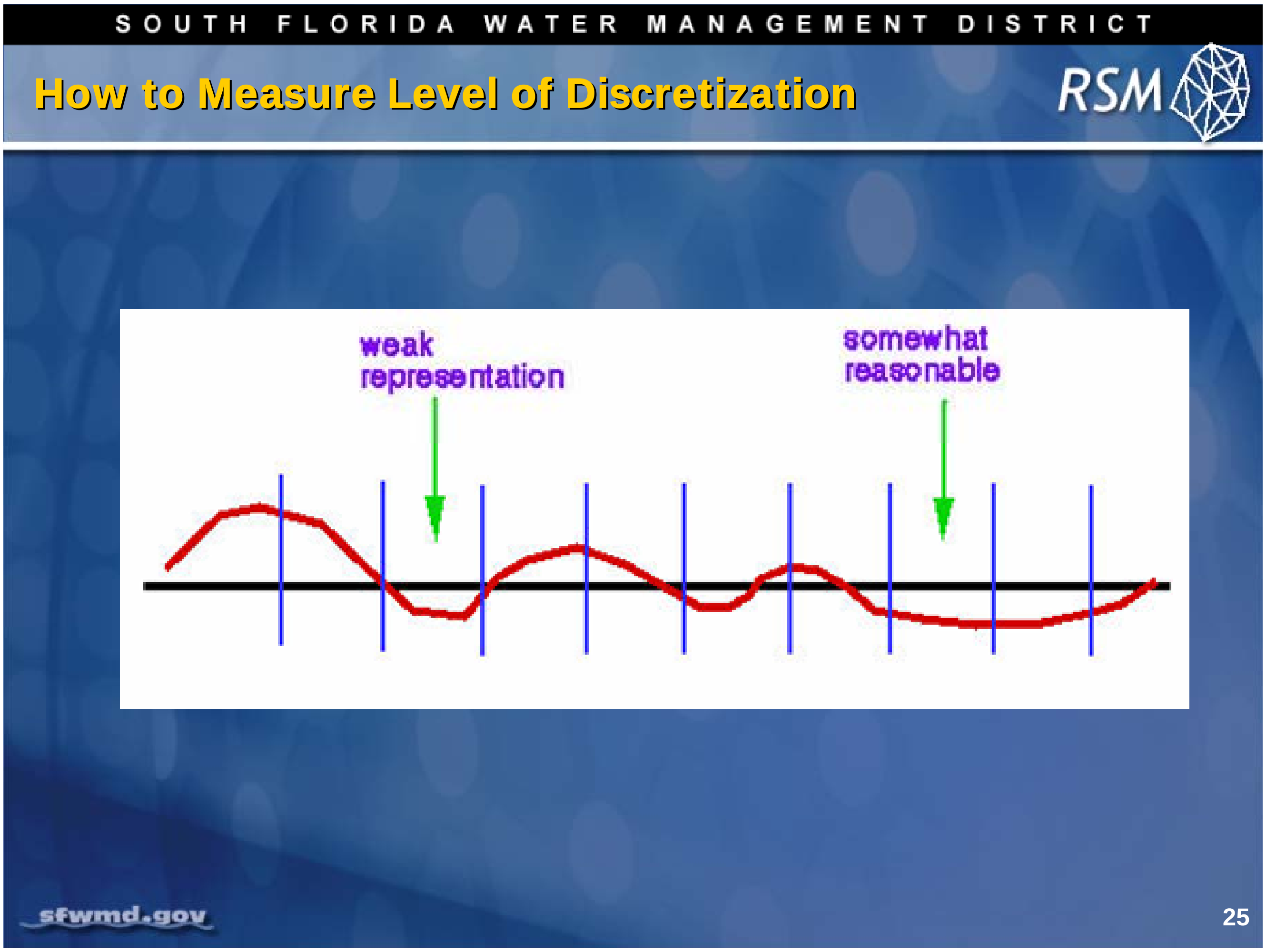
Three types of errors will occur as a result of the application of discrete numerical methods to the solution of differential equations. Choice of timestep and mesh size must be considered in conceptualizing the problem.

NOTE: **Full references are provided on page 6.2.**

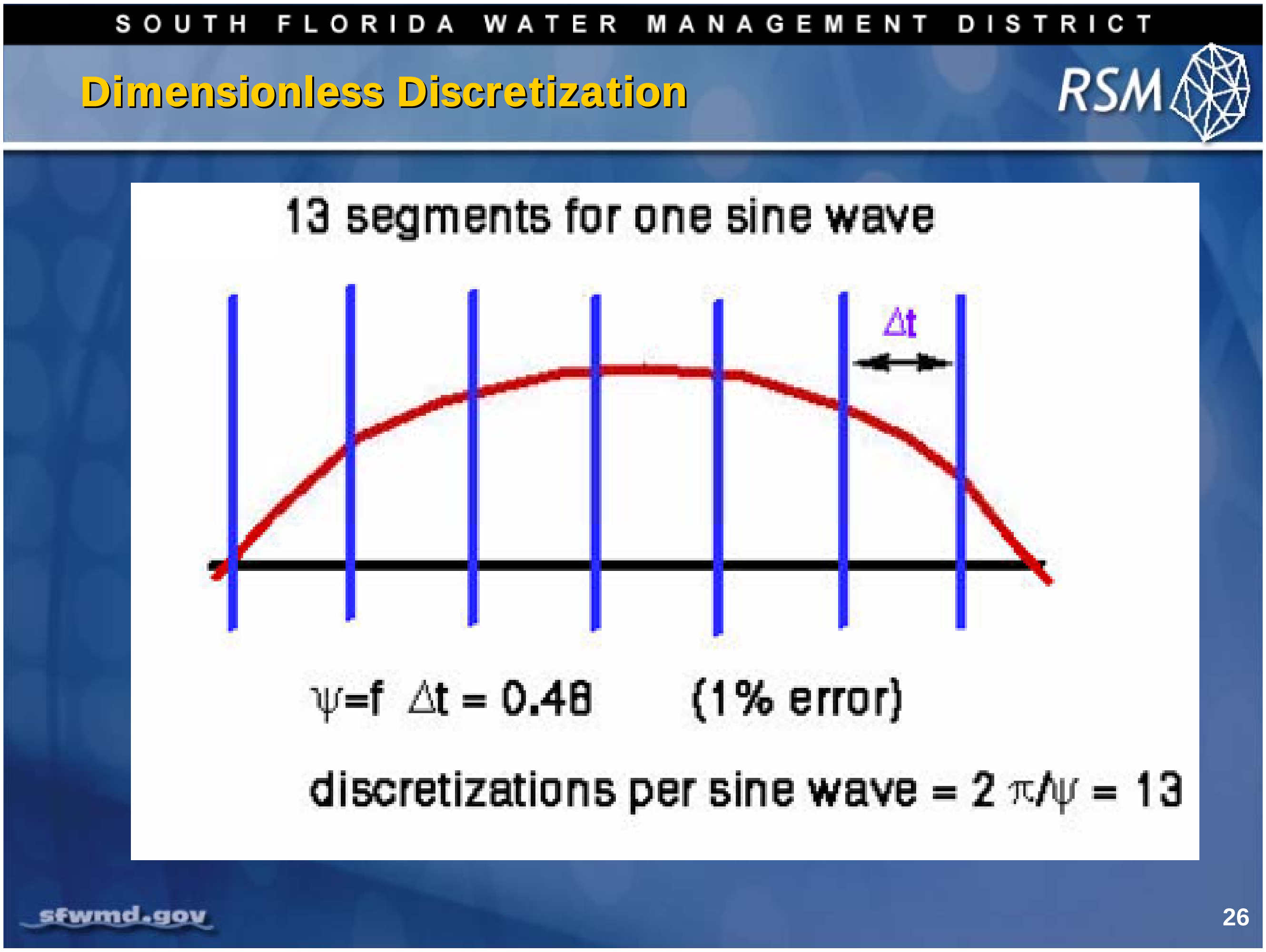


The hydrologic system being modeled consists of many "signals" with frequencies (time domain) and wave numbers (spatial domain). The measured signals such as rain, refET and plant growth through the hydrologic model are processed and the results are compared with other signals (e.g., groundwater and stream stage, canal flow and runoff).

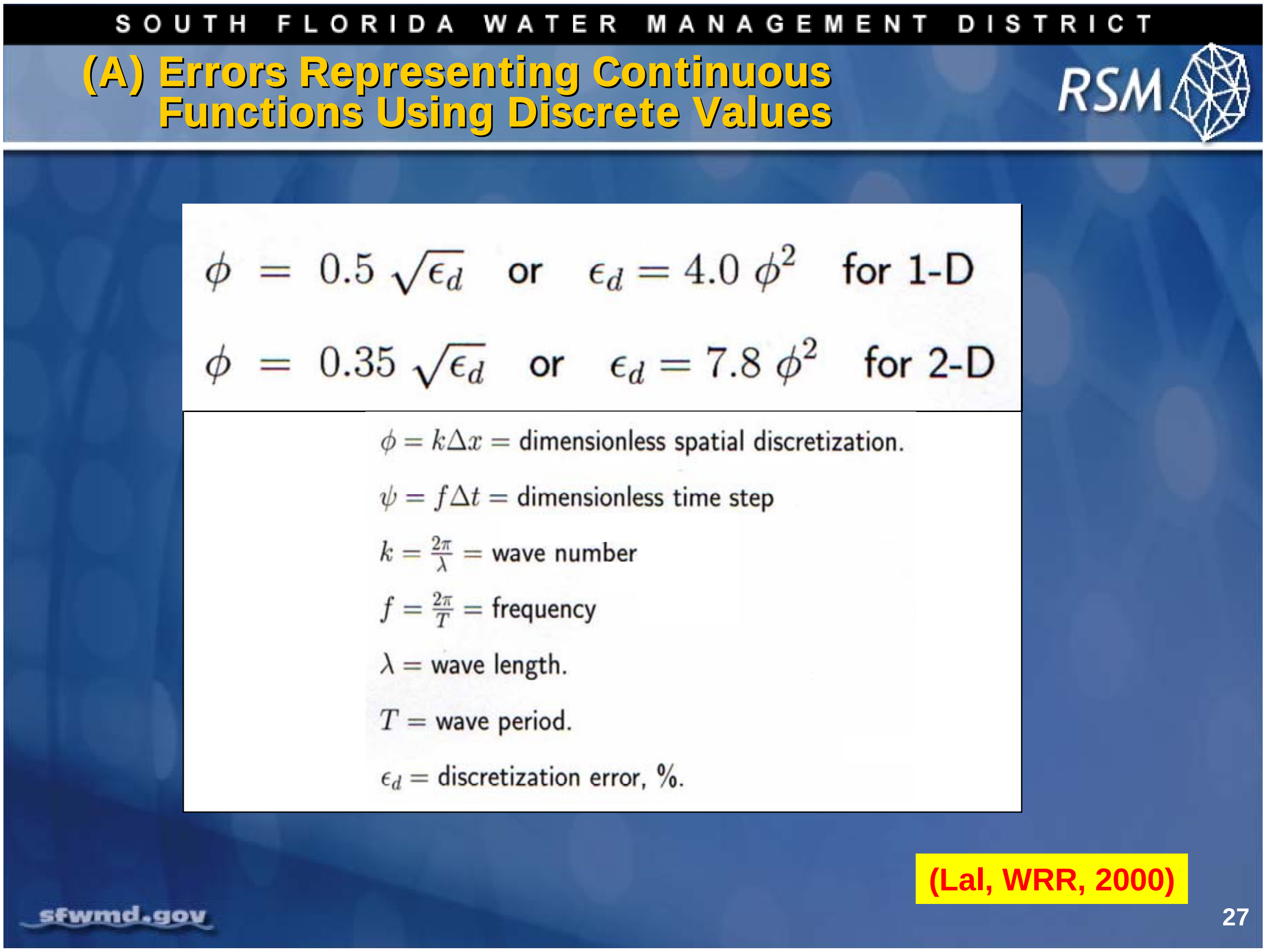
Methods of data collection and discretization of the model affect which environmental signals we can resolve. Given the 10–20 percent uncertainty associated with rain and refET, measurement and the discretization of the numerical method, the uncertainty of the results can be considerable.



We know from time series analysis that only signals with a frequency greater than the frequency of measurement can be resolved. Therefore, better signal resolution depends upon more measurements per signal cycle.



If there are 13 segments per one sine wave, you can resolve the signal with 1 percent error. With the daily timestep model, we are unable to resolve environmental signals with less than a biweekly frequency.



The error that results from discretization has been determined for 1-dimensional and 2-dimensional numerical simulations.

Error is a function of the:

- Frequency and wave number of the signals
- Time step and spatial increment

(Error occurs even though the model appears to produce accurate results.)

In the calibration process, the model output is matched to measurements of groundwater and canal stage, which are more accurate than our inputs (< 4 percent). As part of the calibration process, "effective" parameter values are created providing optimum value for a spatial and temporal discretization.

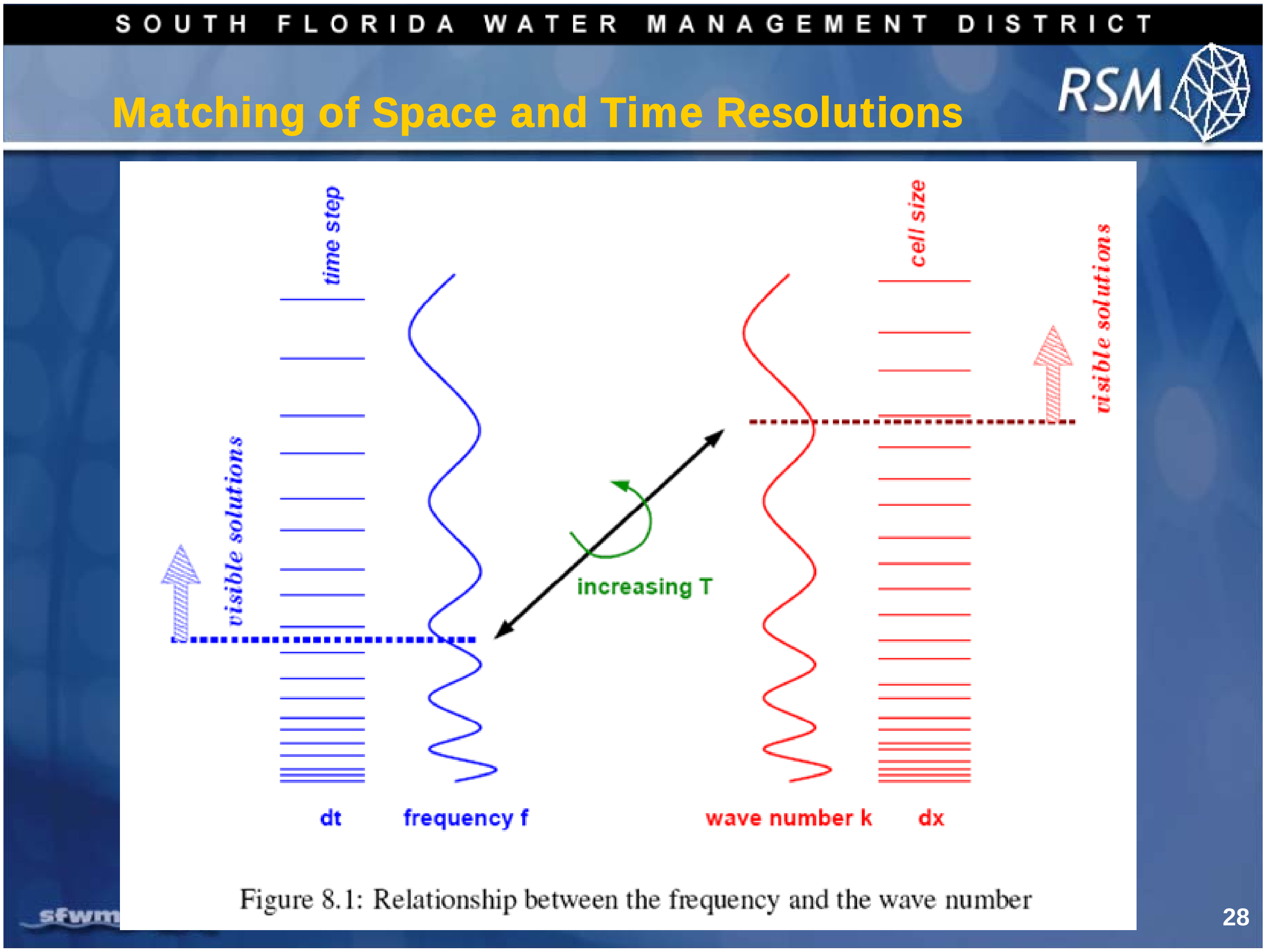


Figure 8.1: Relationship between the frequency and the wave number

As a result of this analysis, evaluating the potential error incurred due to temporal and spatial discretization is necessary.

Additionally, make certain that Δt is matched with Δx or computational resources will be wasted trying to overcome discretization error in the other domain.

SOUTH FLORIDA WATER MANAGEMENT DISTRICT

(B) Non-Matching Space and Time Discretizations

RSM

$$f = Kk^2 \quad \text{For 1D}$$

$$f = 2Kk^2 \quad \text{For 2D}$$

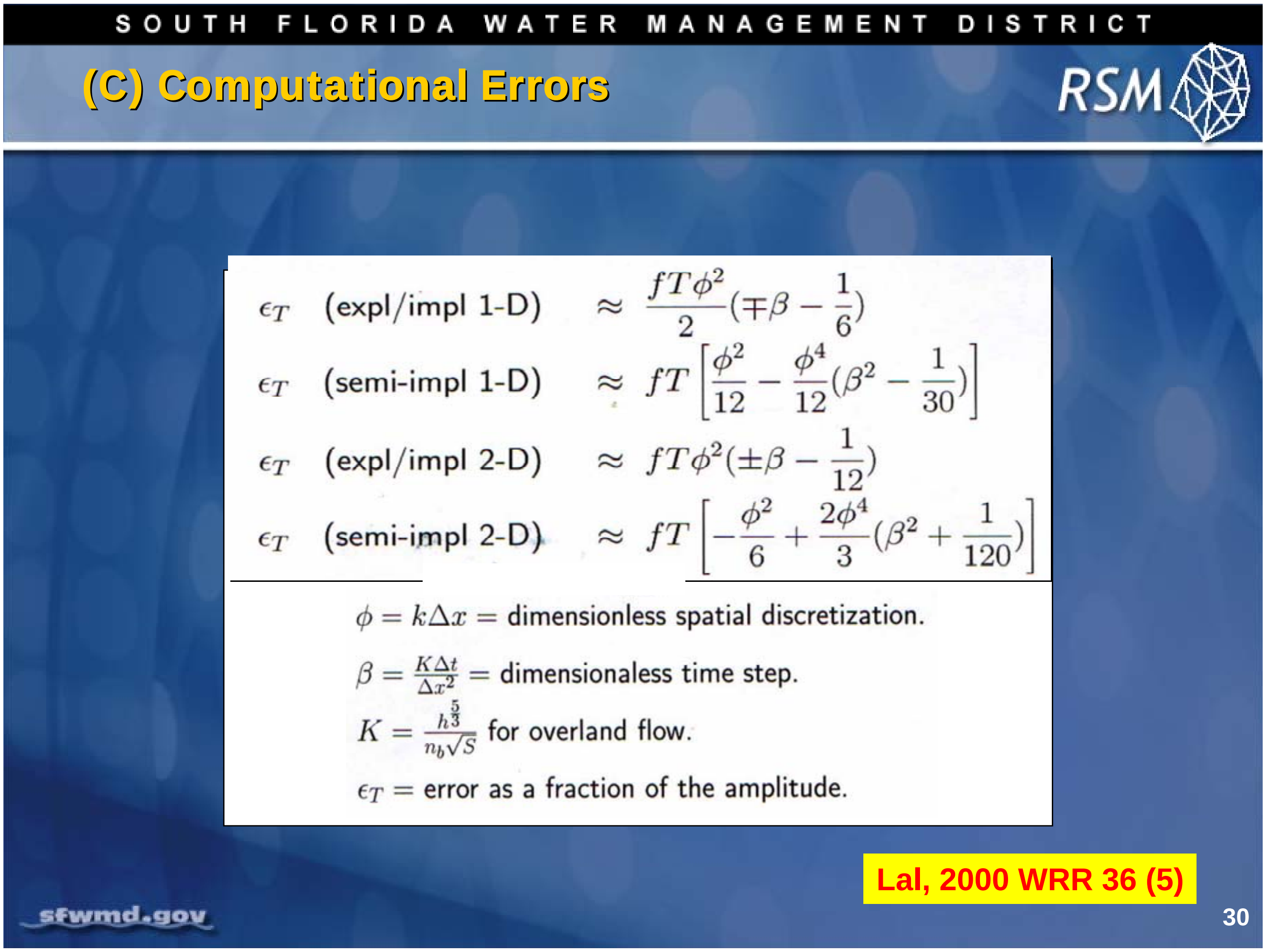
For overland flow in south Florida, $$K = \frac{h^{5/3}}{n_b \sqrt{S}}$$

or $$K = 250 m^2 / s$$

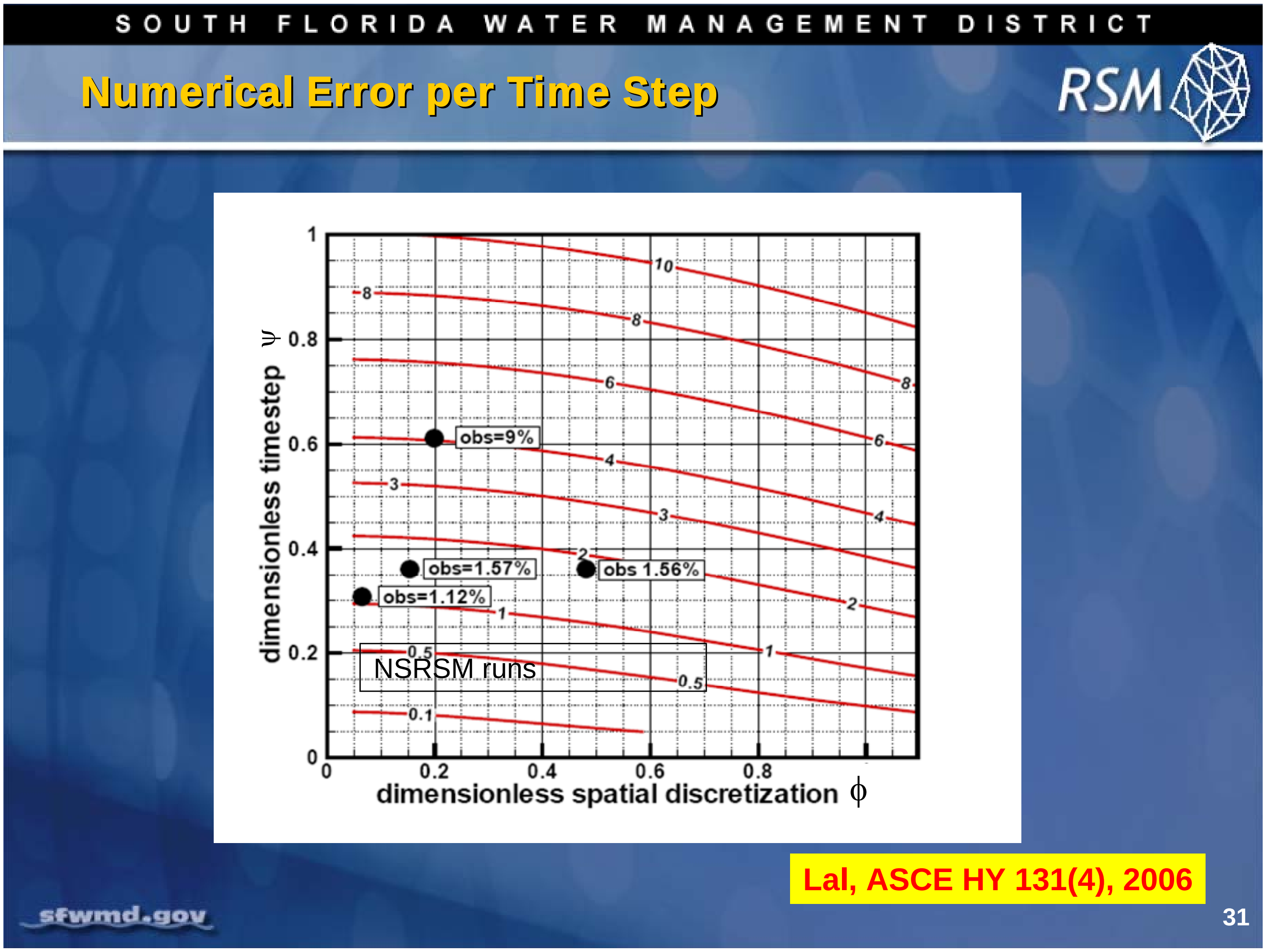
For groundwater flow in south Florida, $$K = 2m^2 / s, s_c = 0.25$$

sfwmd.gov 29

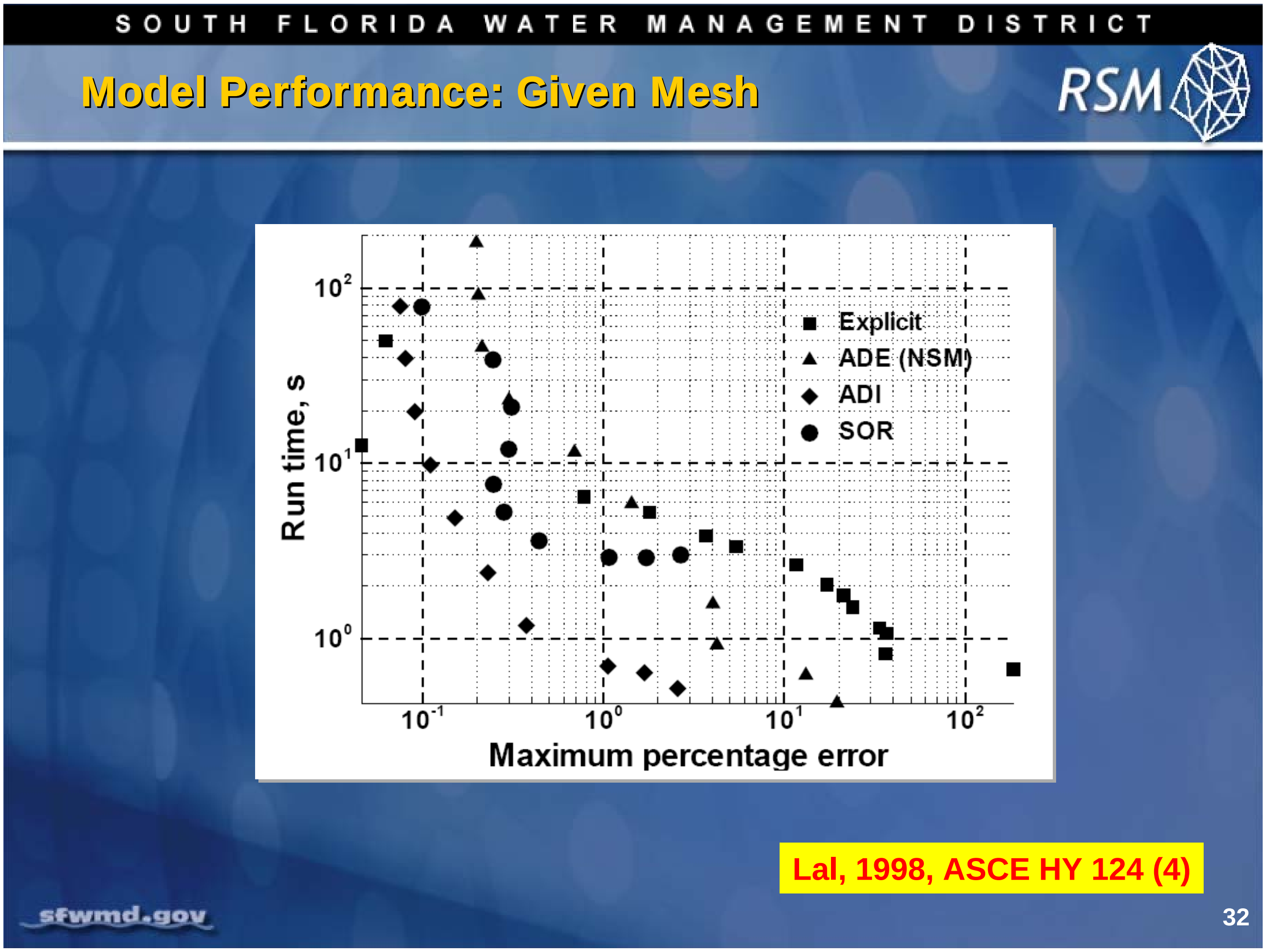
There is a relationship between the wave number and the frequency that can be defined for overland flow using the Manning Equation, and for groundwater using Darcy's Law. This allows us to specify the appropriate Δt and Δx.



The relationship between k and *f* can be used to derive the truncation error as a function of Δt and Δx and K (or T).

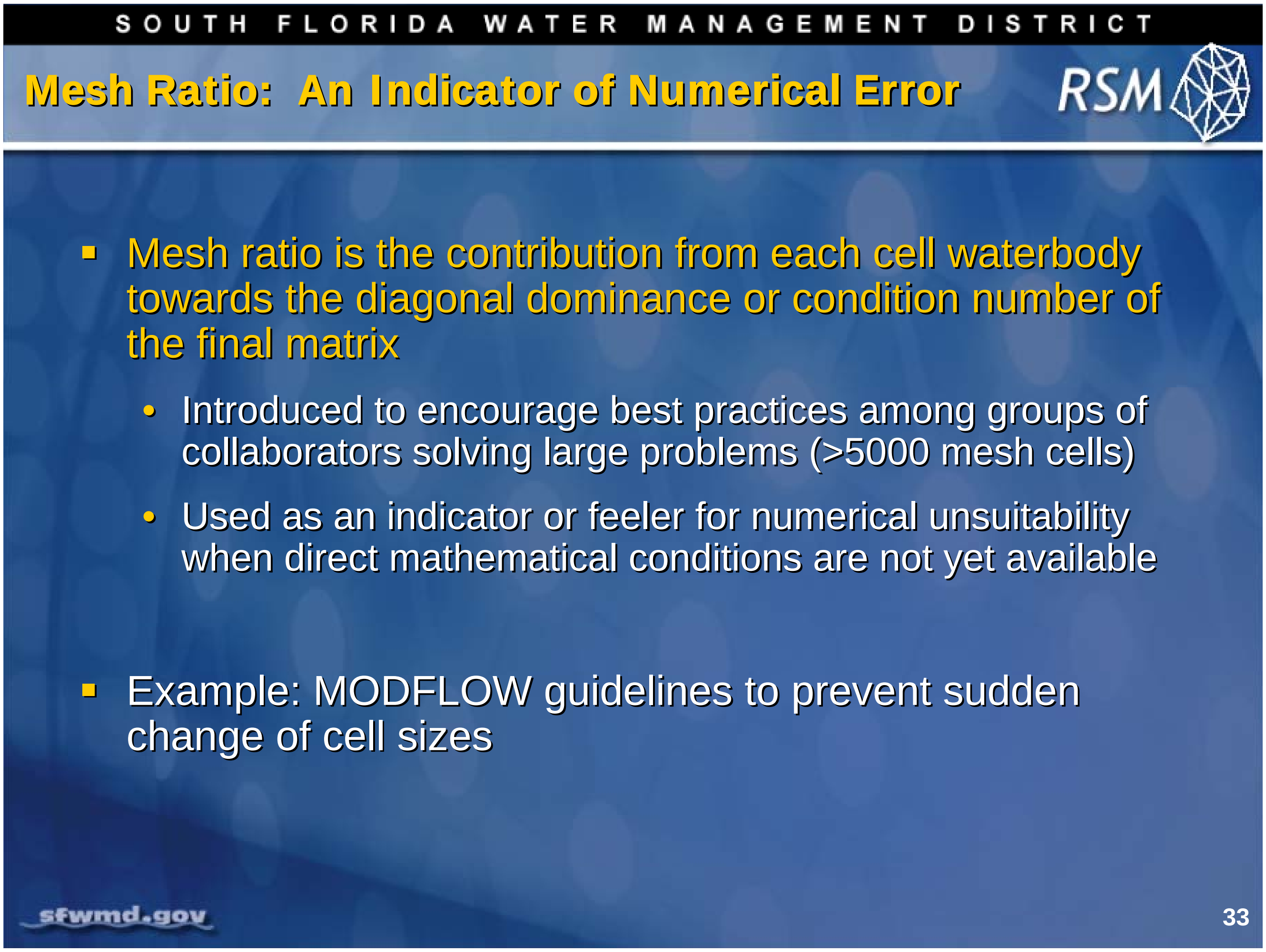


By making appropriate choices for timestep and cell size, truncation error can be kept low. The graphic above represents truncation error, in terms of Ψ and ϕ, for the relationships presented on the previous slide.



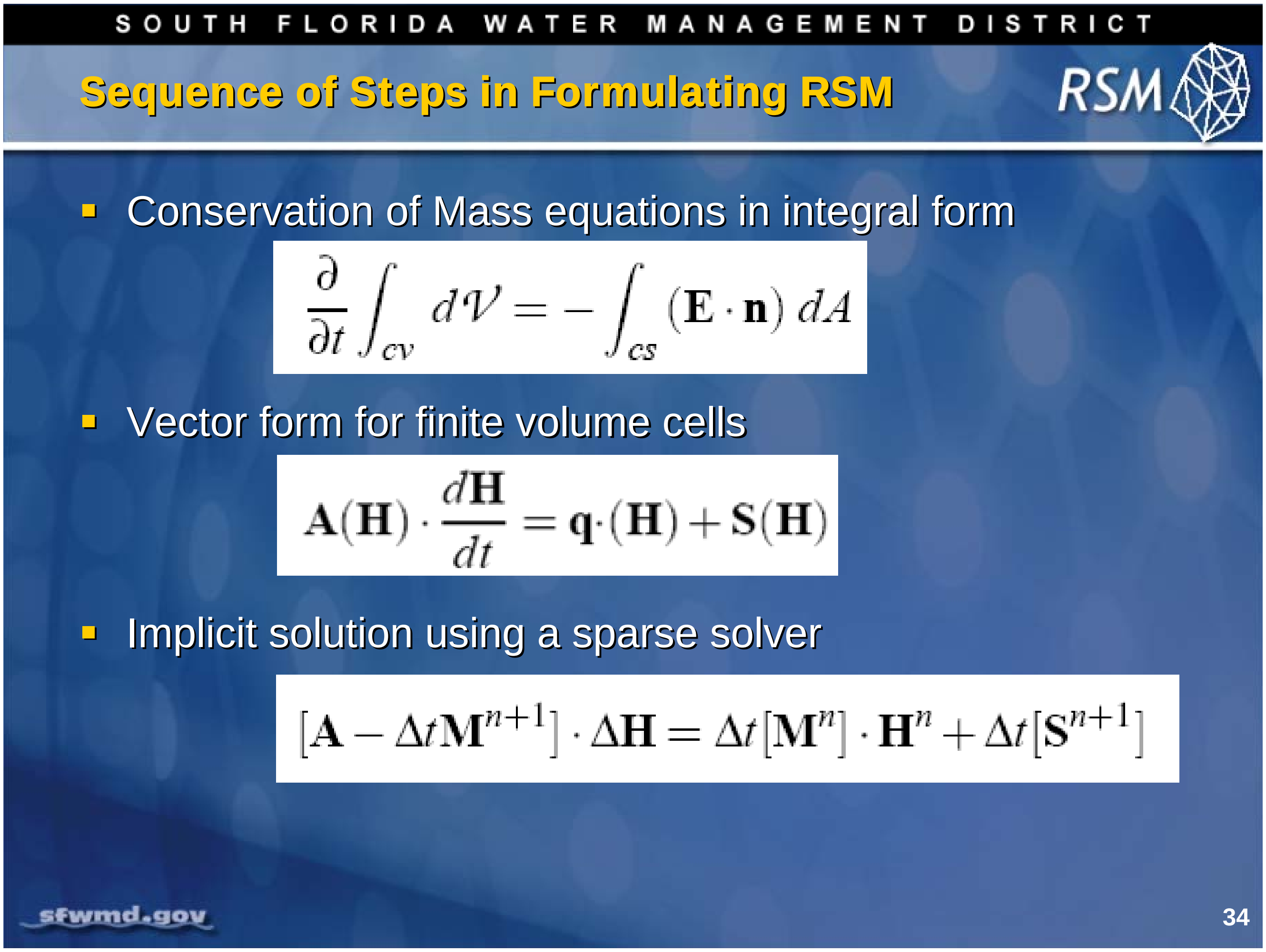
The amount of numerical error is a function of the mesh density, which increases the run time. The explicit solution method has the greatest run time for the percent of incurred error, while the alternating-direction implicit (ADI) method has the smallest run times.

To prevent numerical errors, keep the mesh size and timestep small. However, reducing the error results in increasing run times.



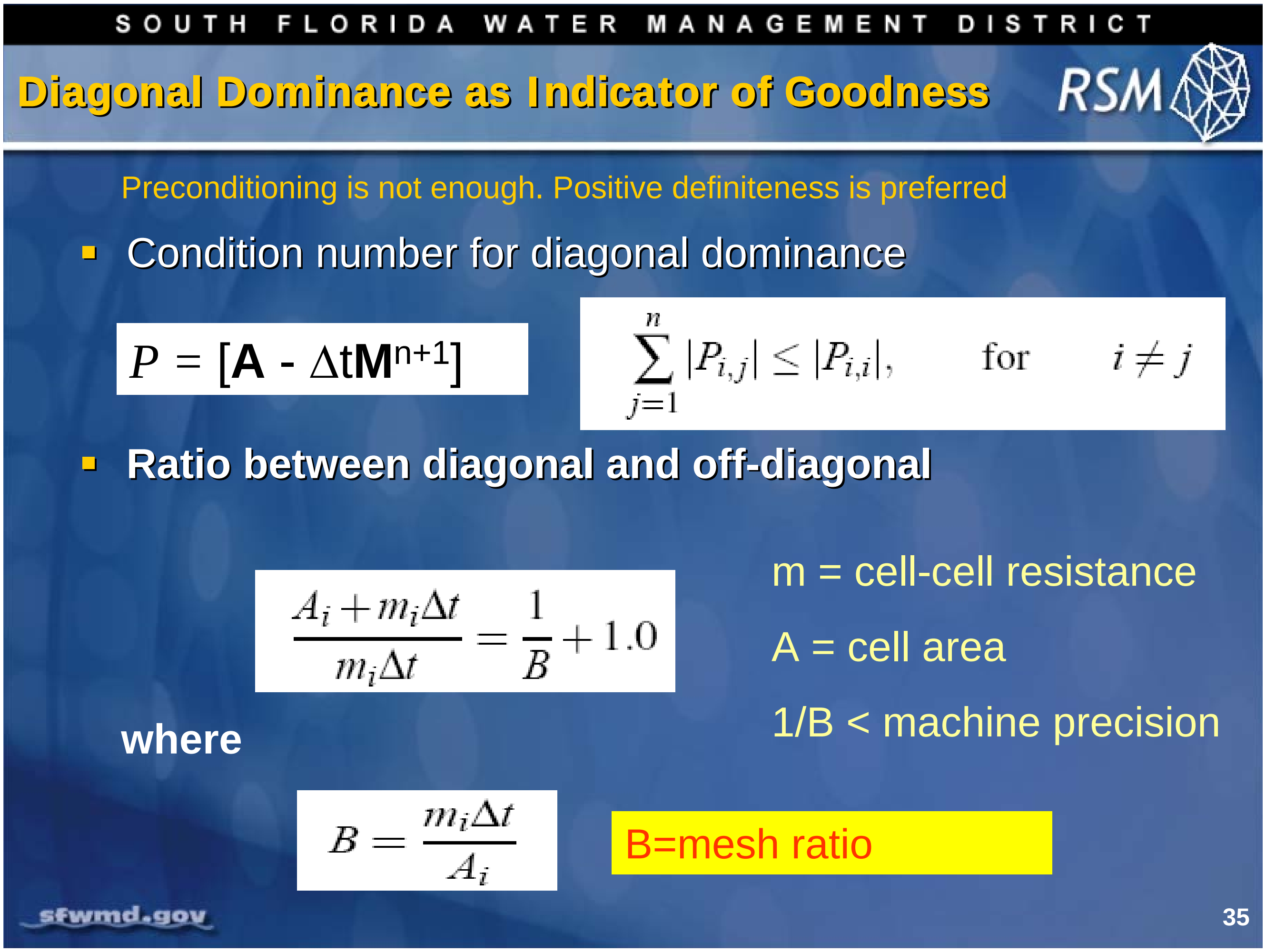
Mesh ratio is presented as a guideline for determining appropriate cell size within a large mesh.

The concept of a “mesh ratio” was developed to indicate the impact of each cell waterbody towards the diagonal dominance or condition number of the matrix. The mass residual of the numerical solution is a function of the mesh ratio, which is a direct result of poor conditioning of a large matrix, particularly when using sparse iterative solvers.



Mass residuals are a result of the computational methods used to solve the Conservation of Mass. The equation can be written in vector form where **A(H)** is a diagonal matrix with elements that are the effective areas of the waterbodies:

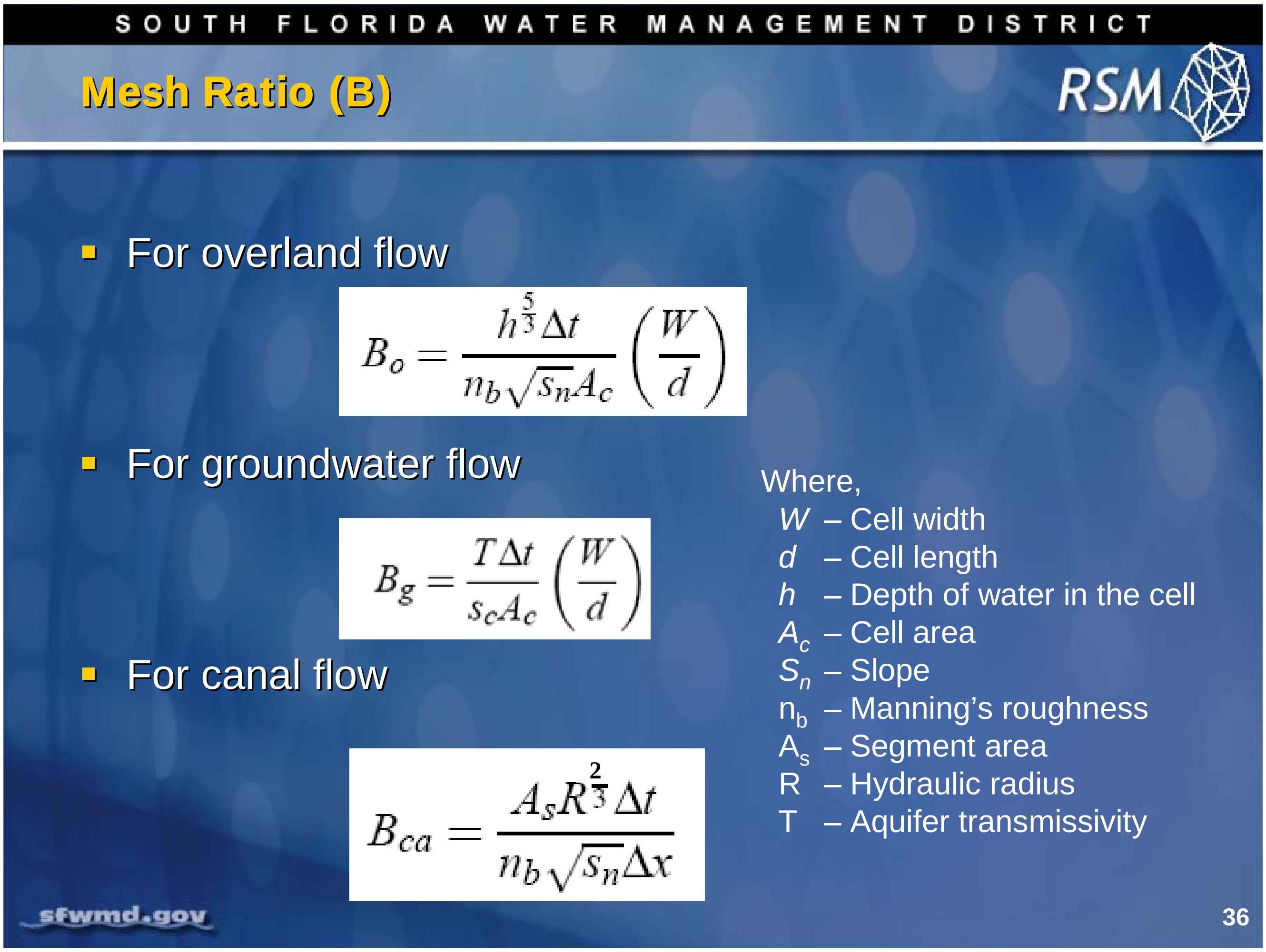
- **H** which is the vector of average heads
- **q(H)** is the vector of water flows into each waterbody
- **S(H)** are the water source terms
- **M** is the global resistance matrix (which was discussed in detail in Lecture 2, RSM Theory Overview)



Condition number for diagonal dominance is a good indicator of the condition of the matrix for iterative solvers. Numerical error and instability will occur when the matrix of linear equations for all of the waterbodies is not diagonally dominant.

Typically, this is because every watermover associated with the waterbody has a counterpart in the diagonal term and the matrix is symmetric. However, under some circumstances the ratio of the diagonal to the off-diagonal terms can become much greater than one where B is small.

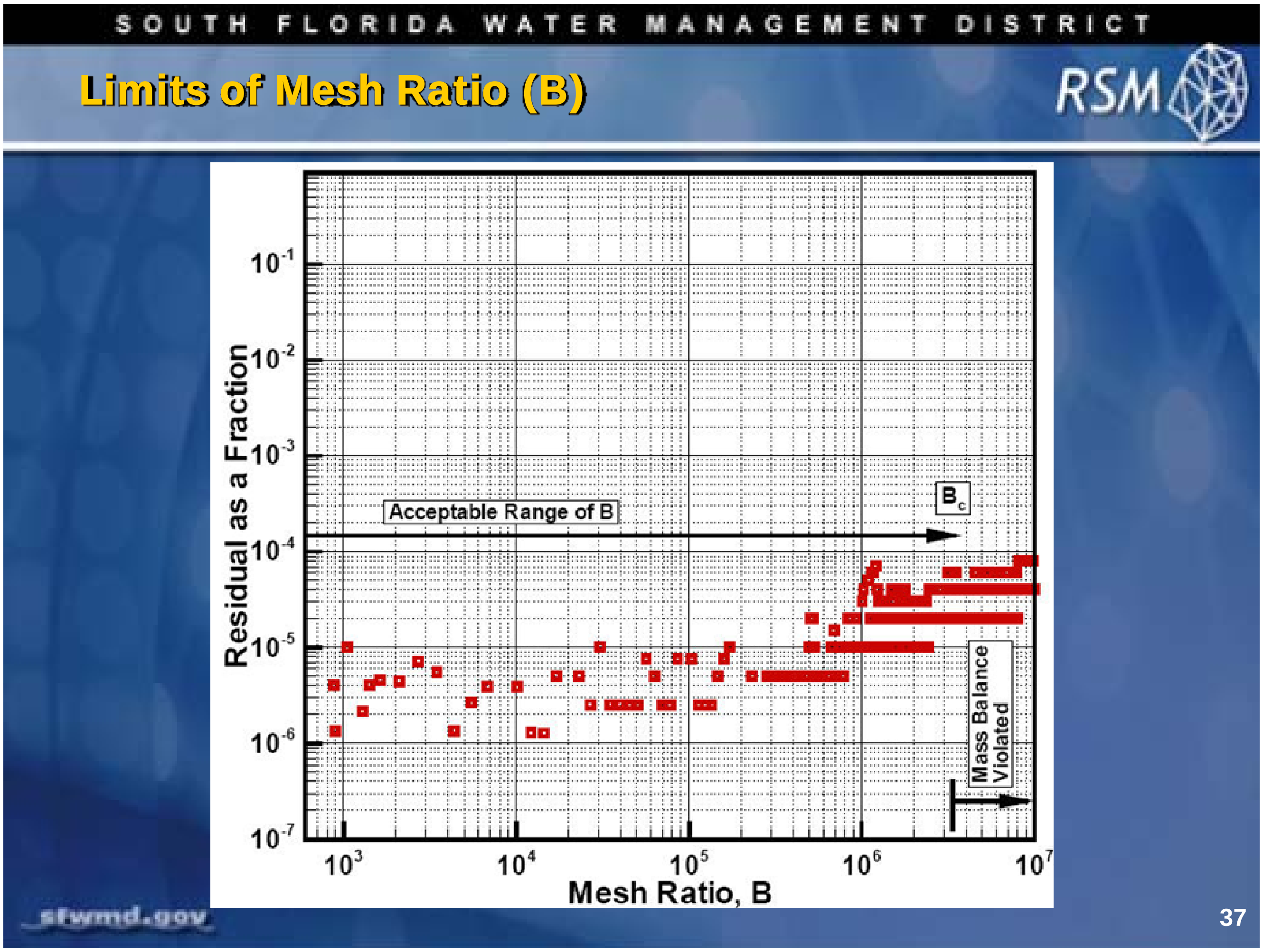
Large values of "mesh ratio" (B), the ratio between diagonal and off-diagonal, lead to convergence errors and instability.



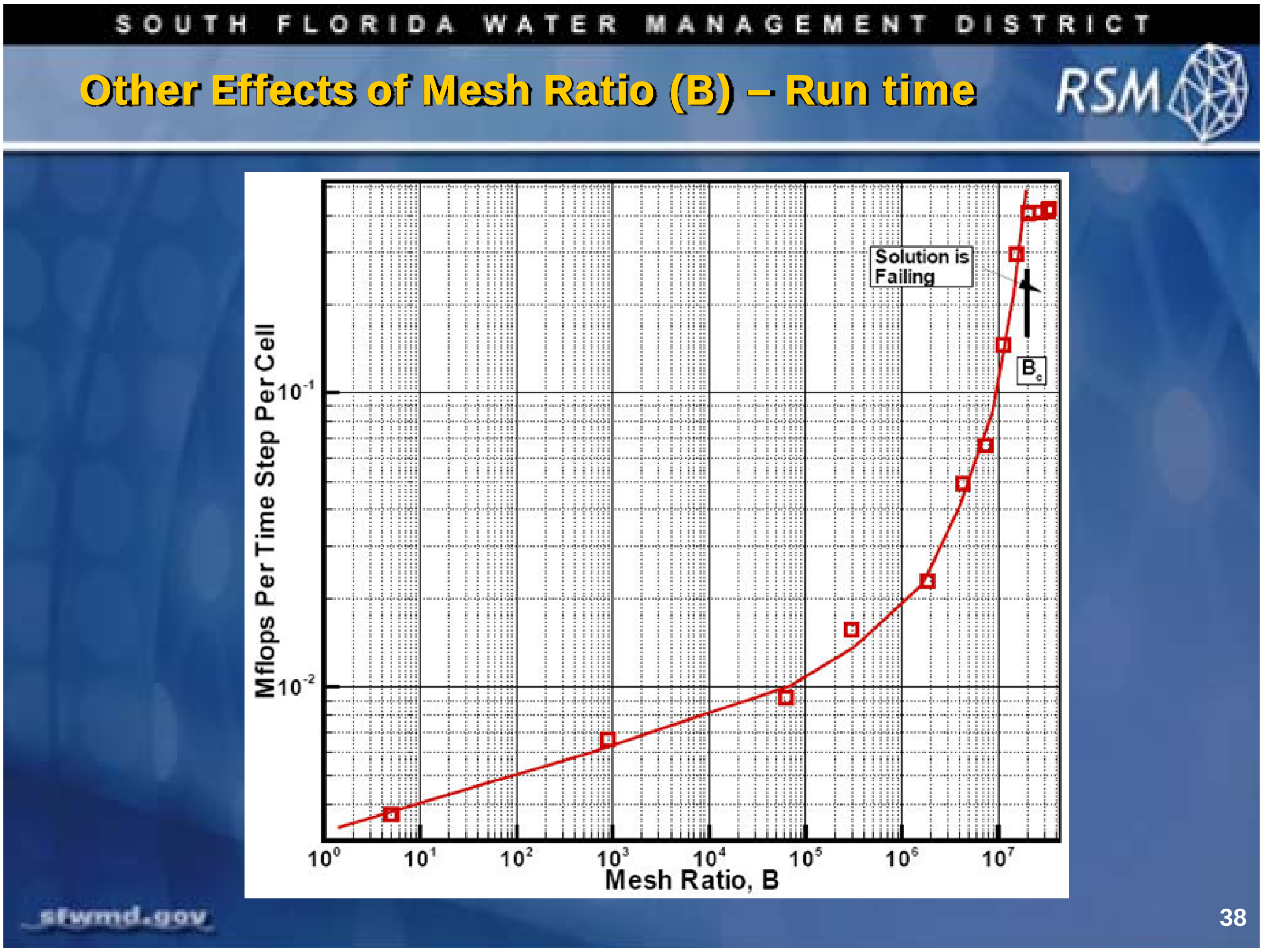
The mesh ratio terms presented on this slide were developed for the different flow regimes in the RSM. They are based on the Manning Equation or Darcy's Law.

For mesh cells, the mesh ratio is a function of the cell dimensions and cell area.

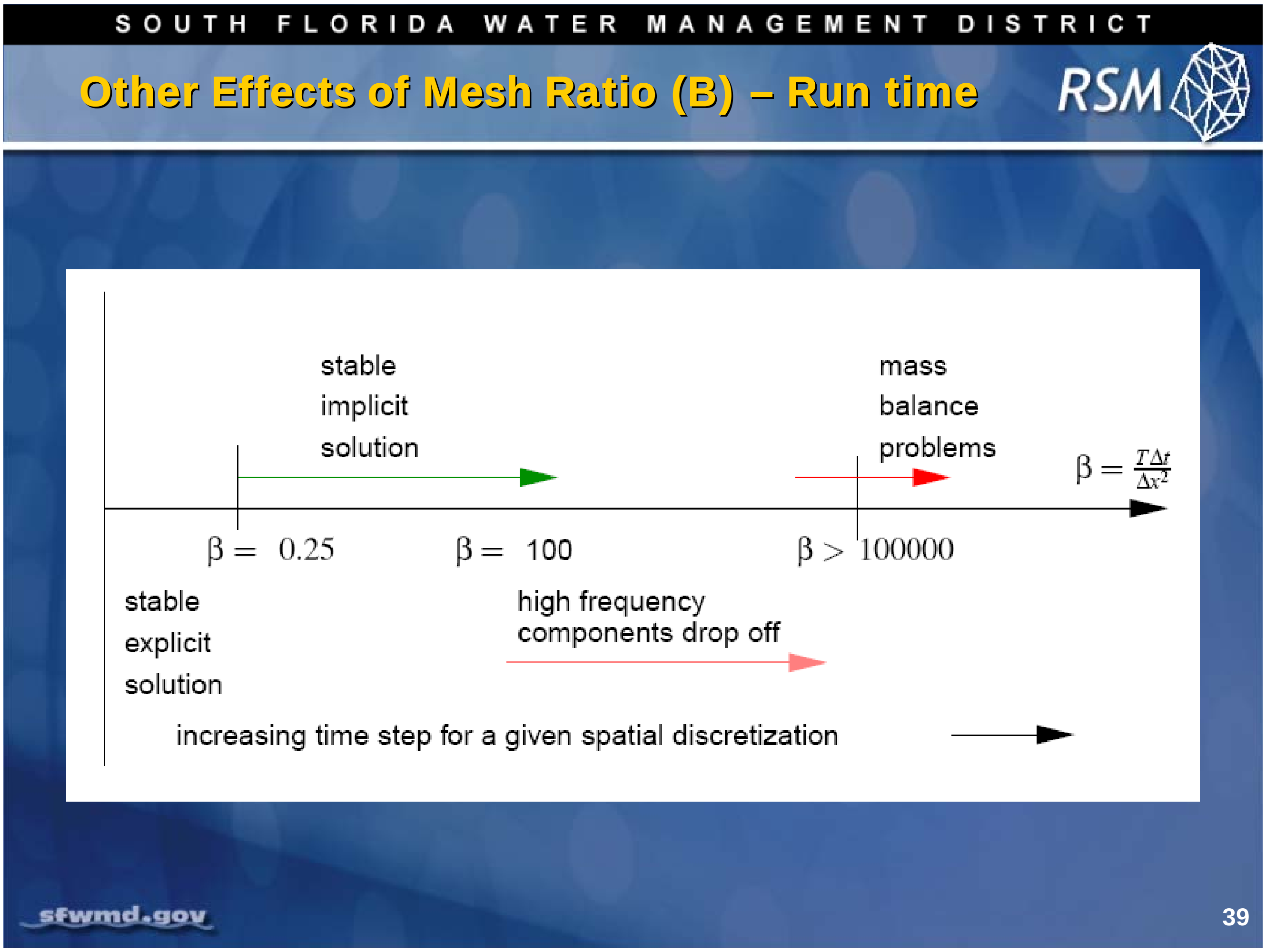
For overland flow, the mesh ratio becomes greater when the cell area becomes too small or the water depth becomes too large.



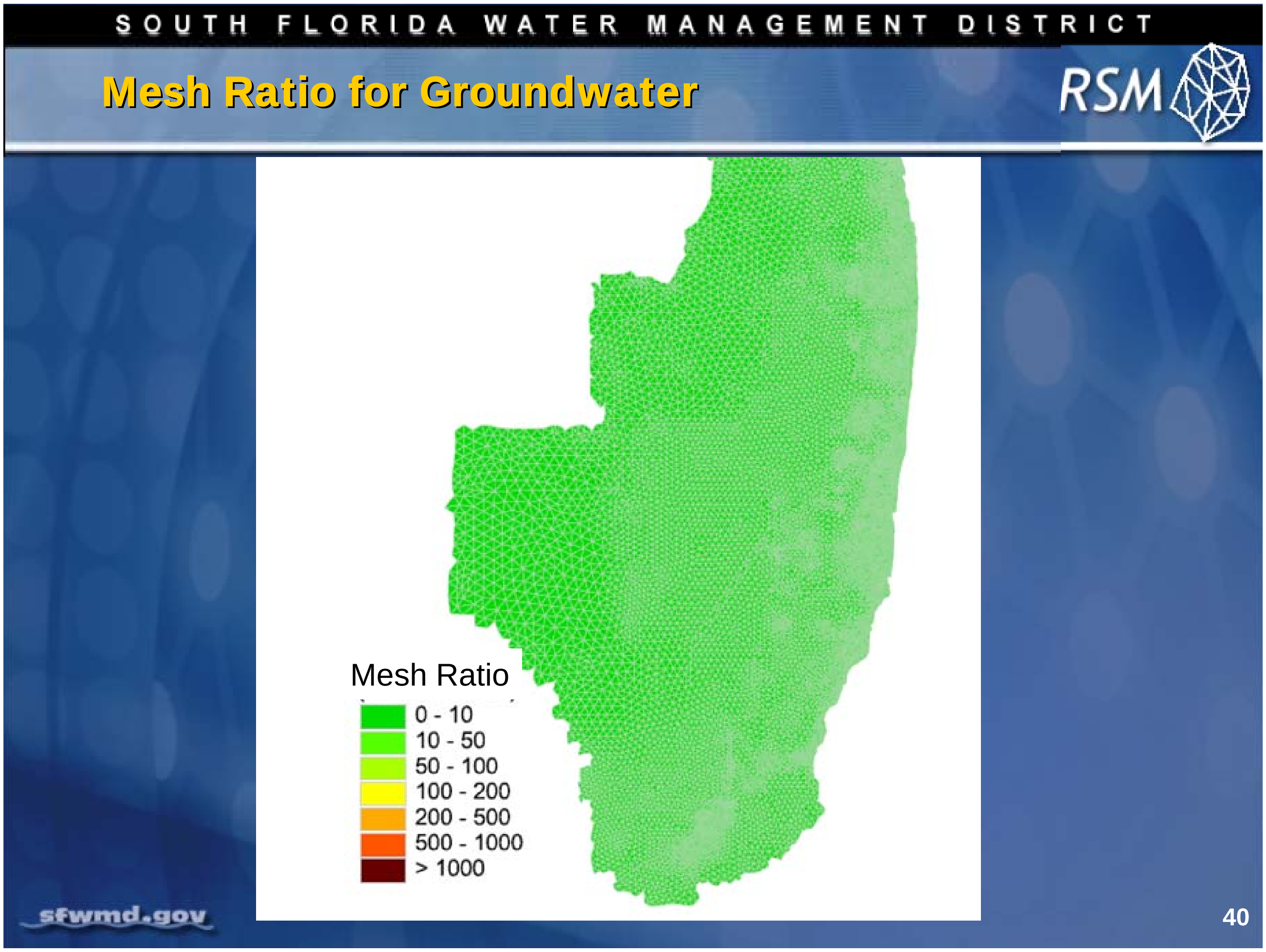
There is a range in mesh ratio (B) values where the residual from the mass balance are acceptably small. For values of B between 10^5 and 10^7, the residuals become marginally acceptable. For B > 10^7, the results from the mass balance will have too much error.



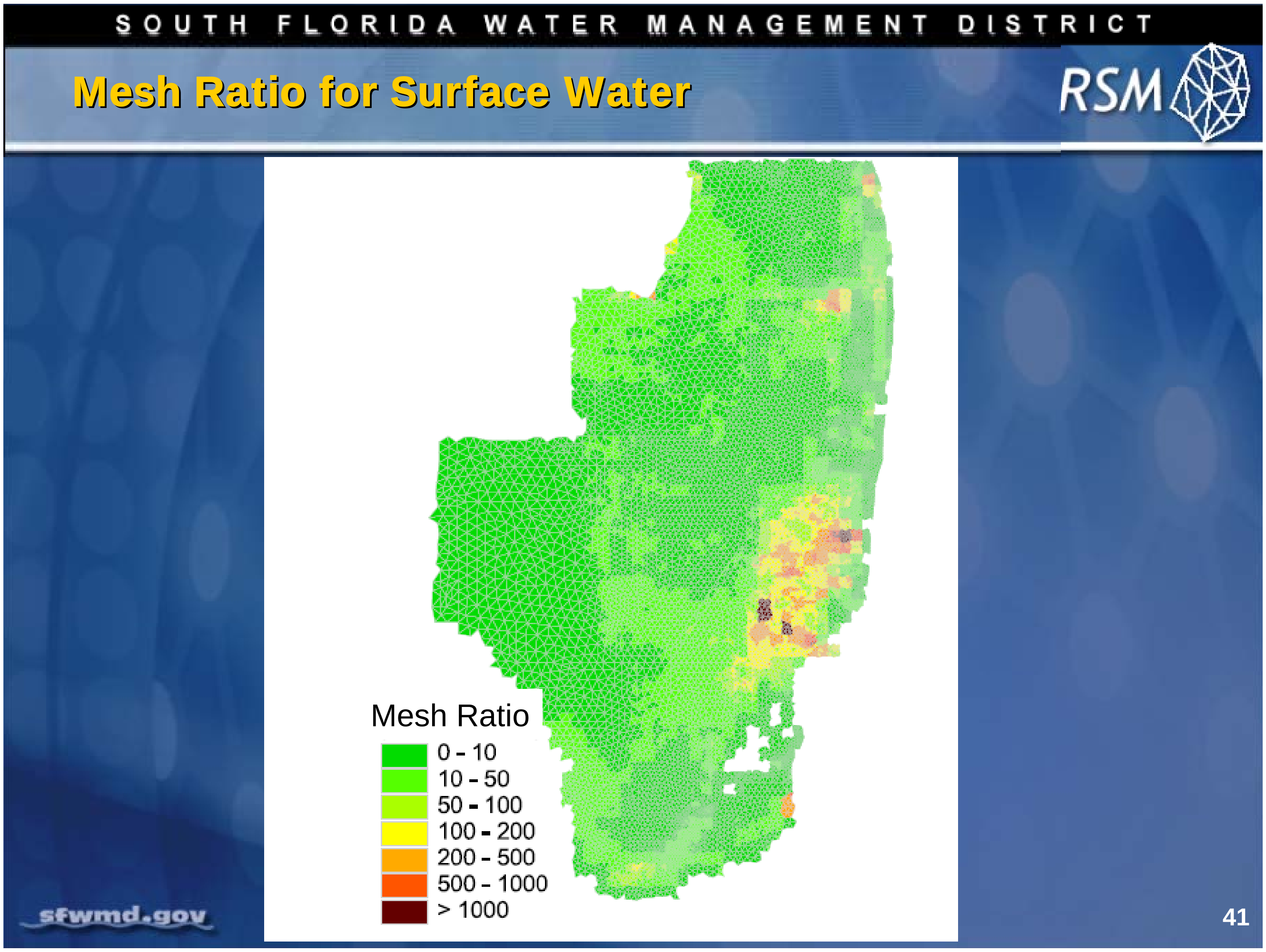
As the mesh ratio B increases, the run time will increase because the sparse matrix solver will require more iterations to achieve an acceptable result.



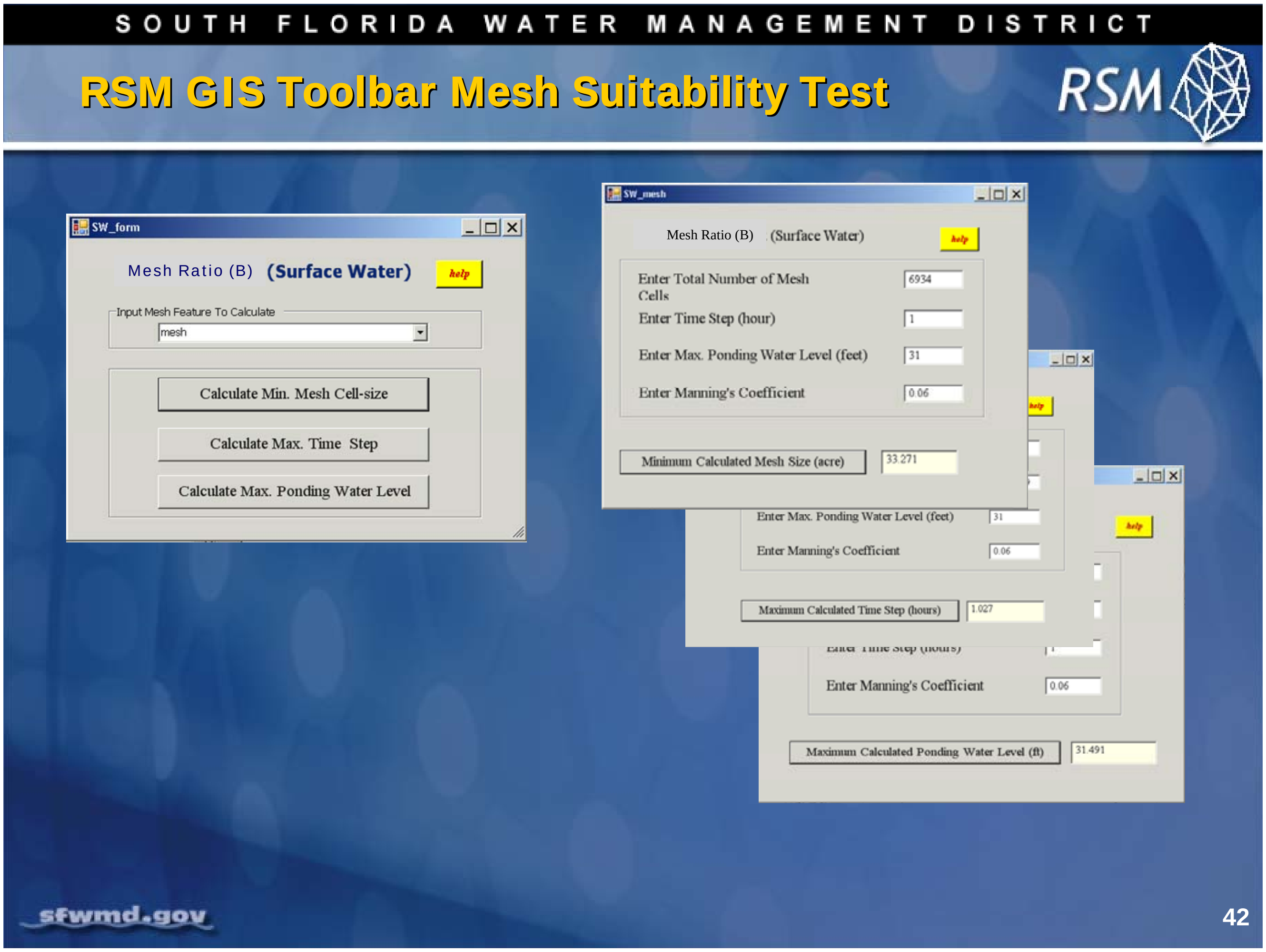
The RSM will run well when the mesh ratio is small. But, when the mesh ratio becomes large, mass balance errors will occur. This is consistent with the conclusion from the signal processing discussion (slides 24-28), that using small cells will not improve resolution, but may introduce errors.



The results of the mesh ratio for groundwater show that values are generally reasonable for the mesh size and timestep we are using.



For surface water, the mesh ratio for a daily timestep is higher, but it is only problematic in a few areas using a regional-scale mesh.



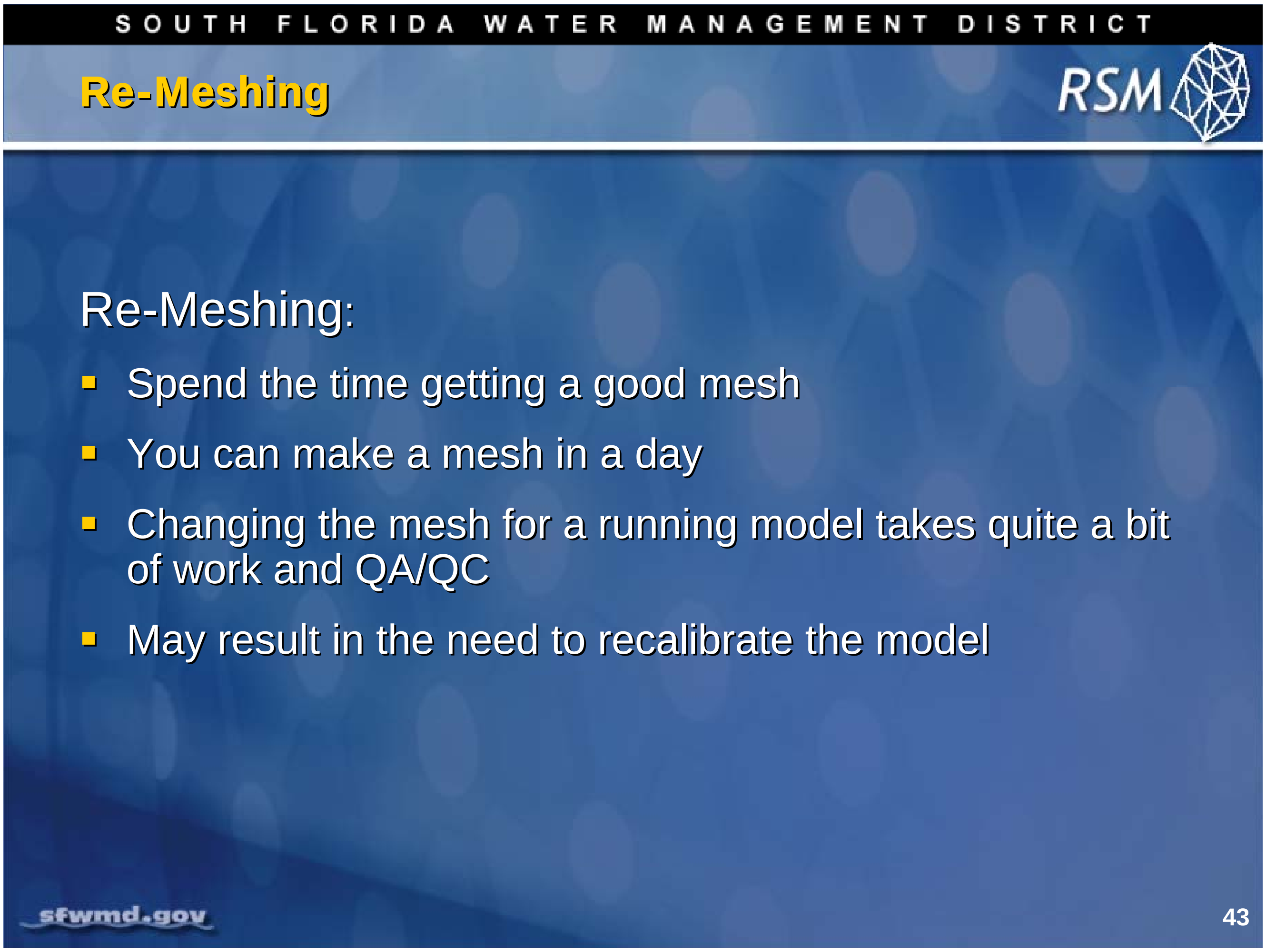
The Mesh Suitability tool can be found on the GIS Toolbar in the Utilities Menu. The tool will automatically calculate the number of cells in your mesh.

Select the mesh layer from your geodatabase as the mesh feature to calculate. There are three options for running the Mesh Suitability tool:

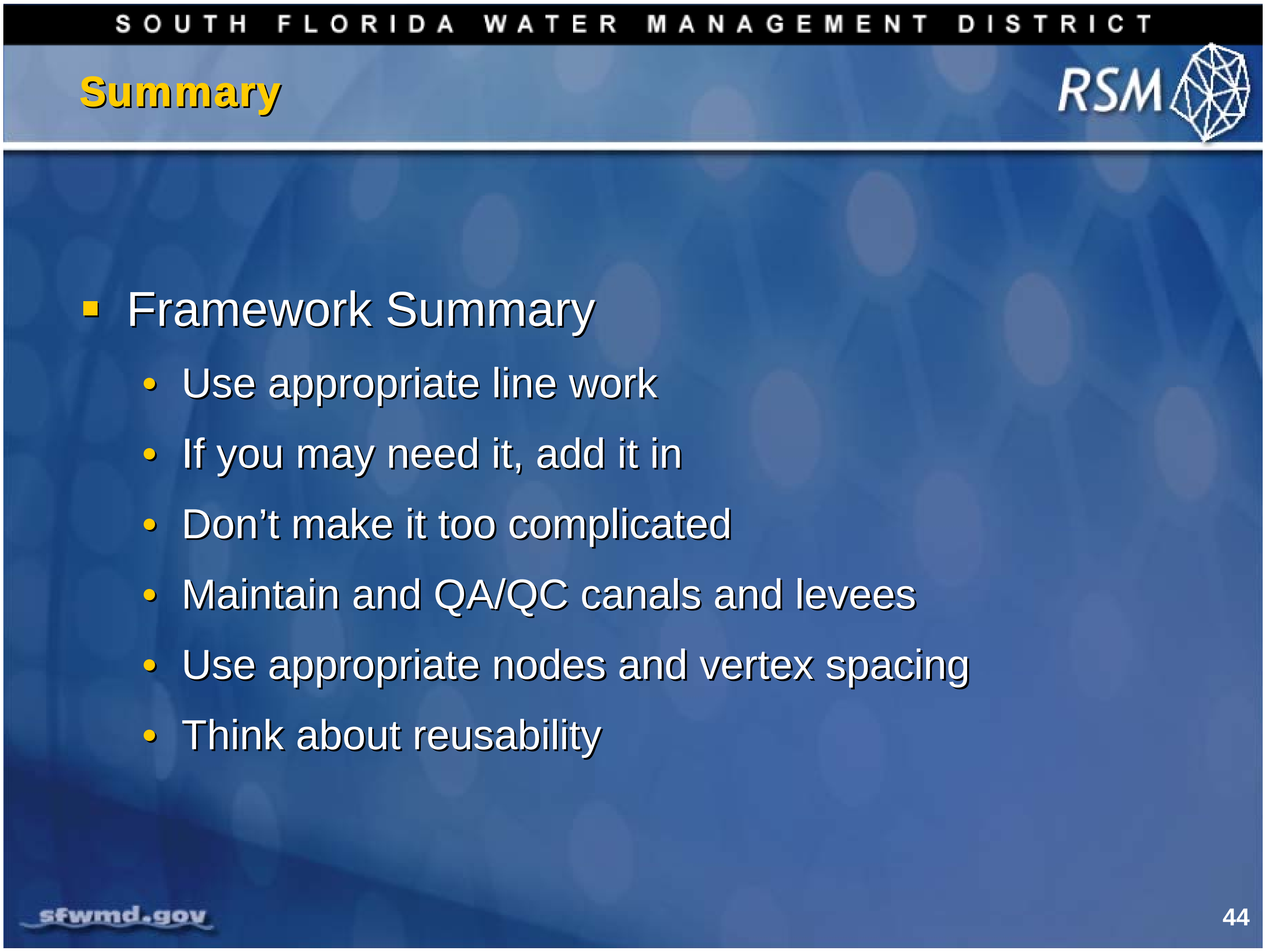
1. Calculate the suitable minimum mesh cell size
2. Calculate the suitable maximum timestep
3. Calculate the maximum ponding water level

Select one of these three options. Enter the timestep that will be used to run your model, the approximate maximum ponding water level you expected in your model run and the average Manning's coefficient used in your model.

The tool will calculate a suitable minimum mesh cell size.

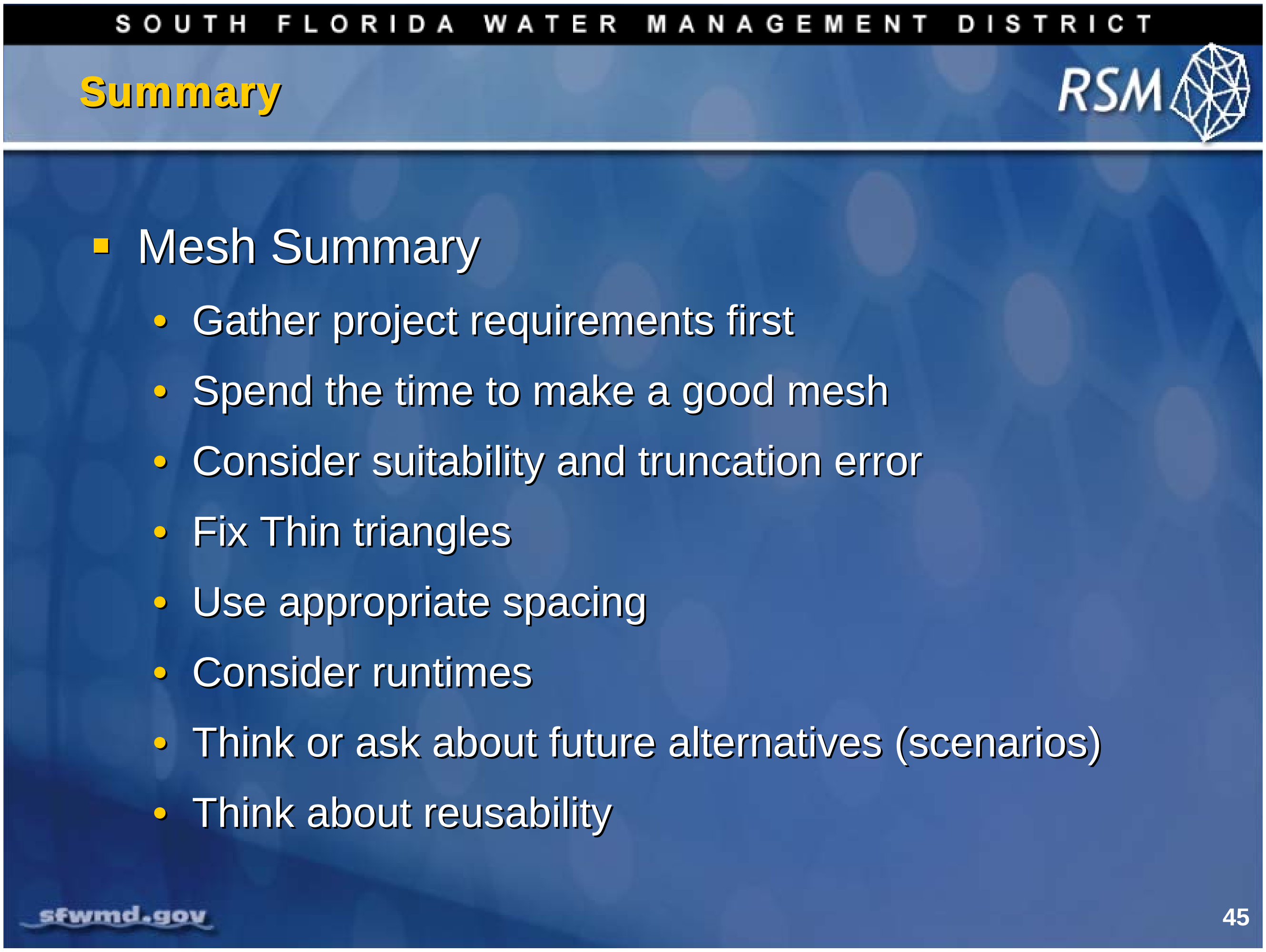


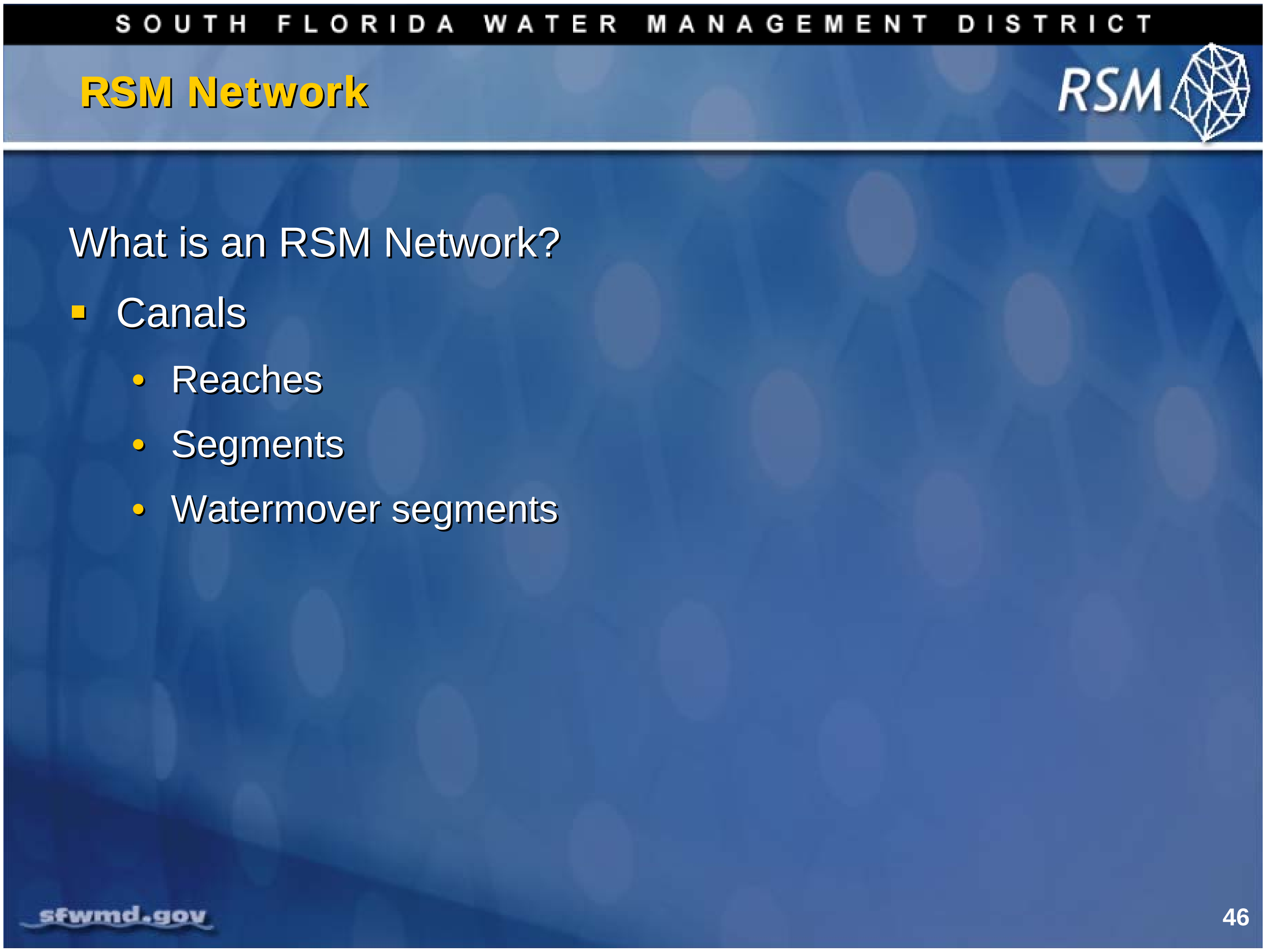
It may require several trials until an appropriate mesh is created.



Development of a sound framework requires careful editing, node and vertices creation. Quality control is also necessary to the formulation of the final product.

Once you create a good framework, remember to save it and archive the file.


SOUTH FLORIDA WATER MANAGEMENT DISTRICT
Summary
RSM
Mesh Summary
Gather project requirements first
Spend the time to make a good mesh
Consider suitability and truncation error
Fix Thin triangles
Use appropriate spacing
Consider runtimes
Think or ask about future alternatives (scenarios)
Think about reusability
sfwmd.gov
45

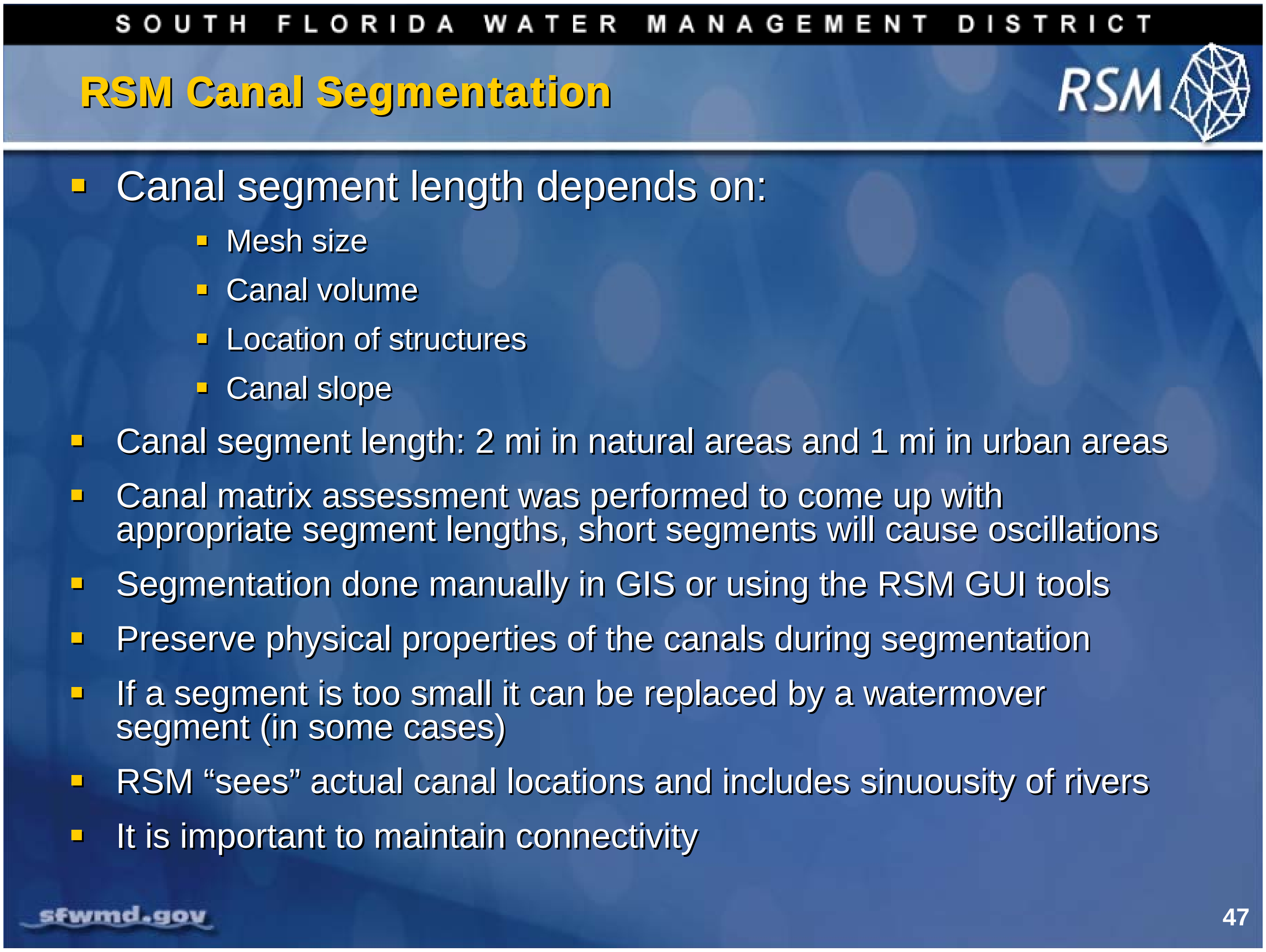


The Regional Simulation Model Network is composed of reaches, segments and watermover segments.

Reaches are a collection of segments that are located between two structures in the canal network. Reaches are used to apply zonal canal properties to groups of segments.

Segments are the waterbodies that make up the canal network. Long canals are segmented to create "consistent" volumes. Segment volume is derived by the Hydrologic Simulation Engine (HSE) based on bottom elevation, side slope, segment length and "current" stage. The current RSM canals are based on "as-built" information and field data; and the RSM canals must closely match their real world locations.

The watermover segment is a specialized segment that is created to connect a diversion structure to a canal segment. It is used to help conceptualize the model and to maintain the default direction of flow in the network, but it does not get "modeled" by the HSE.

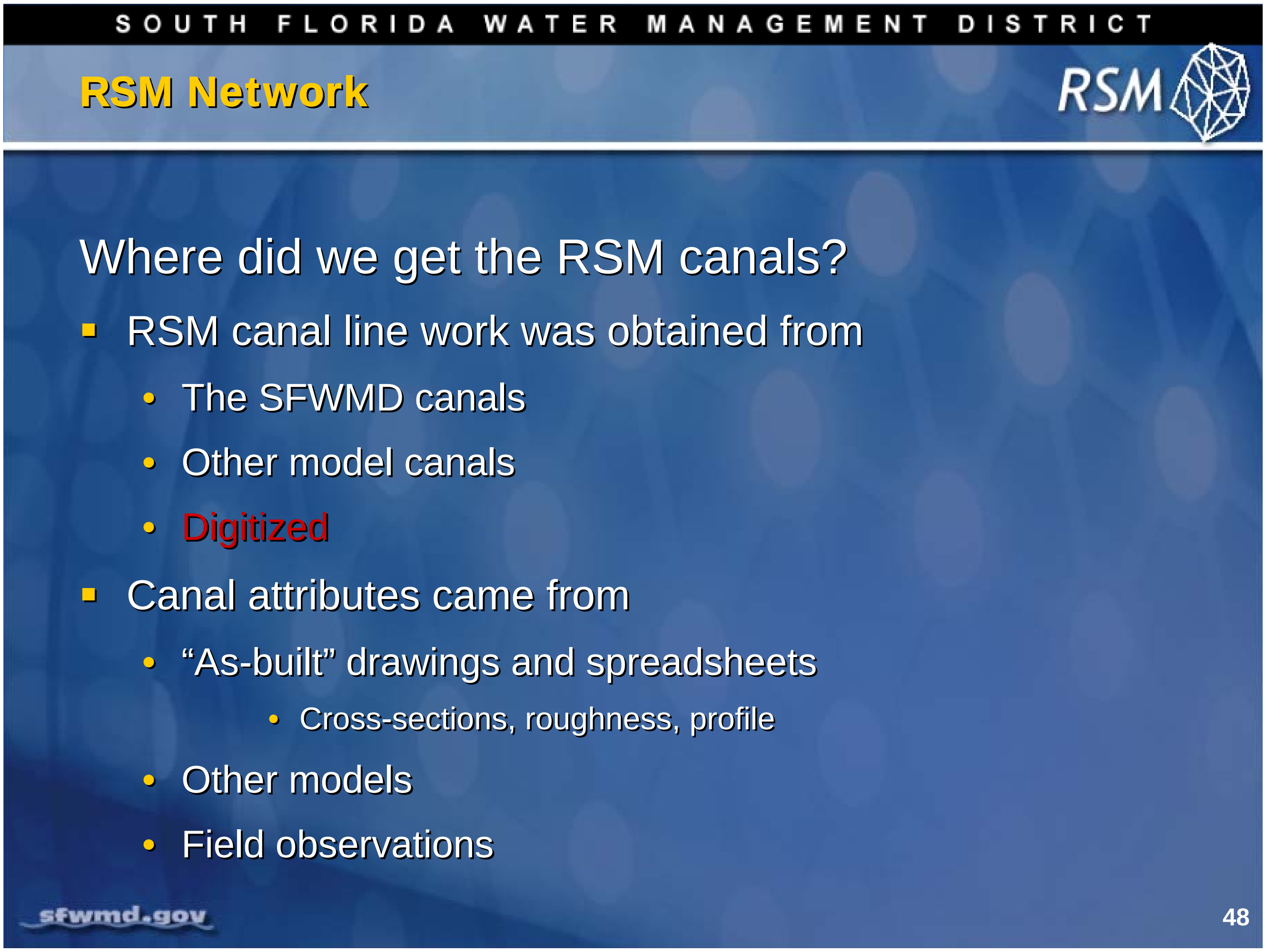


Canal segmentation is an important concern in the RSM.

Segments that are too long will not respond correctly, or effectively, to flow events. Segments that are too short will cause oscillations in flow.

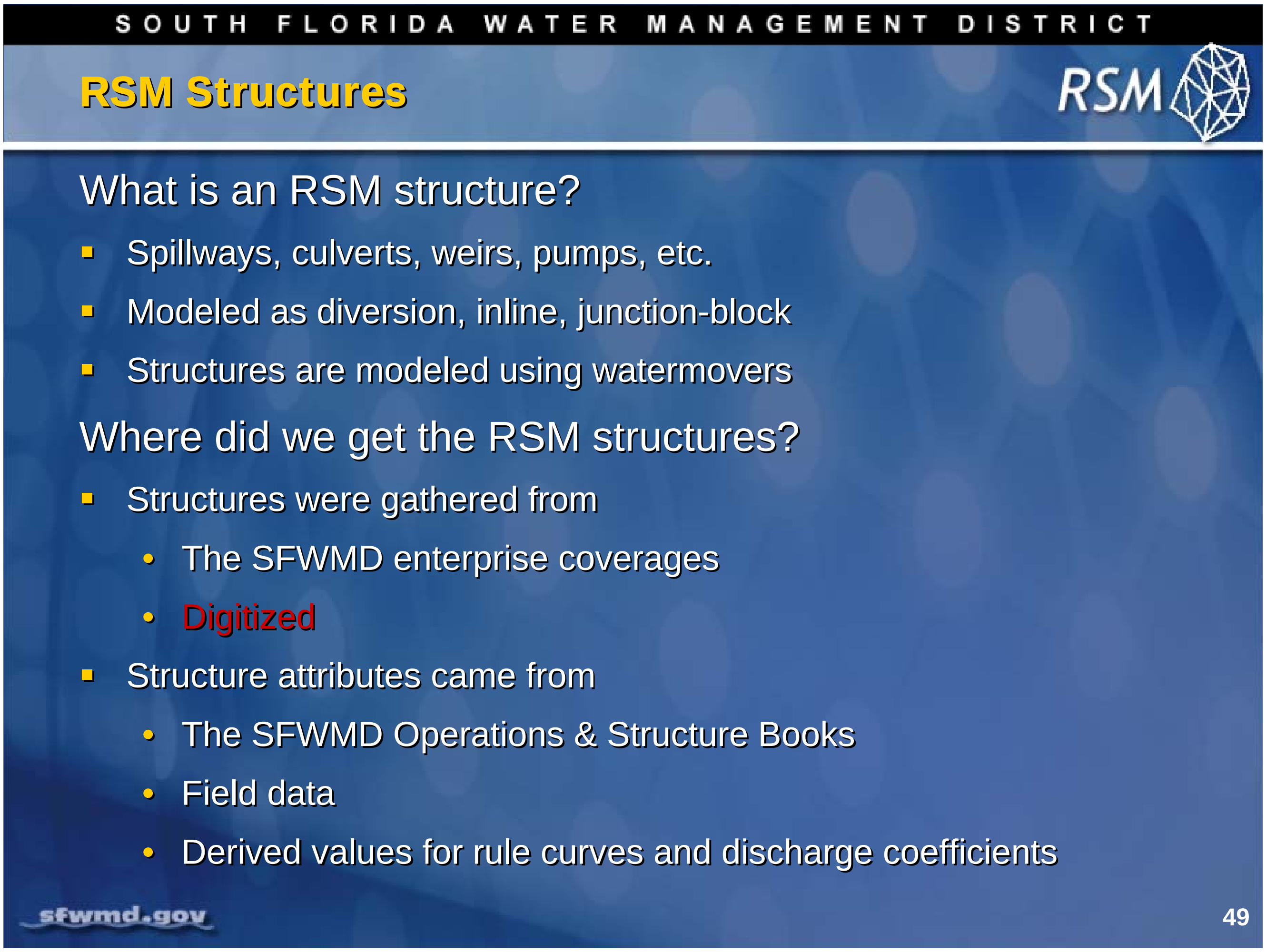
The size of the segments depends on mesh size; segments should be much larger than the cells. Canal segments should be long enough or deep enough to have sufficient volume so that they do not dry out. At least three segments need to be between structures: the steeper the canal, the more segments need to exist.

During the segmentation process of either splitting segments or creating bigger segments, the physical properties of the canal should be maintained. Because the RSM tracks the physical location of the canals, it is important to map the sinuosity of the rivers and canals accurately. The connectivity of the canals must be maintained. If there is a gap in the canal, water will not flow.



Standard canal information for the Regional Simulation Model was obtained from several sources:

- Line work was obtained from the SFWMD canal coverages and other organizations (for secondary canals). Some of the secondary canals and watermover segments had to be digitized. These coverages required considerable review and revision so that the line work was consistent with levee coverages and other GIS coverages.
- Attributes were obtained from the canal documentation. Where documentation was unavailable, field observations were made. The attribute data required careful review and some modification to make the data compatible with the RSM model. For example, canals that passed through small lakes had to be modified to provide a uniform canal segment separated from the lake. Canal attributes that were modified for other models were adapted for the RSM.

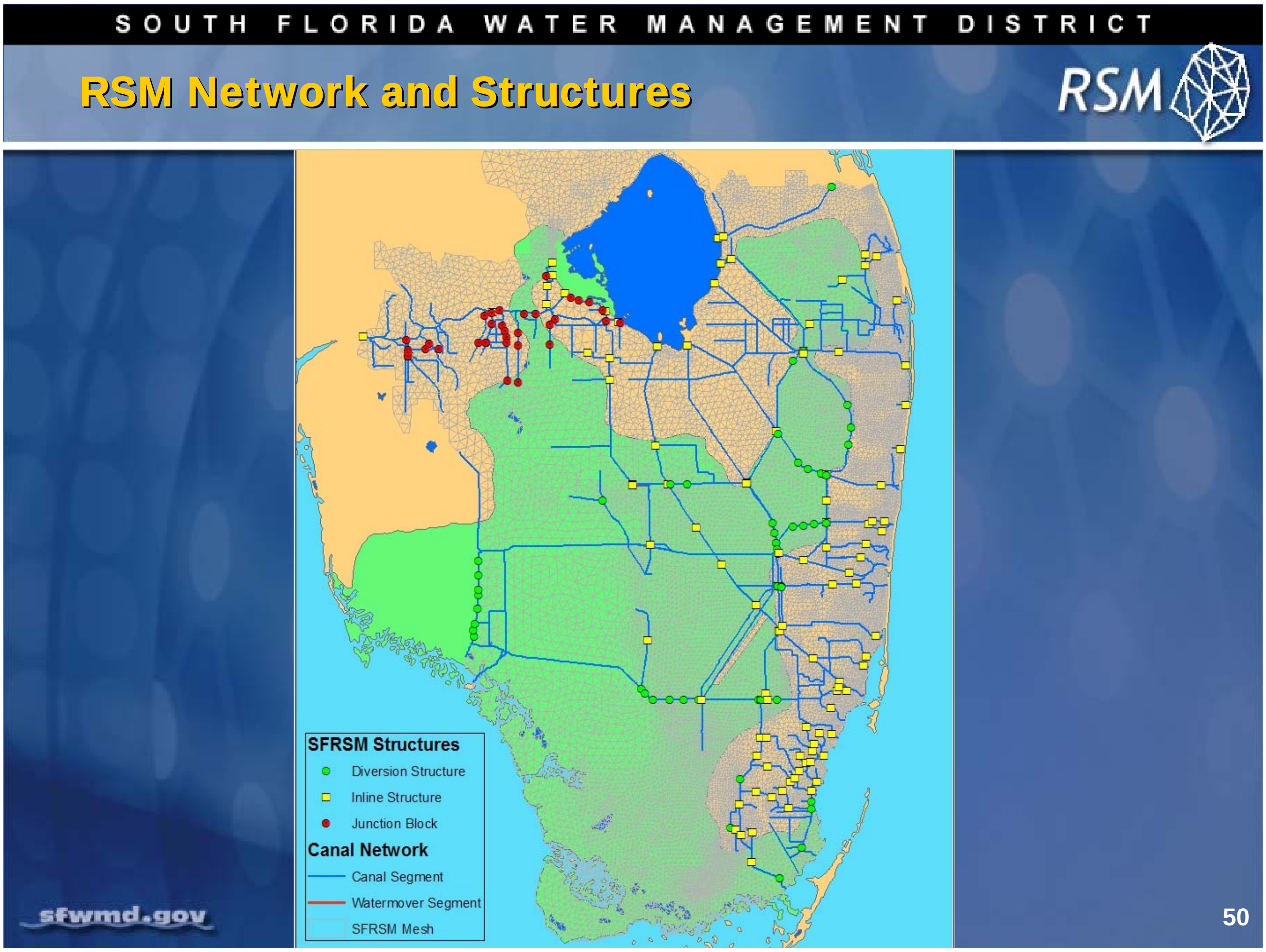


There are many different types of structures connected to the primary and secondary canals in south Florida. These include gated-weirs, pumps, spillways, culverts and other structures.

Many structure locations have more than one water control structure that can be operated separately. A common combination is a pump and a gated-weir. There are also locations with multiple culverts.

The RSM contains 25 types of watermovers that can represent the structures. The watermover types range from culvert and weir equations to generic pump discharge. Lecture 10: Waterbodies and Watermovers, provides a detailed discussion of the watermovers in the RSM.

Data for the RSM structure information was obtained from the SFWMD enterprise geodatabases, Structure Books and the Operations Department. Many structures were digitized. And, considerable effort was expended to verify that structures were accurately located and correctly connected to canal line work.

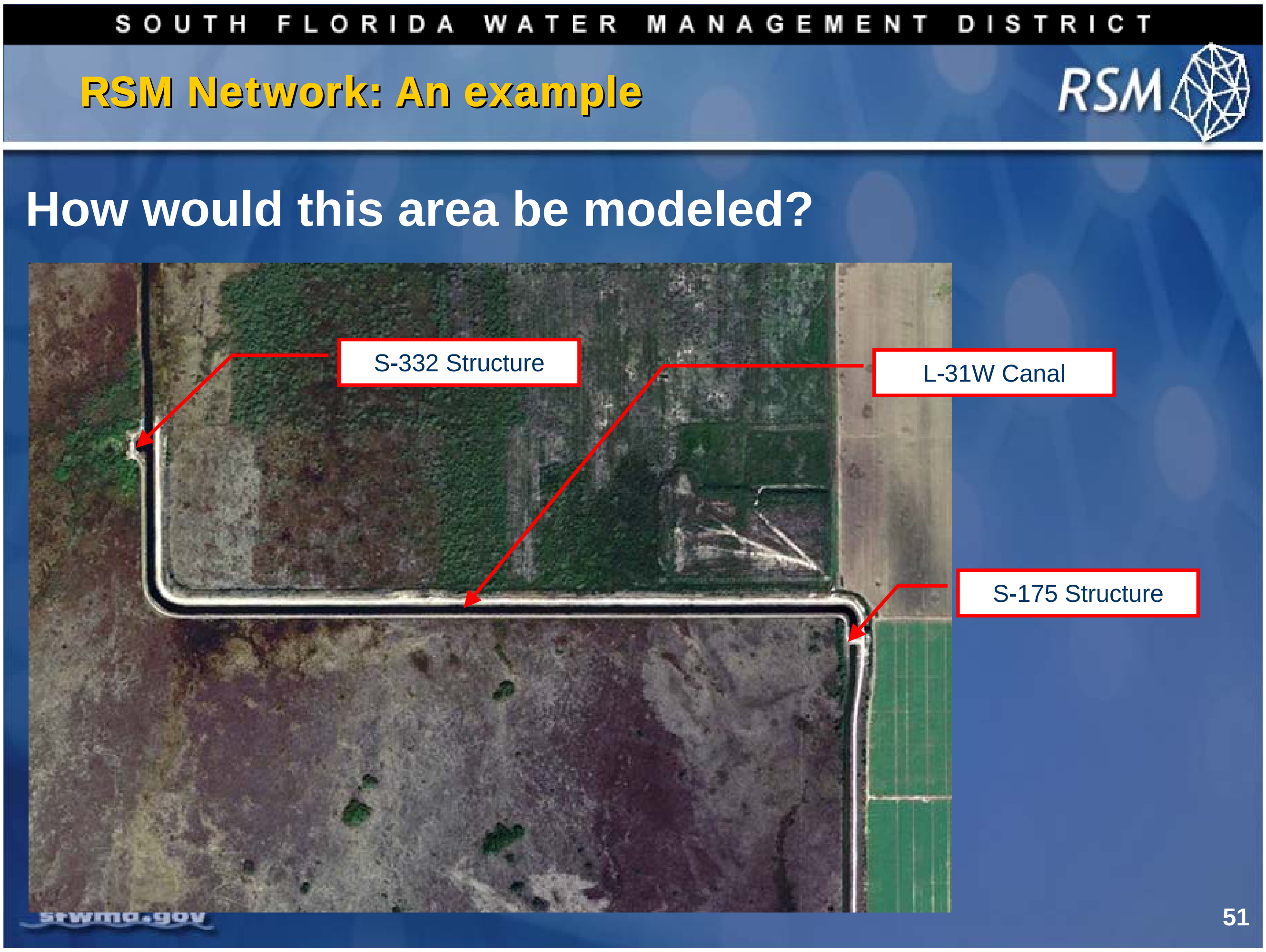


The map in this slide shows the standard SFRSM canal feature class. It includes the primary canal system and significant secondary canals. Other canals may be added to a subregional canal network to obtain the appropriate configuration.

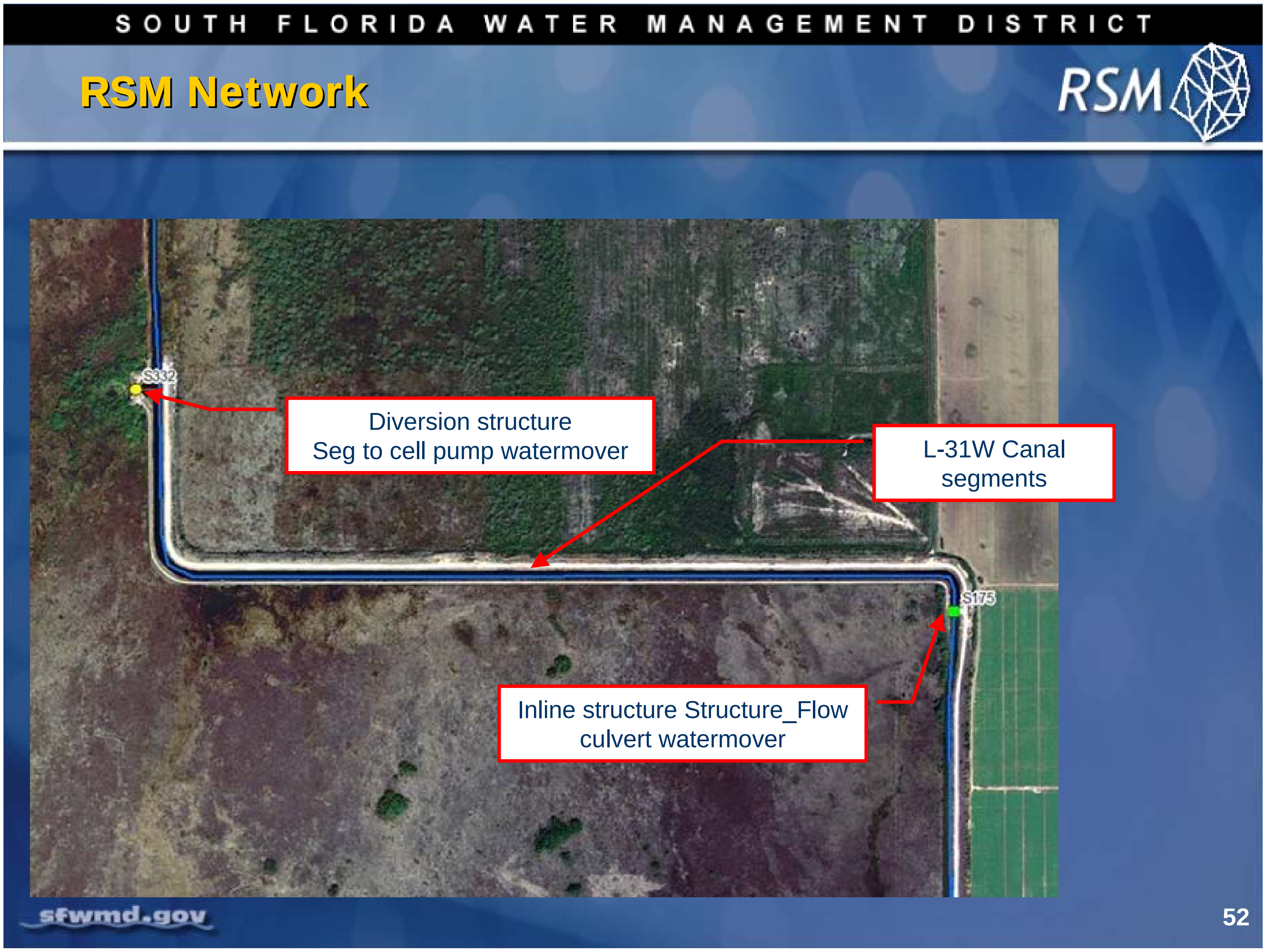
In this example, three types of structures were identified:

- Inline: The typical gated-weir.
- Diversion: Typically culverts that divert water from canals into marshes.
- Junction: Structures with both pumps and weirs.

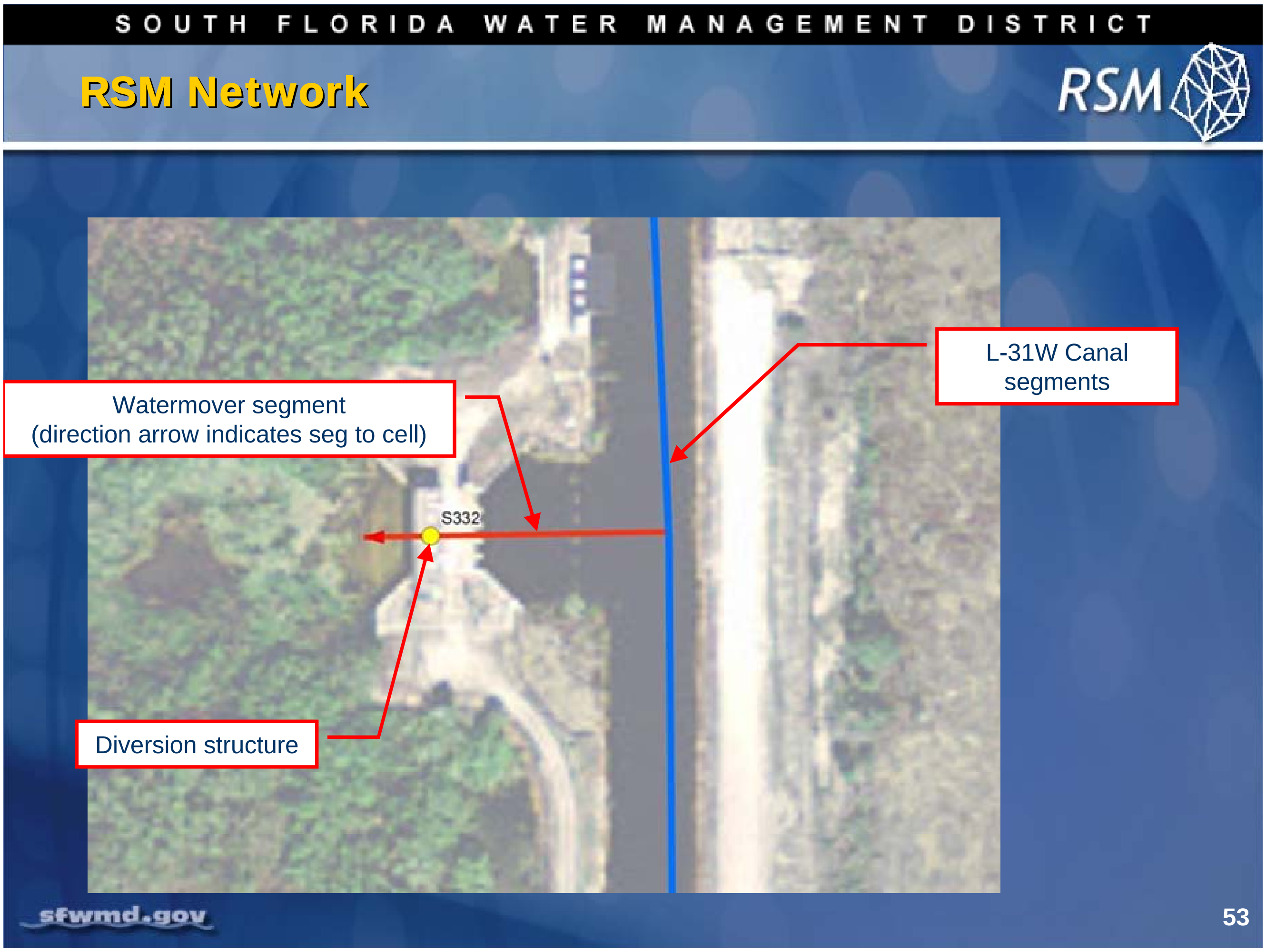
The water level of the upper “pool” is maintained by operating either the pump or the weir to restrict flow or augment flow in the canals. There is rarely any gravity flow along the canal. These are typical operating strategies of the secondary canals in the Caloosahatchee basin.



In this example, this section of the L-31W Canal has two structures.

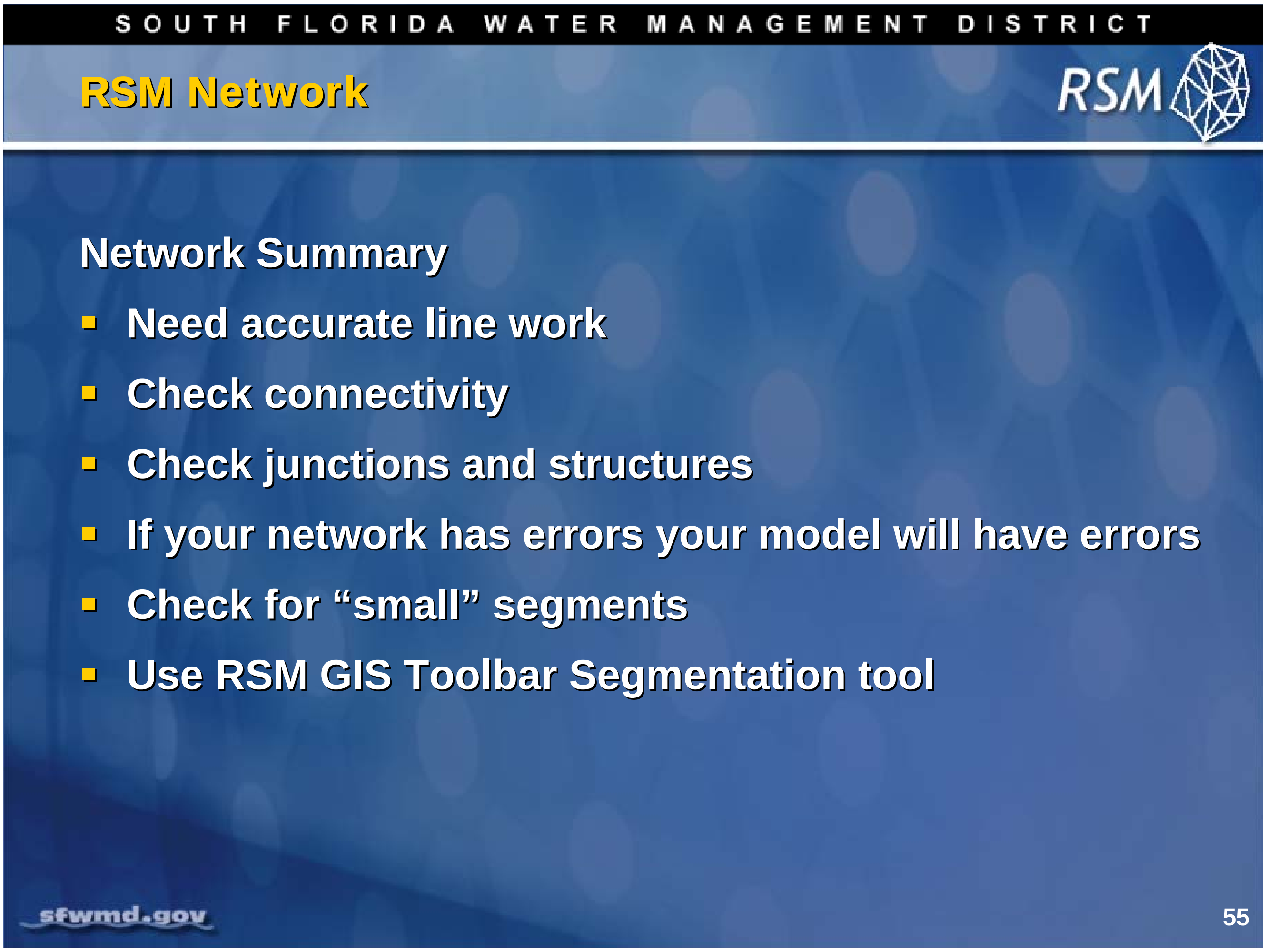


Using high quality aerial photography, the canal and levee line work can be digitized. And, junction blocks can be created at the structure locations.



For the S332 structure, a small watermover segment can be created to link the watermover to the canal. The structure is located at the end of the watermover segment.

In this example, the S175 structure is located at the junction between segments on the L-31W canal.



Accurate line work and connectivity are important concerns for the network. In addition to a visual check, run the RSM with only the network components to ensure all is connected correctly.

KNOWLEDGE ASSESSMENT

(pre- and post-lecture quiz to assess efficacy of training materials)

1. Why is the framework important?
2. What are the components of the framework?
3. What are hazards for frameworks?
4. What are good rules-of-thumb for the density of vertices?
5. What problems can occur with mesh cells?
6. What is the importance of evaluating mesh suitability?
7. What is the range for the mesh ratio to avoid error and excess run time?
8. What problems occur if the mesh ratio is too large?
9. What is the appropriate length for canal segments?
10. What problems occur when the segments are the wrong size?
11. Why is the canal network pre-built?
12. What kinds of structures are included in the canal network?
13. What are the critical sources of RSM error resulting from the canal network?

Answers

1. The framework determines the boundaries of the mesh cells and is critical for ensuring that levees, WCDs, impoundments and hub HPMs can be represented correctly in the model, and water budgets can be calculated for the appropriate area.

2. The components of the framework include levees, major roads, political and project boundaries, and areas for which water budget calculations are required.

3. Problems with the mesh result from poor line-work in the framework and too many constraints or the wrong type of constraints.

4. The density for vertices on the framework for natural areas should be 10,000 ft intervals while the vertex intervals in developed areas should be 2,000 ft.

5. Common problems with mesh cells include thin cells, obtuse cells, poor distribution of cell sizes and cells that don't conform to appropriate boundaries.

6. The concept of mesh suitability is to determine if errors are incurred from having a mesh cell size that is too large or too small in relation to the cell gradient and model time step.

7. The mesh ratio, B, should be between 10^5 and 10^7.

8. With a large mesh ratio there can be problems with mass balance.

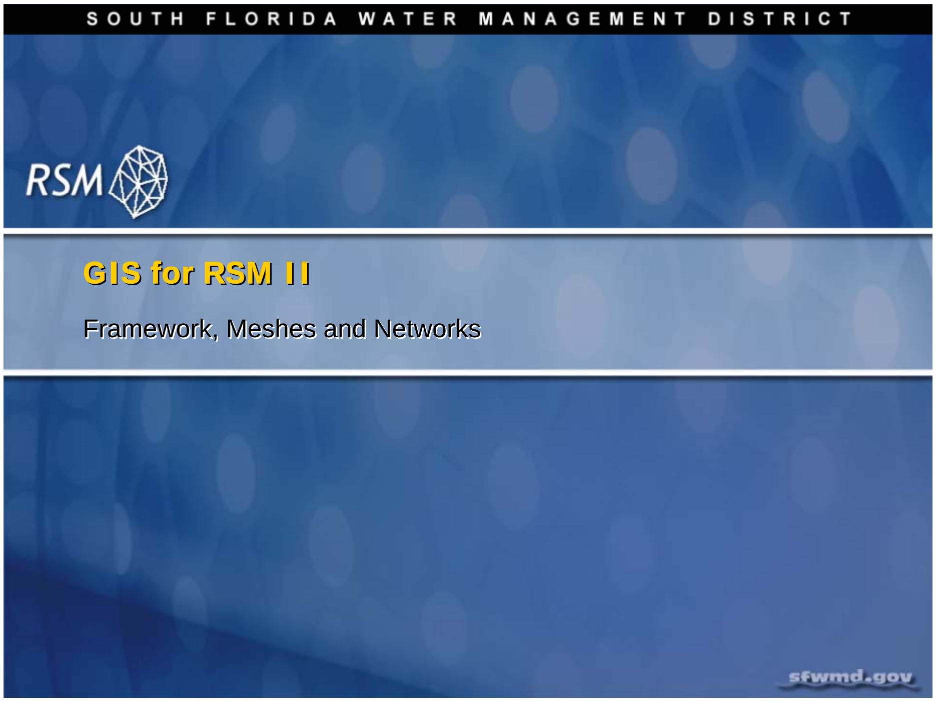
9. Canal segments should be 2 miles in natural areas and 1 mile long in developed areas.

10. Segments that are too small will cause oscillations and segments that are too large will cause numerical errors.

11. The canal network was pre-built because of the difficulty of creating the correct connectivity, including the correct segment properties, re-segmenting, and locating the appropriate structures in the canal network.

12. The canal network includes in-line, diversion and junction structures.

13. RSM errors occur due to inaccurate line-work for the canals, segment hydrologic properties and poor segment lengths.



Lab 6: Create Complete RSM Implementation

Time Estimate: 1.5 hours

Training Objective: Build a complete Regional Simulation Model

In Lab 4, you created a simple Regional Simulation Model (RSM) mesh and canal network with simple boundary conditions. In this lab you will build a complete RSM implementation.

NOTE:

For ease of navigation, you may wish to set an environment variable to the directory where you install the RSM code using the syntax

```
setenv RSM <path>
```

For SFWMD modelers, the path you should use for the NAS is:

```
/nw/oomdata_ws/nw/oom/sfrsm/workdirs/<username>/trunk
```

```
setenv RSM
/nw/oomdata_ws/nw/oom/sfrsm/workdirs/<username>/trunk
```

Once you have set the RSM environment variable to your trunk path, you can use $RSM in any path statement, such as:

```
cd $RSM/benchmarks
```

Training files are currently located in the following directories:

```
INTERNAL_TRAINING
|
|____data
|        |__geographic
|        |__C111
|        |__rain+et
|        |__glades_lecsa
|        |__losa_eaa
|        |__BBCW
|
|____trunk
|        |___benchmarks
|        |___hpmbud
|
|____labs
```

Files for this lab are located in the **labs/lab6_GMS** directory. Additional materials in the directory include:

lab6.wmv

WRR_ERR.pdf

AQUIFER_PARAMETER_0.pdf

PERFORMANCE-1998.pdf

Activity 6.1: Generate a Mesh for a Model Domain

Overview

Activity 6.1 This activity includes two exercises:

- **Exercise 6.1.1** Build a mesh using the Groundwater Modeling System (GMS)
- **Exercise 6.1.2** Import a mesh into the Regional Simulation Model (RSM)

In this activity, you will create a mesh [*.2dm] file using the Groundwater Modeling System (GMS) and then import the mesh into the RSM. If you do not have access to GMS software, read through Exercise 6.1.1, and proceed to Exercise 6.1.2.

Exercise 6.1.1 Build a mesh using the GMS

1. Start GMSv6
2. From the Project Explorer panel, right click to select: **`New`**, **`Conceptual Model`**
   - Name: **`eaamcmesh`** (see **Fig. 6.1**)
   - Type: **`SEEP2D`**
3. Under eaamcmesh, right-click to select: **`New Coverage, Coverage Setup`**
   - Coverage Name: **`eaamcmesh`**
   - From the Sources/Sink/BC Type menu, check: **`Refinement`**
   - On the Areal Properties menu, check: **`Meshing options`**

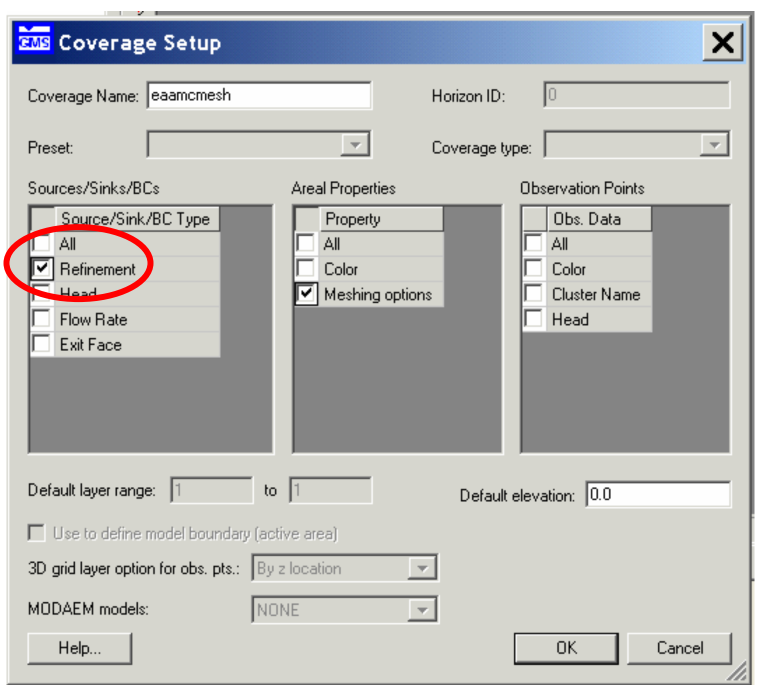


Figure 6.1 Coverage Setup Dialog box

4. From the **File** drop-down menu, select Open: [**`eaaml2_frame_arc_arcs.shp`**]
5. Click the **`Select Elements`** arrow in the static toolbar (see **Figure 6.2**)

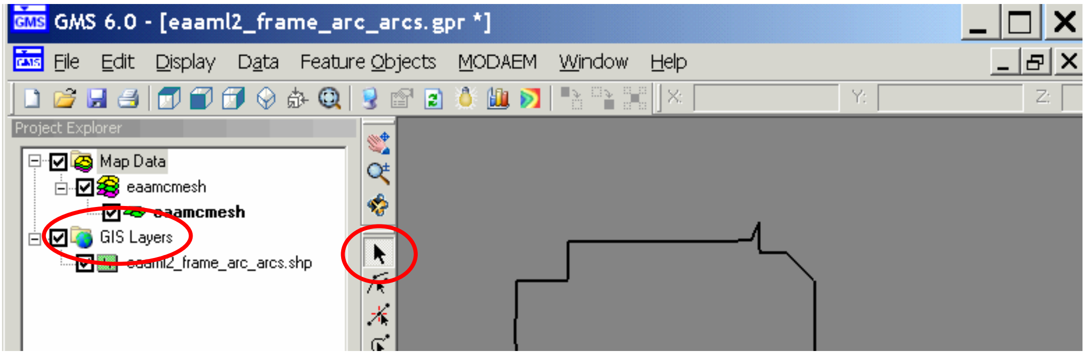


Figure 6.2 Preparing to select elements of the "GIS Layers" shapefile.

6. Select arcs (**Figure 6.3**) that define polygons that surround the area of interest and any additional framework lines such as levees or boundaries that form flow or measurement boundaries.

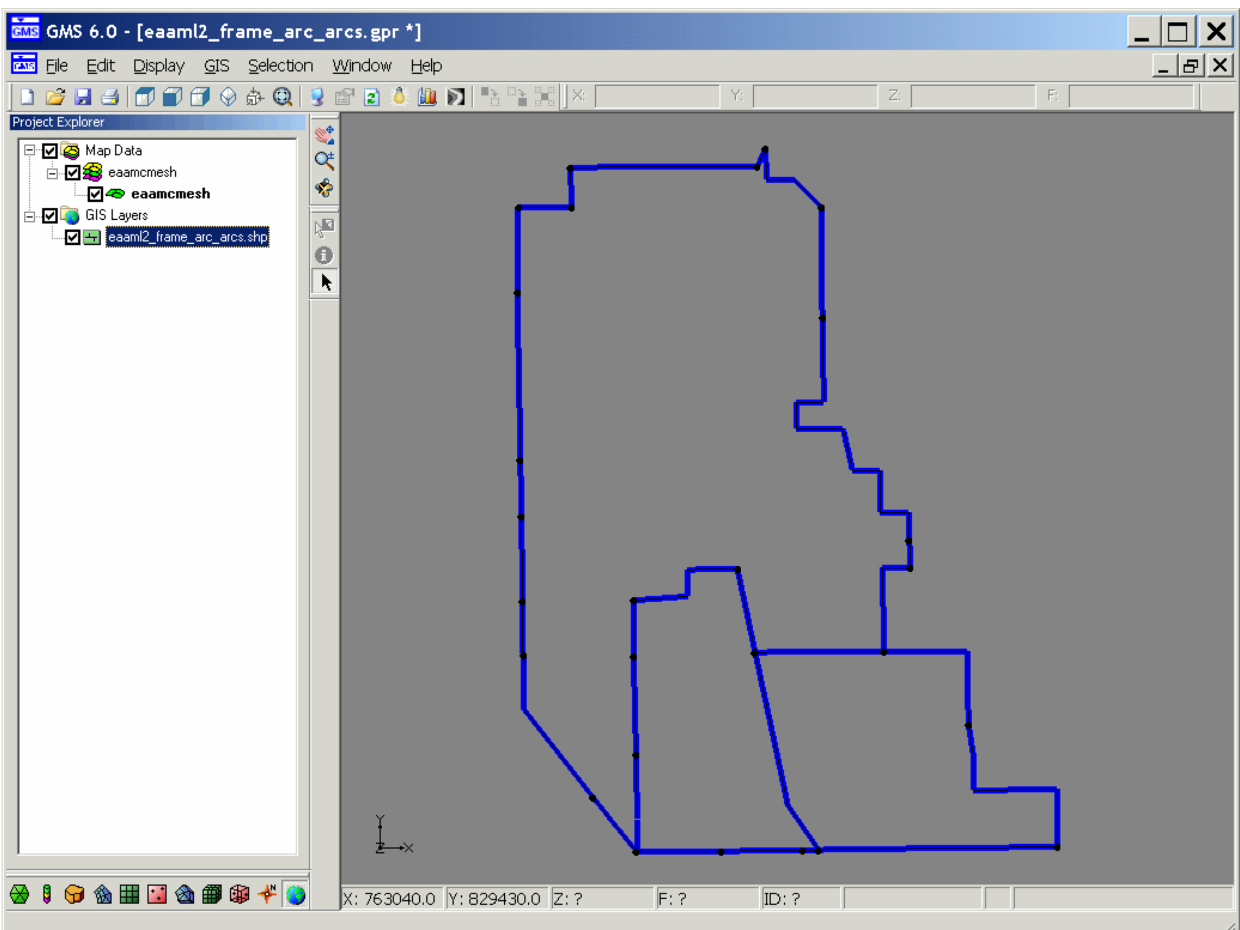


Figure 6.3 Selected arcs that form flow or measurement boundaries.

7. From the GIS drop-down menu, select **`Shapes, Feature Objects`**
   - Select the **`GIS to Feature Objects`** wizard
   - Start wizard: **`Next`**
   - Step 1 of 2: **`Next`**
   - Step 2 of 2: **`Finish`**
8. Save project as **`eaamcmesh.gpr`**

NOTE:

Caution: Save frequently because GMS is prone to crash with large files.

During the mesh creation there is no "undo" for many steps. In some cases, it may be necessary to discard your edits and start the process with a new version of `eaamcmesh.gpr`.

9. In Map mode, select **`Feature Objects, Clean`** (see **Figure 6.4**)

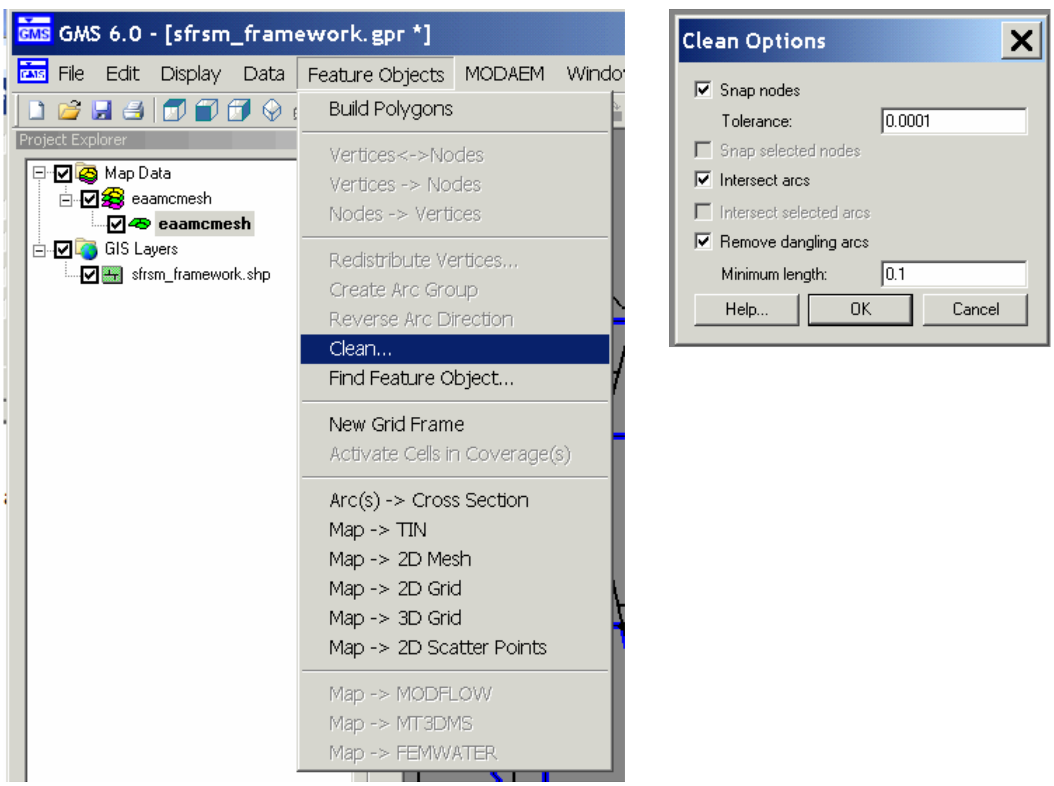


Figure 6.4 Map Mode

10. From the **`Feature Objects`** drop-down menu, select **`Build Polygons`**.

11. Save this project (we are going to create a poor quality mesh and will want to come back to this project to revise it later).
12. From the **`Display`** drop down menu, select the **`Display Options`** dialog box. Check the **`Vertices`** option (see **Figure 6.5**). Results are shown in **Figure 6.6**.

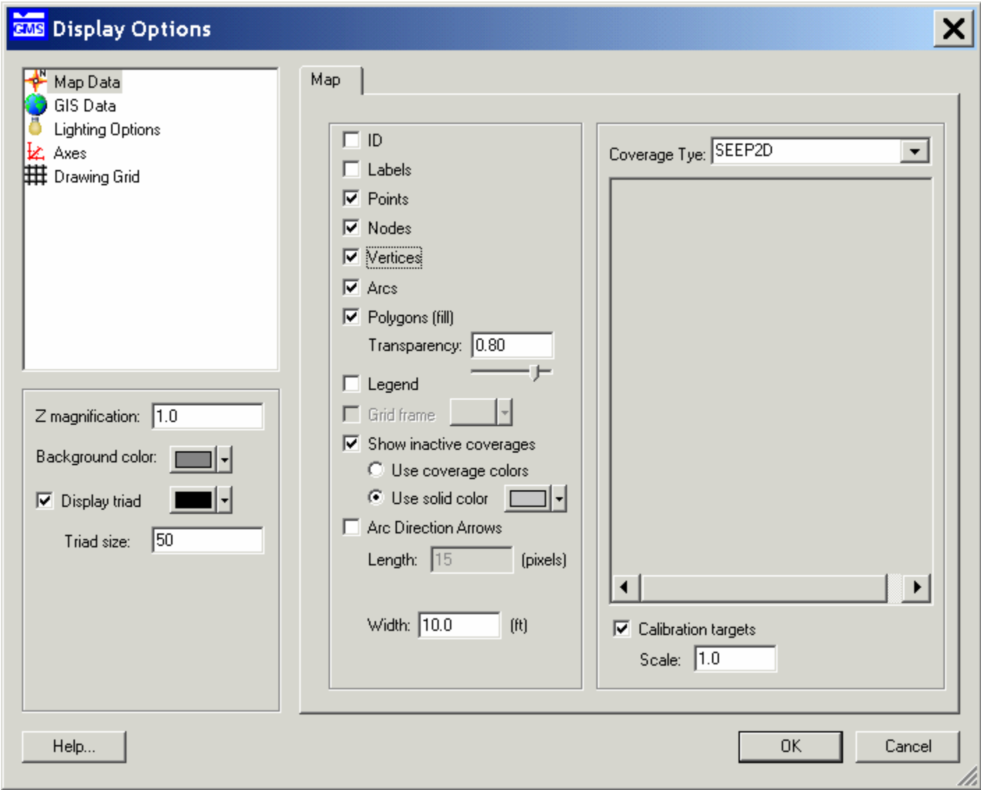


Figure 6.5 Display Options dialog box

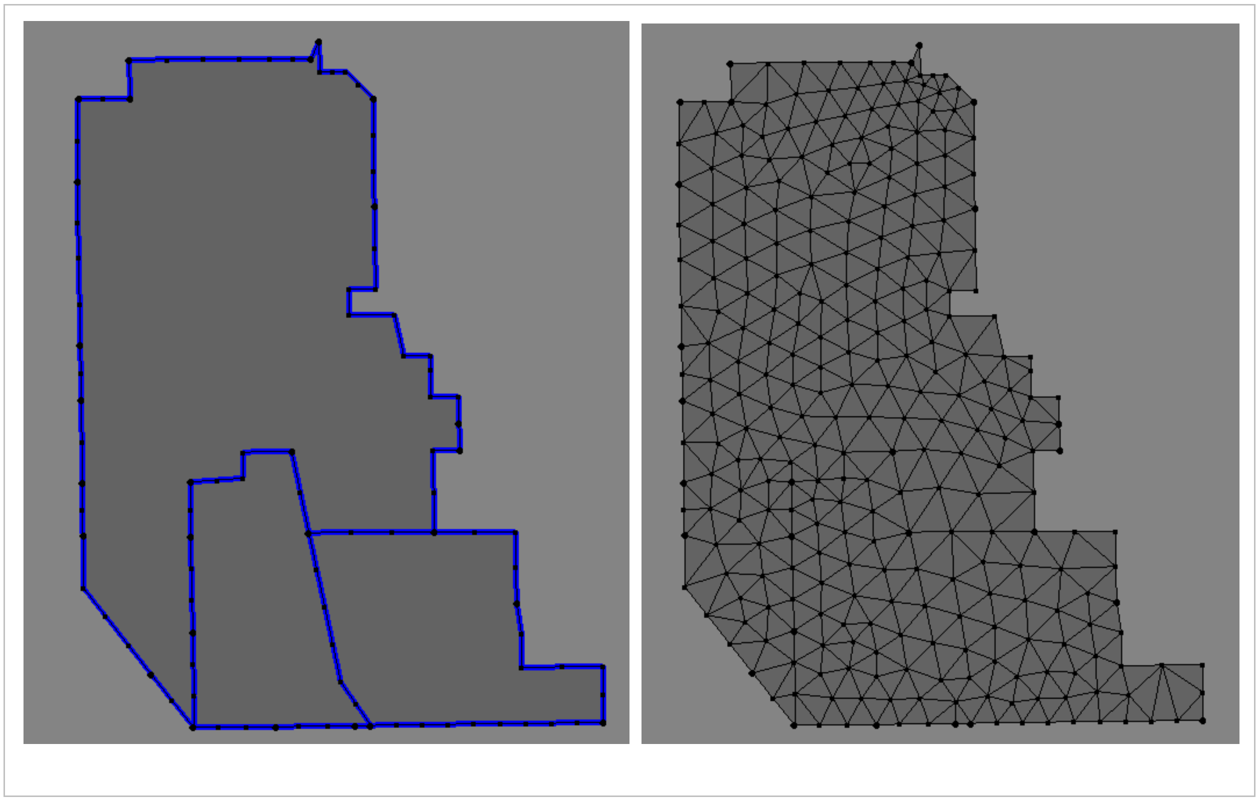


Figure 6.6. Vertices which will be used to generate the mesh (l) and a resulting mesh (r).

13. Edit Redistribute vertices:

    There are two options for redistributing vertices:

    - Addition and removal of selected vertices
    - Use the automated tool for redistributing vertices

    The **`eaaml2_frame_arc_arcs.shp`** has a reasonable distribution of vertices; but, depending on which arcs are selected for the current model framework some vertices may not be useful. These vertices can be removed by selecting the vertices and deleting them. Additional vertices may be added where necessary to create the mesh:

14. In **`Map Mode`**, select **`Feature Objects`**.
15. Select **`Map, 2D Mesh`**.

16. Observe results:

    If the result is a poor distribution of mesh cells, select **`New`** project and reopen **`eaamcmesh.gpr`** and edit vertices.

    - Create new mesh.
    - Automated vertices redistribution.
    - Select **`eaamcmesh`** coverage.
    - Click **`Select Elements`** in the static toolbar
    - In the **`Edit`** Menu, click **`Select All`**

17. From the **`Feature Objects`** drop-down menu, select **`Redistribute Vertices`** (see **Figure 6.7**):

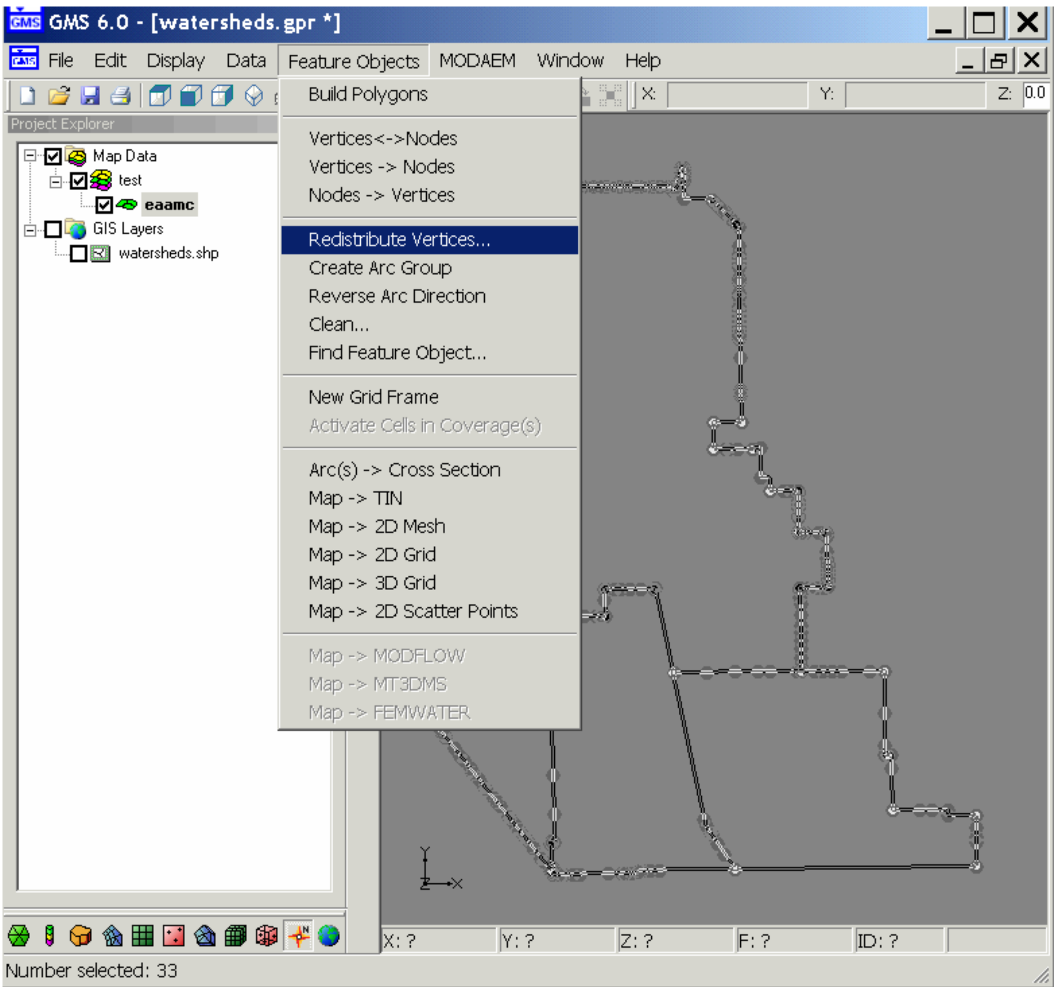


Figure 6.7 **`Redistribute Vertices`** window

- Specify **`Min/Max Spacing`** from the **`Arc redistribution`** drop down menu in the **`Redistribute Vertices`** dialog box (see **Figure 6.8**)

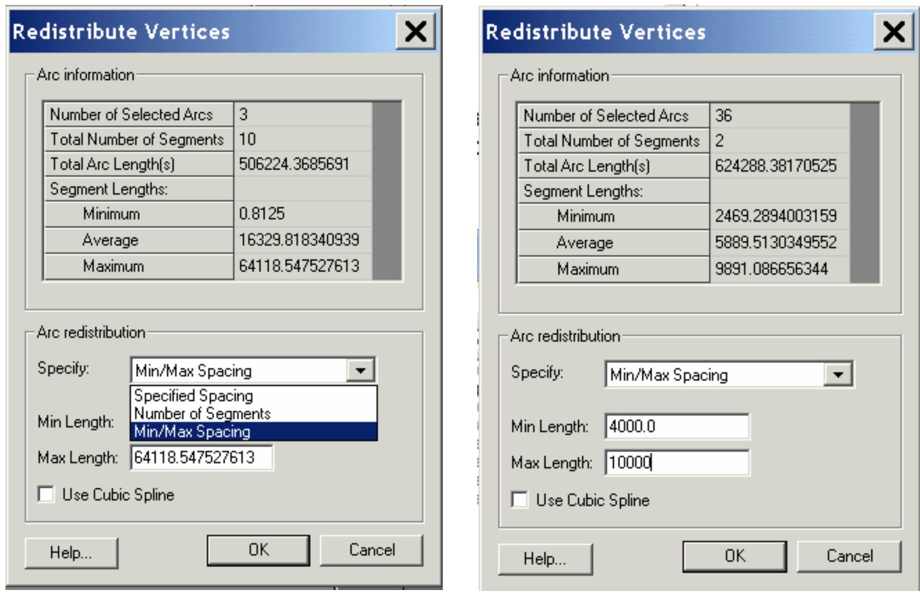


Figure 6.8 **`Redistribute Vertices`** dialog box

- Enter **`Min Length`**: **`5000`**
- Enter **`Max Length`**: **`15000`**

18. In **`Map Mode`**, select **`Feature Objects`**.
19. Select **`Map, 2D Mesh`**. Results are shown in **Figure 6.9**.

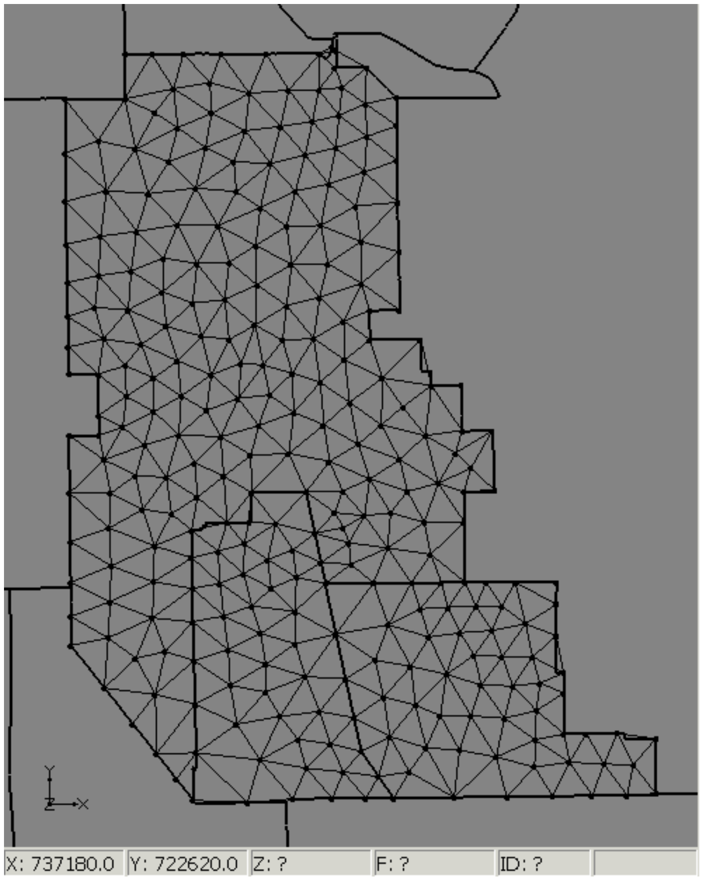


Figure 6.9 A new 2D mesh based on the redistributed vertices.

20. In the **Project Explorer** panel, highlight "**2D Mesh Data**"
21. Right-click **2D Mesh Data, Export**
22. In **Export 2D Mesh**, create **eaamc4.2dm** file (see **Figure 6.10**).

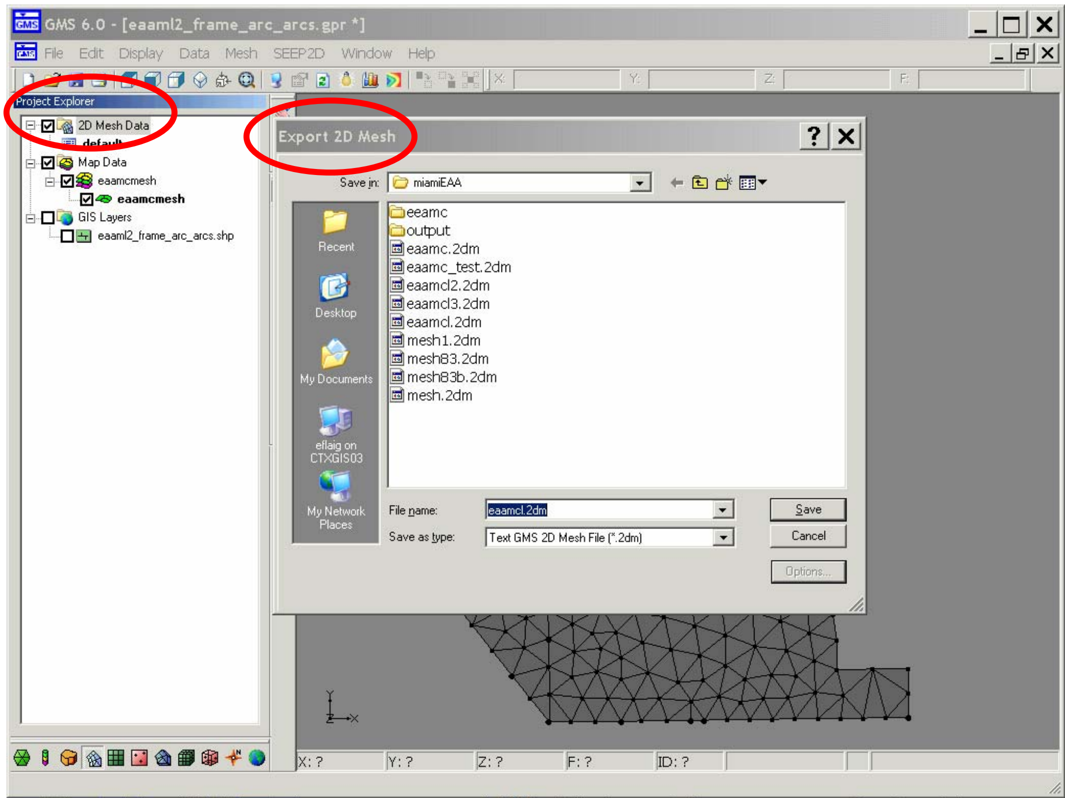


Figure 6.10 **Export 2D Mesh** dialog box

23. Save and exit GMS.

Exercise 6.1.2 Import a mesh into the RSM

The objective of this exercise is to import the mesh into the Regional Simulation Model (RSM). If Exercise 6.1.1 was not completed, use **`$RSM/labs/lab6_GMS/eaamc3.2dm`** rather than the **`eaamc4.2dm`** created in Exercise 6.1.1.

Once the mesh is created based on the SFRSM standard framework in GMS format, it can be used to create the necessary RSM attribute files.

24. Open ArcMap and open RSM GIS ToolBar.
    - From the RSM GIS ToolBar, select Mesh, select Import Mesh, select Load SFRSM Template, as showing in **Figure 6.11**.

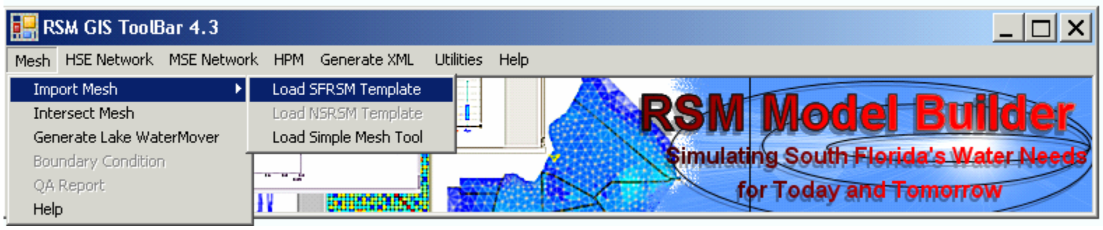


Figure 6.11 RSM GIS ToolBar

25. Implement **Complex Import** as displayed in **Figure 6.12**:
    - Select Template:

      `$RSM/data/geographic/geodatabase_templates/mesh_import_template.mdb`

    - Input .2DM File:

      `eaamc3.2dm` or `eaamc4.2dm` (from Exercise 6.1.1 or from `$RSM/labs/lab6_GMS` directory)

    - Input FrameWork Shapefile:

      `$RSM/labs/lab6_GMS/eaaml2_fram_arc_arcs.shp`

    - Output GeoDatabase:

      `$RSM/labs/lab6_GMS/eaamc3.mdb`

    Processing can be observed on the ArcMap status line. When complete, close **`Complex Import`**.

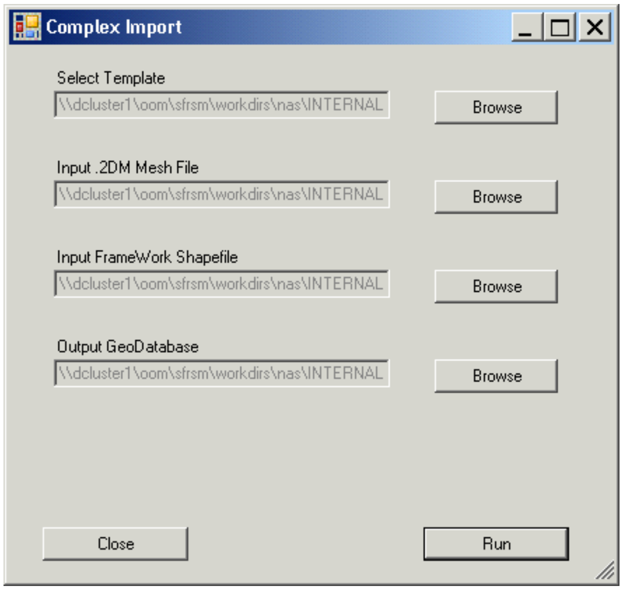
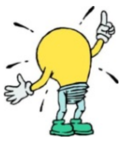


Figure 6.12 `Complex Import` menu interface.

26. When processing is complete, uncheck a few layers to hide them so that you can see your mesh.

HINT



Turn off watersheds and framework.
Right-click the mesh layer on the attribute list (right side of GIS window).
Select Zoom to layer option.
You now have a base RSM geodatabase containing your mesh and base RSM data for developing your model.

27. Click on some of the features using the ESRI **`Identify`** tool ⓘ and examine the attributes in the geodatabase, as shown in **Figure 6.13**.

    The attributes from the **`template_geodatabase`** only include the cell, nodes and cell-ID. In the next activity, the mesh attributes are added.

28. Save your map file and call it **`lab6.mxd`**, saving it in your **`lab6_GMS`** directory.

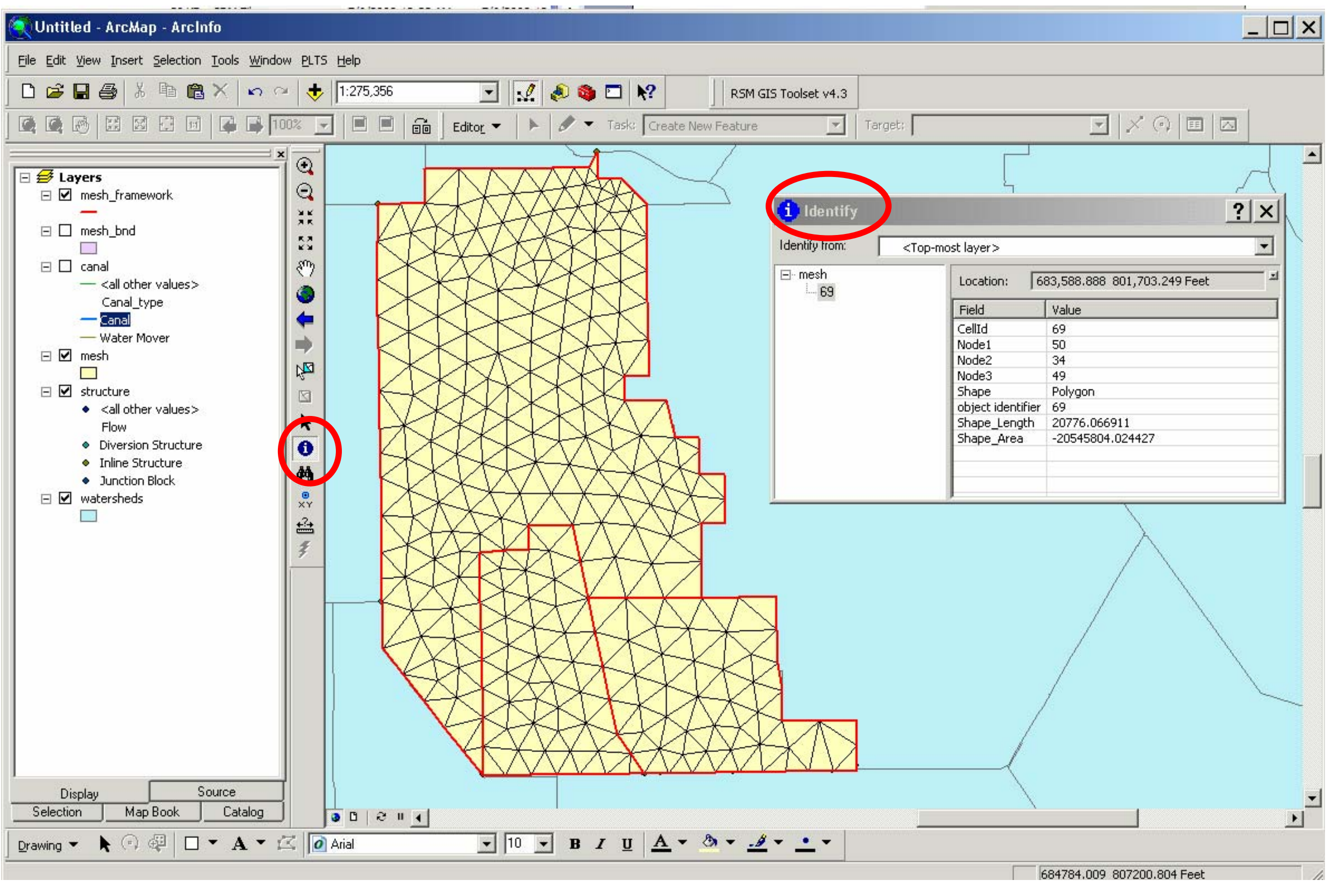


Figure 6.13 ArcMap – ArcInfo window with **`Identify`** tool

Activity 6.2 Add spatial attributes to an RSM model

Overview

Activity 6.2 This activity includes two exercises:

- **Exercise 6.2.1** Add spatial data to ArcMap Mesh
- **Exercise 6.2.2** Add mesh attributes to the RSM

In this activity you will intersect the mesh with attributes from other GIS coverages and generate indexed attribute files which the RSM uses for importing spatial data.

If you do not have access to ArcMap software, there is a video file (**`lab6.wmv`**) in the **`lab6_GMS`** directory that demonstrates Exercise 6.2.1.

Exercise 6.2.1 Add spatial data to ArcMap Mesh

The first step in adding spatial data to the Regional Simulation Model is to extract the data from geographic coverages and add the data to the mesh you created.

29. While still working in ArcMap with **`lab6.mxd`**, click the Intersect Mesh tool in the Mesh drop-down menu on the RSM GIS ToolBar (see **Figure 6.14**).

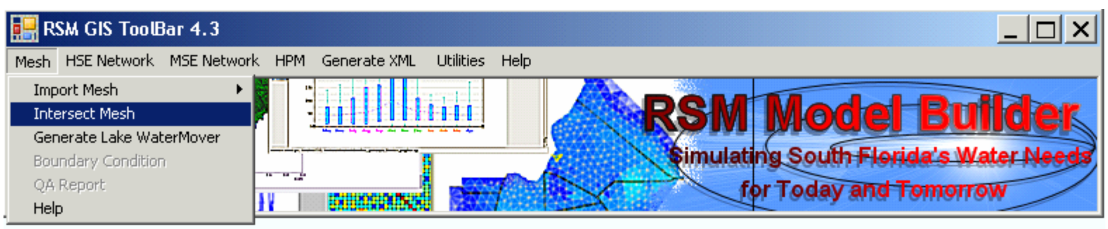


Figure 6.14 RSM GIS ToolBar

30. Select the feature class you want to populate from the **`$RSM/data/geographic`** directory. These include topography (shown in **Figure 6.15**), landuse/land cover, hydraulic conductivity and bottom of the aquifer.

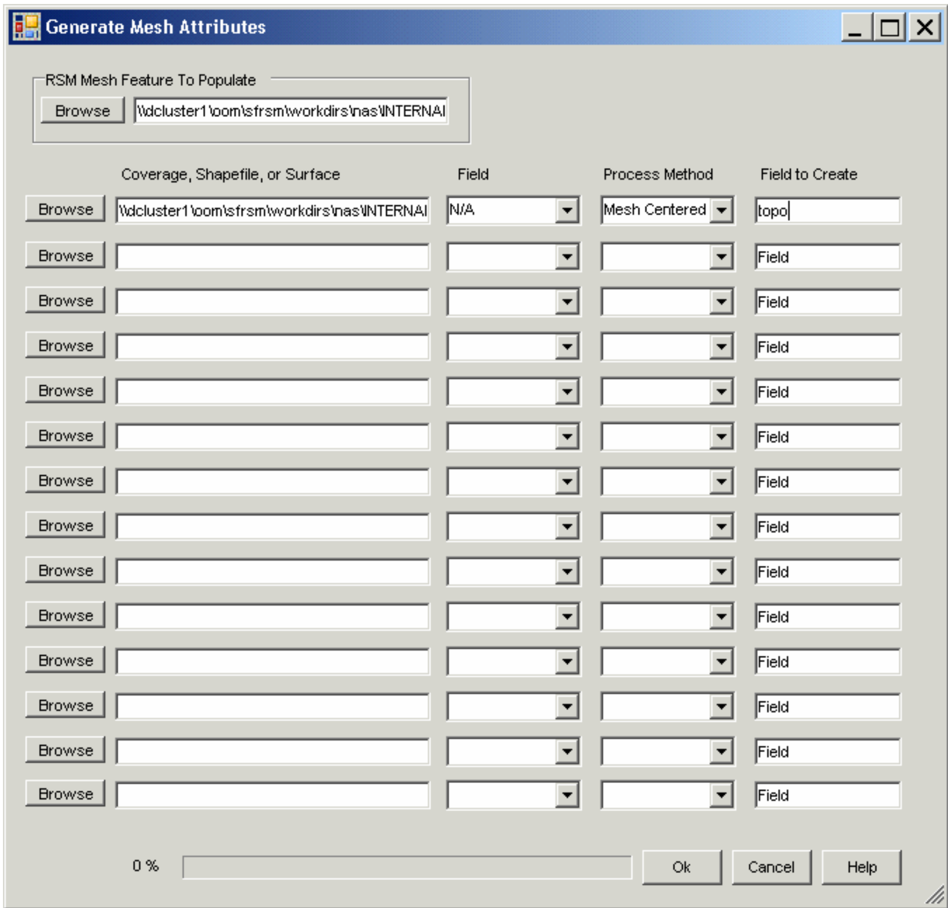


Figure 6.15 **`Generate Mesh Attributes`** menu interface

For topography which is raster data, there is no field. For different spatial data types, different processing methods are preferred. See the RSM GUI User Guide.

- Select the RSM Feature Mesh to Populate:
  `$RSM/labs/lab6_GMS/eaamc3_test.mdb`

- Select **`Coverage, Shapefile, or Surface`** with the appropriate data
  `$RSM/data/geographic/topography/rsm_topo_v2`

- Select the **Field** for the feature class you want to add to your mesh.

- Select a **`Process Method`**. Options are show in **Figure 6.16**. In this case, use **`Mesh Centered`**.

- Provide a name for this new mesh attribute in **`Field to Create`**. For this exercise, call it **`topo`**.

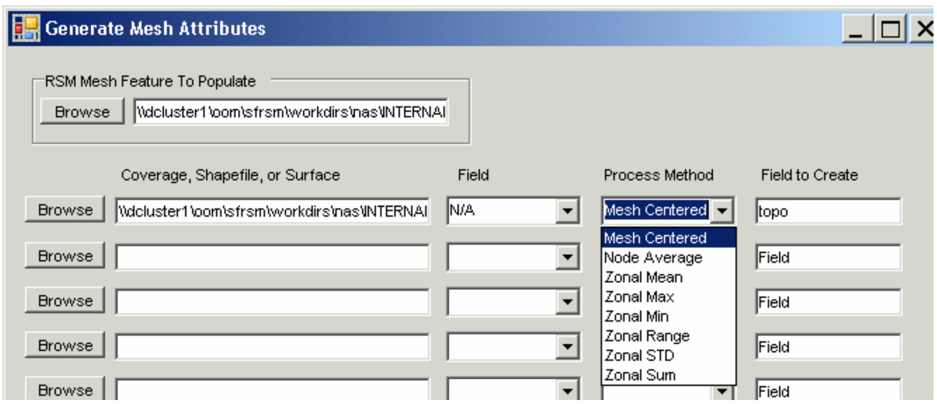


Figure 6.16 Generate Mesh Attributes **`Process Method`** options

31. Click **`OK`** to start the Intersect process.

NOTE:

WAIT! (This may take a few minutes depending on the speed of the server.) You may also get an error message which can be ignored. Click on the upper-right X to quit the tool.

32. When the process is finished use the ESRI **`Identify`** tool ⓘ to select a mesh cell and view the mesh attributes. Your cell should now contain a "**`topo`**" value (see **Figure 6.17**).

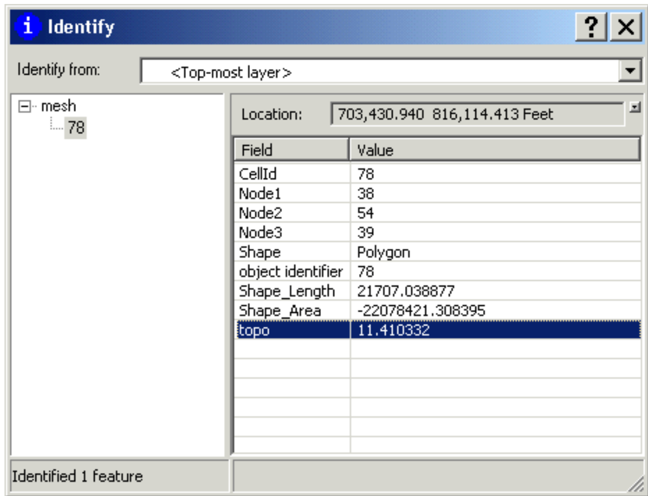
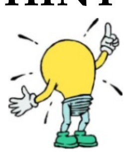


Figure 6.17 Mesh attributes found using the **`Identify`** tool

33. Save your map file in your **`lab6_GMS`** folder and call it **`eaamc.mxd`**.
34. Repeat this process for the following data:
    - Watershed: **`labs/lab/lab6_GMS/eaamc3/mdb/watersheds`**
    - Transmissivity: **`data/geographic/geology/hyd_con_v2`**
    - Bottom of aquifer: **`data/geographic/geology/base_wt_v1`**
    - Land use/land cover: **`data/geographic/landuse/lu95c.shp`**

The attributes will be part of the mesh (See **Figure 6.18**).

HINT



It may be necessary to close eaamc.mxd to add the attributes.

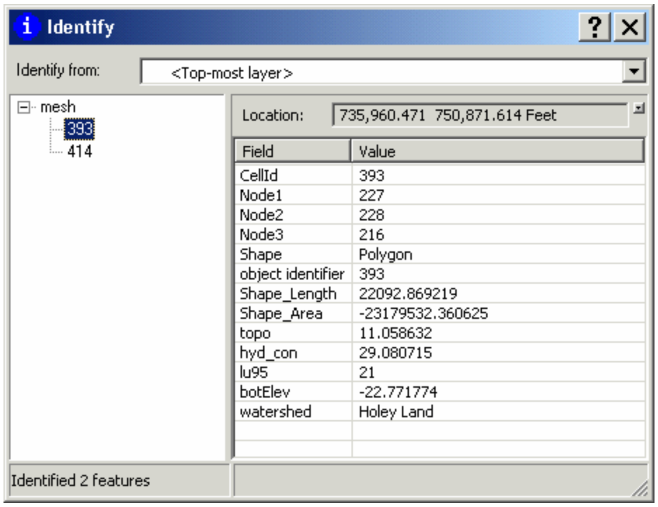


Figure 6.18 Additional mesh attributes are now available

35. Populate the mesh with the appropriate attribute values. These values are implemented in the RSM using ASCII GMS type files. These files are obtained by creating index files using the attribute values:
    - In the RSM GISToolBar, from the HSE Network drop-down menu, click Index Tool, as shown in **Figure 6.19**.

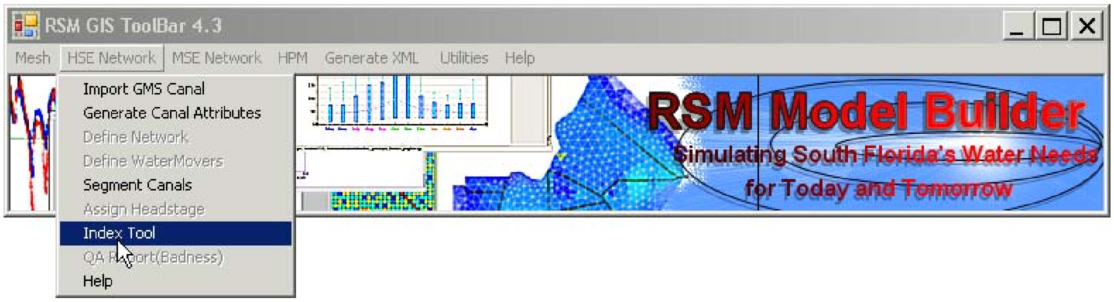


Figure 6.19 RSM GIS ToolBar

In the **`Index Tool`** dialog box, (**Figure 6.20**), select the following:

- Layer: **`mesh`**
- Attributes: **`topo`**
- Header Type: **`Main Header`**

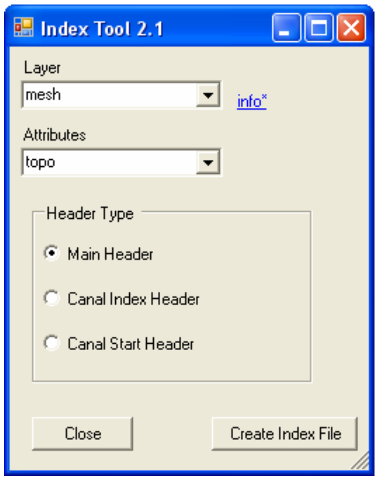


Figure 6.20 **`Index Tool`** dialog box

36. Choose the output file: **`$RSM/labs/lab6_GMS/eaamc3_topo.dat`**
37. Repeat this process for botElev, landuse, watersheds and hydraulic conductivity
38. List the content of the **`botElev.dat`** file and observe the content of the file.

The values are provided for each cell in the mesh in a cellID-ordered list.

Exercise 6.2.2 Add mesh attributes to the RSM

Once the spatial data is extracted from the RSM, the attributes are added to the EAAMC Regional Simulation Model.

In this exercise we will develop the complete RSM for the Everglades Agricultural Area-Miami Canal (EAA-MC) Basin using the mesh (**`eaamcl.2dm`**) created in **Exercise 6.1.1**. We will compare the model to a simple version without the spatial data. Several components can be added to the RSM to create more realistic conditions. The basic components for a mesh-network implementation are found in **`basicRSM.xml`**.

39. Modify the **`basicRSM.xml`**, shown in **Figure 6.21**, to have the following simple boundary conditions and save the model as **`eaam_sim.xml`**:
    - Change the "runDescriptor" to **`simple`**.
    - Add "head" monitors for cells **`93`**, **`178`**, **`415`** and **`430`**.
    - Mesh boundary conditions:

```
<mesh_bc>
<wallhead section="ol_gw" label="general head 1">
    <nodelist> 24 25 26 27 </nodelist>
    <uniform> <const value="8"></const> </uniform>
  </wallhead>
</mesh_bc>
```

    - Network boundary conditions:

```
<network_bc>
  <segmenthead id="201" bcID="401" label="S8">
               <const value="6.0"></const>
  </segmenthead>
  <segmentsource id="213" label="S3">
                 <const value="5.0"></const>
  </segmentsource>
</network_bc>
```

40. Add the spatial topography to **`eaamc.xml`** using the following code:

```
<surface><gms file="eaamc3_topo.dat" mult="1.0"></gms> </surface>
```

41. Add the **bottom elevation** of the aquifer to **`eaamc.xml`** using the following code:

```
<bottom> <gms file="botElev.dat" mult="1.0"></gms>  </bottom>
```

```xml
<?xml version="1.0" ?>
<!DOCTYPE hse SYSTEM "../../trunk/benchmarks/hse.dtd" [
]>
<hse>
  <control
    tslen="24" tstype="hour" startdate="01jan1990" starttime="0000"
    enddate="31dec1991"  endtime="2400"
    alpha="0.900" solver="PETSC" method="gmres" precond="ilu"
    units="english" runDescriptor="test">
  </control>

  <mesh>
    <geometry file="eaamc_test.2dm">   </geometry>
    <mesh_bc>  <!-- fill in -->         </mesh_bc>
    <rain>     <const value="0.0">  </const> </rain>
    <refet>   <const value="0.015">  </const></refet>

    <hpModules>
        <layer1nsm kw="1.0" rd="0.5" xd="2.0" pd="3.0" kveg="0.75" />
    </hpModules>
    <shead>    <const value="12.0"> </const></shead>
    <bottom>   <const value="-24.0">   </const> </bottom>
    <surface>  <const value="12.0"> </const> </surface>
    <conveyance><mannings a="0.0500" detent="0.001"></mannings> </conveyance>
    <transmissivity> <unconfined k = "0.002"> </unconfined></transmissivity>
    <svconverter> <constsv sc="0.2"> </constsv> </svconverter>
  </mesh>
  <network>
    <geometry file="eaamc.map"> </geometry>
    <initial file="eaamc_canal.init"> </initial>
    <network_bc>  <!-- fill in -->    </network_bc>
    <arcs>
      <indexed file="arcs.index">
        <xsentry id="1">
          <arcflow n="0.054"></arcflow>
          <arcseepage leakage_coeff="8.3E-07"></arcseepage>
          <arcoverbank bank_height="3.0" bank_coeff="0.0"> </arcoverbank>
        </xsentry>
      </indexed>
    </arcs>
  </network>

  <output>
    <!-- water budget -->
    <wbbudgetpackage file="wbbudget.nc"/>
    <hpmbudgetpackage file="hpmbudget_mo.nc" dbintl="43200"/>
    <!-- system monitors -->
    <cellmonitor id="1"  attr="head" label="c1" ><dss file="heads" />
    </cellmonitor>
    <segmentmonitor id="300007" attr="head" label="c30"><dss file="heads" />
    </segmentmonitor>

  </output>
</hse>
```

Figure 6.21 The basic components for a mesh-network implementation found in **`basicRSM.xml`**

42. Add the transmissivity of the aquifer to **`eaamc.xml`** using the following code:

```
<transmissivity>
  <unconfined_gms_layer layer="1" file="kheaa.dat"
                        mult=" 2.158E-05" />
</transmissivity>
```

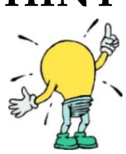
43. Add the initial head of the aquifer to **`eaamc.xml`** using the following code:

```
<shead>  <gms file="eaamc3_topo.dat" mult="0.95"/>   </shead>
```

44. Add the conveyance spatial data which occurs as a function of land cover type:

```
<conveyance>
   <indexed file="lusimple.index">
      <entry id="10" label="MAR">
         <mannings a="1.10" b="-0.77" detent="0.10">
         </mannings>
      </entry>
      <entry id="23" label="SUG">
         <mannings a="0.230" b="0.00" detent="0.10">
         </mannings>
      </entry>
   </indexed>
</conveyance>
```

HINT



The initial head may be set as a fraction of the topography where the domain is near sea level.

Activity 6.3: Using SVconverter

Overview

Activity 6.3 This activity includes one exercise:

- **Exercise 6.3.1** Replace constant storativity with a Stage-Volume Converter

The <**svconverter**> is implemented by using a Lookup table that provides the increased storage for the head above a reference depth below which the storativity is constant (**sc="0.2"**).

Exercise 6.3.1 Replace constant storativity with a Stage-Volume Converter

This function is used to convert from stage(head) to volume for conducting the water budgets in the model. The default implementation is to assign a constant value of the average specific yield of the Surficial Aquifer for storage capacity and hence, the stage-volume converter. However, an improved description would include the storage capacity of the soil and the microtopography and the landscape as illustrated in **Figure 6.22.**

In the default case the storativity is constant to the ground surface. In the microtopography case the storativity gradually increases (see **Figure 6.23**).

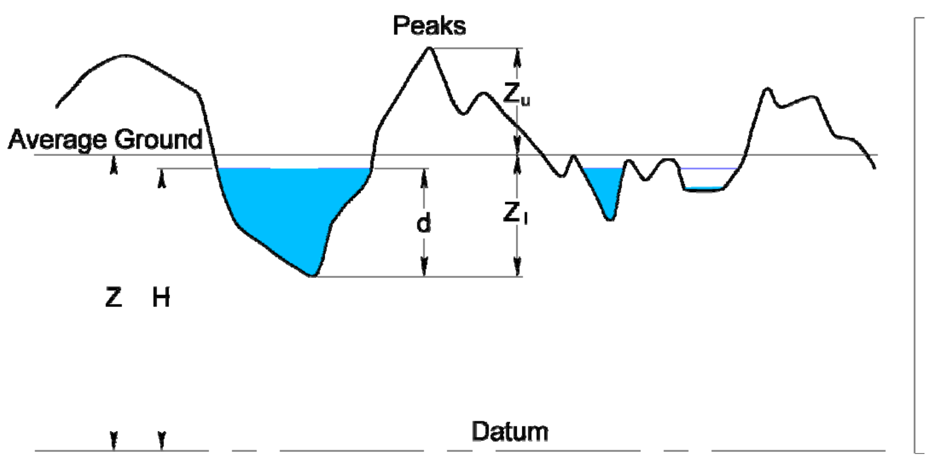


Figure 6.22 Stage-Volume Converter captures storage capacity of the soil, microtopography, and landscape information.

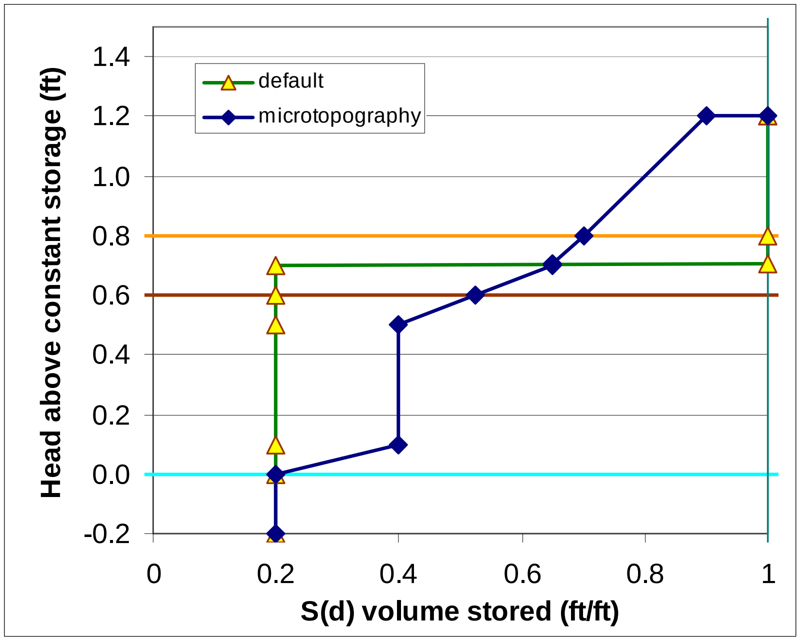


Figure 6.23 Constant storativity versus a stage-volume converter.

45. Substitute the following code for the constant for svconverter in **`eaamc.xml`**:

```
<svconverter>
    <lookupsv sc="0.2" below="0.6" above="0.2">
      <sv>
        0.0  0.0
        0.1  0.04
        0.5  0.20
        0.7  0.33
        1.2  0.78
      </sv>
    </lookupsv>
</svconverter>
```

46. Save the **`eaamc.xml`** file.
47. Change the "**`runDescriptor`**" to "**`spatial`**" and save the model as **`eaamc_sp.xml`**.
48. Run **`eaamc_sim.xml`** and run **`eaamc_sp.xml`** and compare the resulting head time series using HEC-DSSView from the RSM Graphical User Interface (RSMGUI).

Answers for Lab 6

Exercise 6.1.1

Compare results with those in the lab6 directory

Exercise 6.1.2

Compare results with those in the lab6 directory

Exercise 6.2.1

Compare results with those in the lab6 directory

Exercise 6.2.2

Compare results with those in the lab6 directory

Exercise 6.3.1

Compare results with those in the lab6 directory

Index