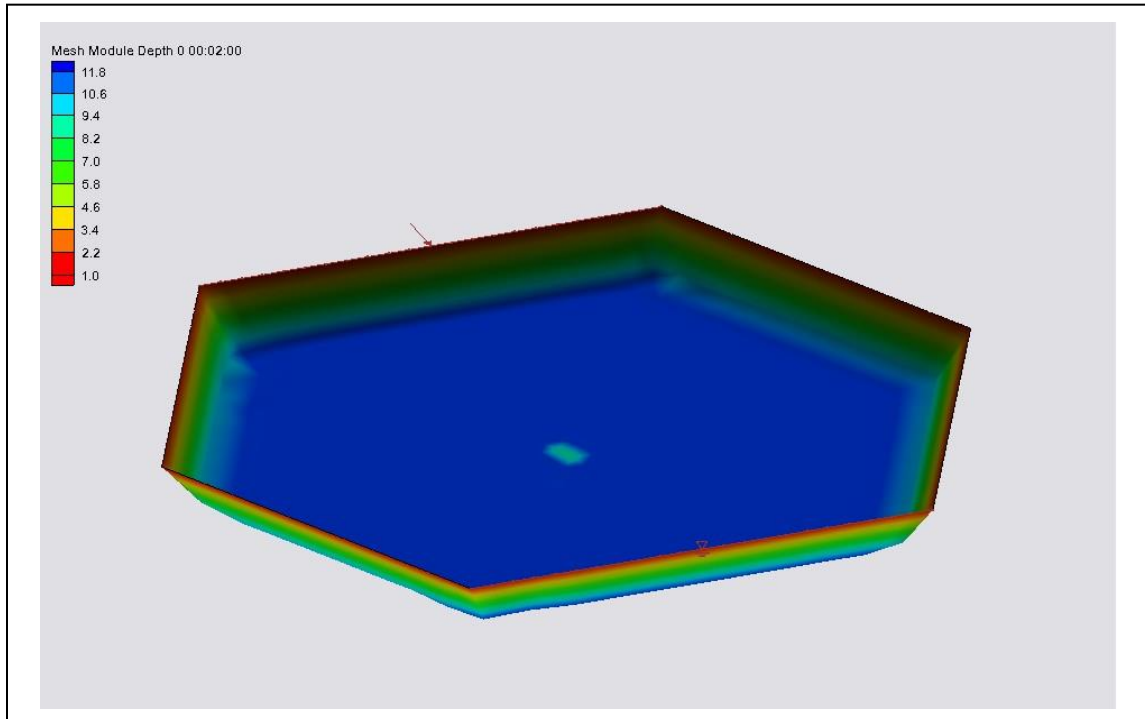*SMS 13.1 Tutorial*
ADH Vessels

Objectives

This tutorial demonstrates the ability of ADH to model the effect of vessels on the hydrodynamics of a simulation using a simple example of a small, shallow hexagon-shaped lake.

Prerequisites
- Overview Tutorial
- ADH Intro Tutorial

Requirements
- ADH
- Mesh Module
- Map Module

Time
- 20–30 minutes

1   Introduction

One of the features of ADH is the ability to model the effect of vessels on the hydrodynamics of a simulation. The vessel's presence is incorporated into the model by applying a pressure field to the water column. The bed shear stresses due to the vessel can also be outputted.

2   Setting up a Mesh and a Vessel

For this tutorial, start with importing a pre-created mesh file. This mesh represents the boundaries of a small shallow hexagonal shaped lake which will be used to simulate vessel movement.

2.1   Opening the Mesh Representing a Pond

To open the mesh:

1. Select *File |* **Open** to bring up the *Open* dialog.

2. Browse to the *data files* folder this tutorial and double-click on "hexlake.2dm" to import it into SMS.

3. Select the **Mesh Module** 🔷.

4. Switch to the ADH model by selecting *Data |* **Switch current Model…** to bring up the *Select Current Model* dialog.

5. Select *ADH* and click **OK** to close the *Select Current Model* dialog.

6. A warning dialog will come up stating that switching models can result in a loss of data. Click **OK**.

7. Select *Display |* **Display Options** to bring up the *Display Options* dialog.

8. Select "2D Mesh" from the list on the left then turn on *Contours*, *Nodestrings*, and *Mesh Boundary*. Turn off all other options on the *2D Mesh* tab.

9. On the *Contours* tab, select "Color Fill" in the *Contour method* section.

10. Click **OK** to close the *Display Options* dialog.

11. Go to *Display* | **Display Projection...** to bring up the *Display Projection* dialog.

12. In the *Horizontal* section, select the *No projection* radio button.

13. Set the *Units* drop-down to "Meters".

14. In the *Vertical* section, set the *Units* drop-down to "Meters."

15. Click **OK** to close the *Display Projection* dialog.

2.2   Creating a Vessel

This lake is a simple ten meter deep pond in the shape of a hexagon. It is about 300–350 meters across. This does not exactly represent an actual lake but it is big enough to contain a moving boat for simulation purposes.

The vessel and path information will be stored as part of an SMS coverage. The vessel attributes will be stored as coverage attributes and the path as arcs within SMS.

The effect of the propeller for vessels can also be simulated. If vessels are turned on, ADH assumes two propellers equally spaced from the middle of the boat.

First, the vessel needs to be created and defined by doing the following:

1. Right-click on "🗺 Map Data" in the Project Explorer and select **New Coverage** to bring up the *New Coverage* dialog.

2. Set the *Coverage Type* to "Vessel" then enter "boat" as the *Coverage Name*.

3. Click **OK**, which brings up the *Vessel Attributes* dialog.

4. In the *Vessel parameters* section, set:

    • *Length* to "15.0".

    • *Width* to "8.0".

    • *Draft* to "3.0".

    • Leave everything else at the default settings.

5. In the *Propeller* section, turn on *Include propeller data* and set:

    • *Type* as "Open wheel"

    • *Diameter* to "0.15".

    • Distance between propellers to "6.0".

    • Leave everything else at the default settings.

This is a big boat for such a small lake, particularly with a three meter draft. However, its large size and excessive draft will help show the boat's movements.

6. Click **OK** to close the *Vessel Attributes* dialog.

Now that the vessel attributes have been defined, the boat path within the domain should be

defined. The paths are defined by feature arcs. Each feature node (endpoint) has an attribute that defines the speed of the boat at that point. The speed along the arc will be interpolated from the values at the endpoints. Intermediate vertices along the arcs will change the boat's direction but do not affect the vessel speed.

To have a speed change at a vertex, convert the vertex to a node by doing the following:

1. Select the " boat" coverage to make it active.

2. Using the **Create Feature Arc** tool, create an arc from about (-50, -80) to (75, 115).

If wanting more precision in placing the arc, the **Select Feature Point** tool can be used to select and move a specific node. The Edit Window can then be used to directly enter the *x* and *y* coordinates. This is just a simple path that will be changed later.

3. Using the **Select Feature Point** tool, double-click on the first node to bring up the *Vessel Node properties* dialog.

4. Set *Boat speed at point* to "0.5" then click **OK** to close the *Vessel Node properties* dialog.

5. Double-click on the second node to bring up the *Vessel Node properties* dialog.

6. Set *Boat speed at point* to "1.5" then click **OK** to close the *Vessel Node properties* dialog.

7. Drag the " boat "coverage onto the " hexlake" mesh. A link will be added under a new " ADH Vessel folder" under the mesh (see Figure 1).
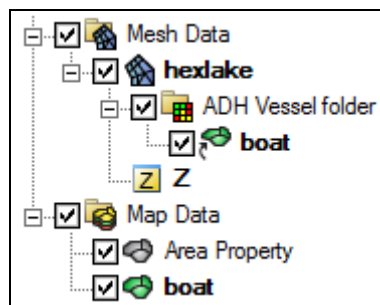

Figure 1   Project Explorer with "boat" link under mesh module

3   Setting the Model Parameters

The next step is to set up the model parameters for the ADH simulation:

1. Click on the " hexlake" mesh in the Project Explorer to make it active.

2. Select *ADH | **Model Control…*** to bring up the *ADH Model Control* dialog.

3. On the Solver tab in the Operation section, enter "40" for Increment memory allocation block size.

4. Turn on *Include vessel stress effects (requires propellers on boats)*. This will make use of the propeller information set previously entered.

5. On the *Time* tab in the *Simulation* section, select the *Dynamic* radio button.

6. Set *Start* to "7/1/2009 1:00:00 PM."

7. Set *Duration* to "15" and select "minutes" from the drop-down.

8. In the *Time Step Control* section, click on the large **Time Step Sizes** button to bring up the *Time Series* dialog.

9. Select "day" from the *Time* drop-down and "second" from the *Time step size* drop-down.

10. Set row *1* to be "0.0" and "30.0".

11. Set row *2* to be "10.0" and "30.0".

12. Click **OK** to close the *Time Series* dialog.

13. On the *Output* tab in the *Output Times* section, select the *Add by specifying a range* radio button.

14. Enter "0.5" in *Start at,* and select "minutes" from the drop-down.

15. Enter "30.0" in *End at*.

16. Set *Increment* to "30.0" and select "seconds" from the drop-down.

17. Click the **Add** button.

18. Since materials or sediment data are not changing, leave the other tabs as they are. Click **OK** to close the *ADH Model Control* dialog.

ADH requires initial conditions for the model to be entered by doing the following:

1. Click on the " hexlake" mesh in the Project Explorer to make it active.

2. Select *ADH* | **Hot Start Initial Conditions…** to bring up the *ADH Hot Start Initial Conditions* dialog.

3. In the *Depth (required)* section, select *Constant water surface* and enter "1.0" as the water *Elevation*.

4. Click **OK** to close the ADH Hot Start Initial Conditions dialog.

4   Saving the Project

Before running the simulation, it is recommended the project be saved by doing the following:

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

2. Enter "hexlake_001" as the *File name* and select "Project Files (*.sms)" for *Save as type*.

3. Click the **Save** button to save the file and close the *Save* dialog.

Additional projects for modeling tests may be saved by incrementing the number in the name of the file, or by entering a completely different name for the project file.

5   Setting Boundary Conditions

An inlet and an outlet to the lake need to be created. The water flowing past is another way to see the effects of the boat. A constant flowrate and water surface elevation will be used throughout this simulation. Two nodestrings will be created: one for flowrate and one for tailwater (water surface elevation), as shown in Figure 2.
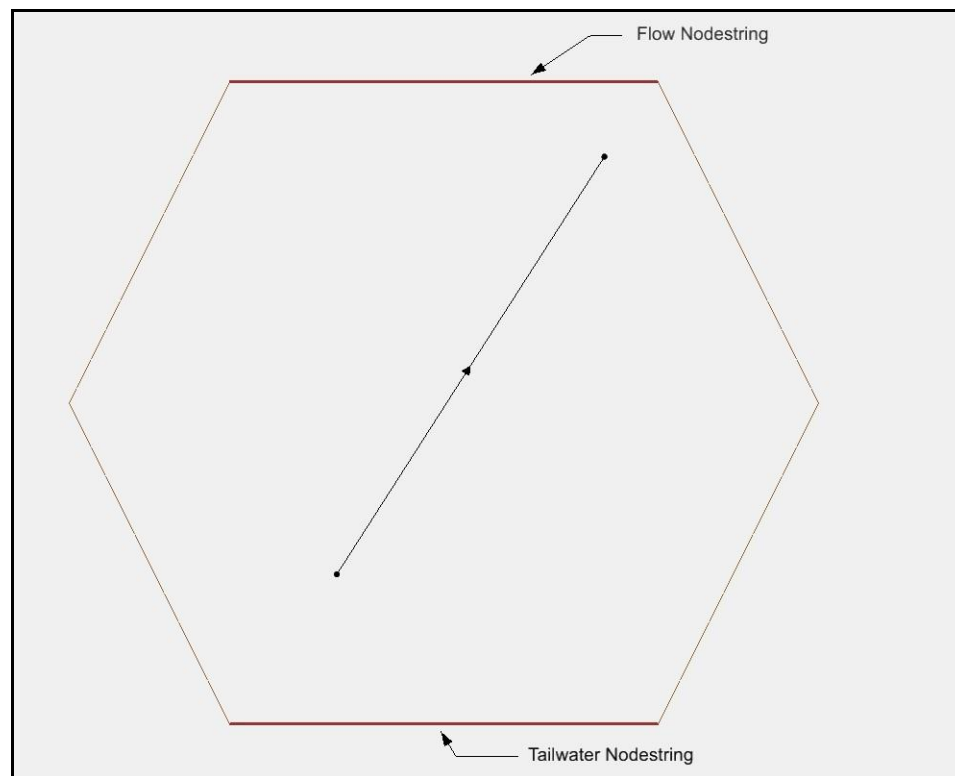

Figure 2   Boundary nodestring locations

To create the inflow nodestring:

1. Select the "⬡ hexlake" mesh to make it active.

2. Using the **Create Nodestring** ⋀ tool, click on the upper left corner of the mesh.

3. Hold down the *Shift* key and double-click on the upper-right corner of the mesh, creating a nodestring across the top edge of the hexagon (labeled "Flow Nodestring" in Figure 2).

Nodestrings are always created right to left as if facing downstream. Holding down the *Shift* key makes sure all vertices and nodes along the nodestring are selected. Not doing so will cause results to be inconsistent and even wrong.

4. Using the **Select Nodestring** ⬆ tool, select the inflow nodestring.

5. Right-click the nodestring selection box and select *Boundary Condition* | **Assign...** to bring up the *ADH Boundary Condition Assignment* dialog.

6. On the *Flow* tab, select "Flow (per unit length)" from the drop-down.

7. Select "Manning's n" for *Friction*.

8. Enter "0.03" for Manning's n roughness.

9. Click on the large **Curve undefined** button under *Flow data* to bring up the *Time Series* dialog (Figure 3).

10. In the *Curve Information* section, click the **New…** button to bring up the *Specify Curve Name* dialog.

11. Enter "Flow data" and click **OK** to finish the naming and close the *Specify Curve Name* dialog.
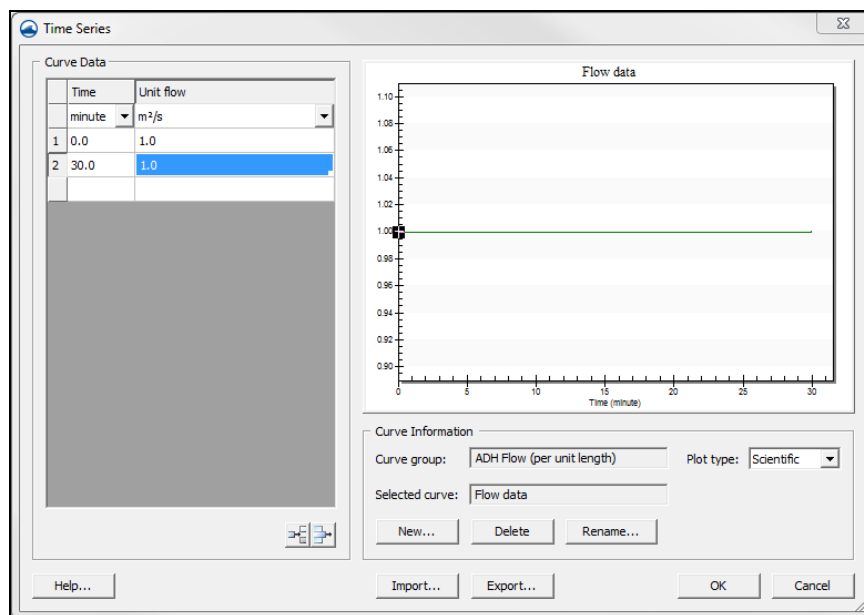

Figure 3   Time series dialog for flow data

12. Select "minute" from the drop down in the *Time* column and "m^2/s" from the drop-down in the *Unit flow* column.

13. In row *1*, enter "0.0" and "1.0".

14. In row *2*, enter "30.0" and "1.0".

15. Click **OK** to close the *Time Series* dialog.

16. Click **OK** to close the ADH Boundary Condition Assignment dialog.

To create the tailwater nodestring, do the following:

1. Using the **Create Nodestring** ⌃ tool, make another nodestring across the bottom edge of the mesh, starting at the lower-left corner and ending on the lower-right (right to left as if facing downstream, and be sure to hold the *Shift* key down

while double-clicking at the end of the nodestring).

2. Using the **Select Nodestring** ⬆ tool, select the tailwater nodestring.

3. Right-click the nodestring selection box and select *Boundary Condition | **Assign...*** to bring up the *ADH Boundary Condition Assignment* dialog.

4. On the *Flow* tab, select "Water surface elevation" from the drop-down.

5. Select "Manning's n" for *Friction*.

6. Enter "0.03" for Manning's n roughness.

7. Click on the large **Curve undefined** button under *Flow data* to bring up the *Time Series* dialog.

8. In the *Curve Information* section, click the **New…** button to bring up the *Specify Curve Name* dialog.

9. Enter "Water surface elevation data" and click **OK** to finish the naming and close the *Specify Curve Name* dialog.

10. Select "minute" from the drop down in the *Time* column and "m" from the drop-down in the *WSE* column.

11. In row *1*, enter "0.0" and "1.0".

12. In row *2*, enter "30.0" and "1.0".

13. Click **OK** to close the *Time Series* dialog.

14. Click **OK** to close the ADH Boundary Condition Assignment dialog.

6   Defining the Boat Path

So far, the boat path is rather simple. It goes from one corner of the lake to the other with the boat traveling at about 1 meter per second on average. The boat finishes its path in just under 4 minutes, and then stops there and floats in place for the remainder of the 15 minute simulation.

A more complicated path can be created by doing the following

1. Select the "🐢 boat" coverage to make it active.

2. Using the **Create Feature Arc** 𝄐 tool, add additional segments to the arc as shown in Figure 4.
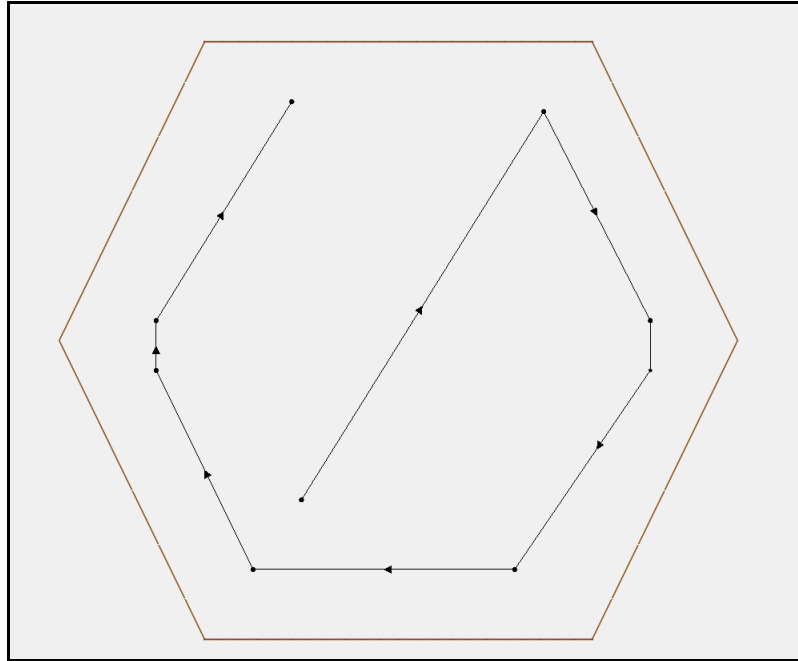
Figure 4   The full boat path

As mentioned previously, vertices can be changed to nodes to allow the boat speed to change.

3. Using either the **Select Feature Vertex** tool or the **Select Feature Point** tool, select the vertex or node closest to the coordinates in the table and change the coordinates in the Edit Window for that vertex or node according to the table. See Figure 5 for the number of each node or vertex.

Figure 5   Nodes and vertices numbered

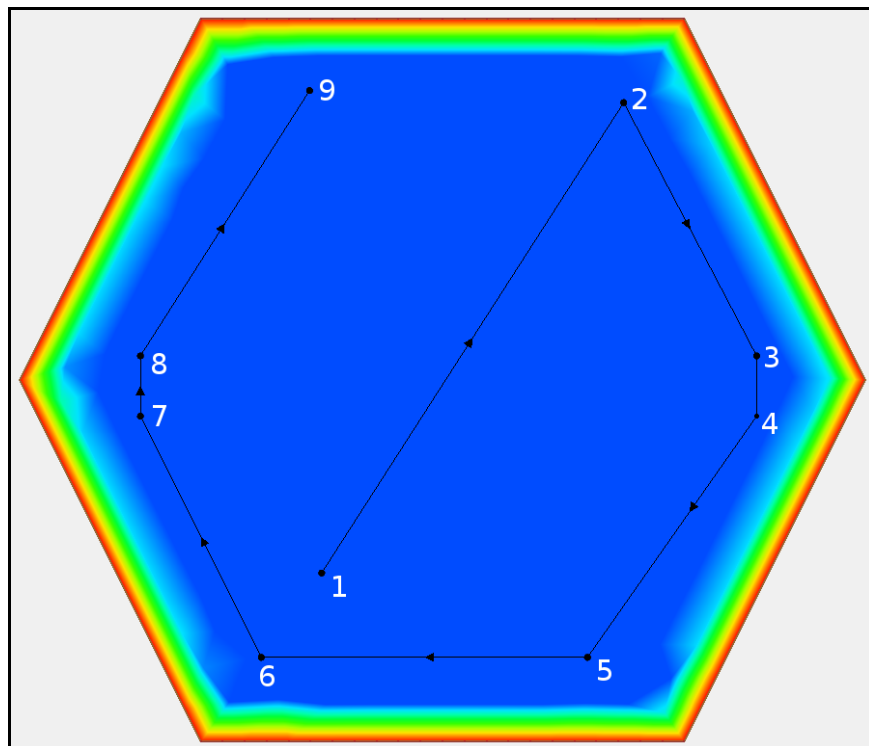4. Convert any vertices to nodes (except for point #4 in Figure 5) by right-clicking on the point and selecting **Nodes ↔ Vertices**.

#	X	Y	Type	Speed
1	-50	-80	Node	0.5
2	75	115	Node	1.5
3	130	10	Node	1
4	130	-15	Vertex	--
5	60	-115	Node	2
6	-75	-115	Node	10
7	-125	-15	Node	10
8	-125	10	Node	0.3
9	-55	120	Node	0.3

5. Double-click on each node to bring up the *Vessel Node properties* dialog.

6. Enter the correct speed for each node according to the above table and click **OK** to exit the *Vessel Node properties* dialog.

Nodes are able to change the speed of the boat at that location. Vertices simply interpolate their speed based on the nodes before and after them. When finished, the boat path should look like Figure 4.

This path simply takes the boat around the edges of the lake. The boat starts out at a speed of 1–2 m/s. Near the bottom it throttles up to 10 m/s (36 kph), then slows down quickly to almost a stop. The boat then drifts the rest of the way at a very low speed.

7   Model Check and Simulation Run

It is always a good idea to save the project and then run the Model Checker before running the simulation.

To save the project, do the following:

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

2. Enter "hexlake_002" in the *File name* field and select "Project Files (*.sms)" from the *Save as type* drop-down.

3. Click on the **Save** button to save the project and close the *Save As* dialog.

To run the model check, do the following:

1. Select the "🏠 hexlake" mesh to make it active.

2. Select *ADH* | **Model Check...**

3. Click **OK** if there are no model checks violated and continue to step 5.

4. If there are any potential issues, follow the directions in the *Fix* section of the *Model Checker* dialog. Go back to step 2 again until there are no model check violations.

5. Select *ADH | **Run ADH*** to bring up the *Pre-ADH* dialog. Pre-ADH will launch in the model wrapper and should finish in a few seconds.

Upon successful completion of Pre-ADH, the **Abort** button in the model wrapper will change to **Run ADH**.

6. Click **Run ADH** to start ADH and bring up the *ADH* dialog.

7. When ADH finishes, turn on *Load solution* and click **Exit** to close the *ADH* dialog.

8. If asked to select time units, select "seconds."

9. Select the "⊞ Depth" dataset to make it active.

10. Step through the time steps. The boat location is visible as it moves along the left edge, zips around the bottom right corner, and then slows down for the final segment.

The vessel stress file, "hexlake_str.dat," can be opened from the same location where the ADH files were saved.

The "↗ Overland Velocity" dataset is also present. To view the information from this dataset:

1. Select *Display | **Display Options…*** to bring up the *Display Options* dialog.

2. Select "2D Mesh" from the list on the left then turn on *Vectors*.

3. Click **OK** to close the *Display Options* dialog.

4. Now step through the time steps again, and see the direction of flow throughout the simulation.

8 Conclusion

This concludes the *ADH Vessel* tutorial. If the user continues to model, be sure to save a new project for each modeling instance so files are not overwritten.