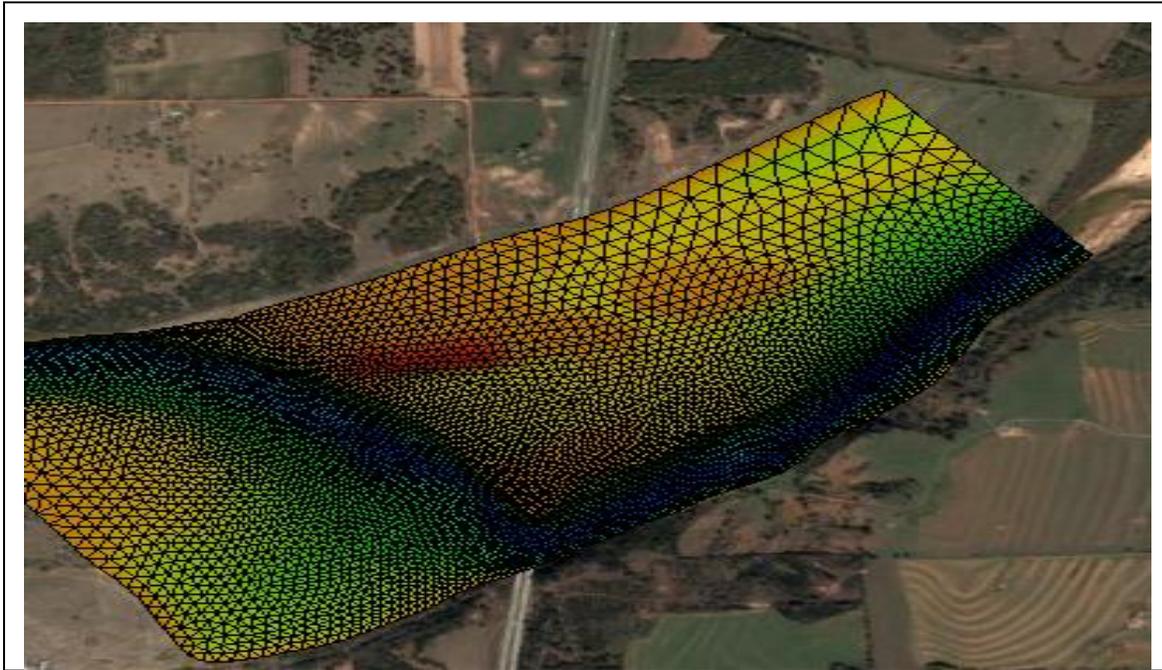


## SMS 12.2 Tutorial

### ADH Hydrodynamics



#### Objectives

This tutorial will introduce how to prepare and run a basic ADH model using the SMS interface.

#### Prerequisites

- Overview Tutorial

#### Requirements

- ADH
- Mesh Module
- Scatter Module
- Map Module
- GIS Module

#### Time

- 30-60 minutes

**AQUAVEO™**



<b>1</b>	<b>Introduction .....</b>	<b>2</b>
<b>2</b>	<b>Background Data.....</b>	<b>2</b>
2.1	Units .....	3
2.2	Topographic Data.....	3
2.3	Background Image .....	4
2.4	Display Options.....	4
<b>3</b>	<b>Building a Conceptual Model .....</b>	<b>5</b>
3.1	Model Extents .....	5
3.2	Area Properties.....	6
3.3	Meshing Properties.....	8
<b>4</b>	<b>Model Parameters .....</b>	<b>9</b>
4.1	Creating the Mesh .....	9
4.2	Material Properties .....	9
4.3	Boundary Conditions .....	10
4.4	Time Control .....	11
4.5	Output Control .....	12
4.6	Initial Conditions.....	12
<b>5</b>	<b>ADH Model Execution .....</b>	<b>13</b>
<b>6</b>	<b>Viewing the Solution.....</b>	<b>14</b>
<b>7</b>	<b>Conclusion.....</b>	<b>14</b>

## 1 Introduction

---

ADH is a state-of-the-art Adaptive Hydraulics Modeling system developed by the Coastal and Hydraulics Laboratory, ERDC, USACE ([www.chl.erd.c.usace.army.mil](http://www.chl.erd.c.usace.army.mil)). It is capable of handling two-dimensional shallow water problems.

One of the major benefits of ADH is its use of adaptive numerical meshes that can be employed to improve model accuracy without sacrificing efficiency. It also allows for the rapid convergence of flows to steady state solutions. ADH contains other essential features such as wetting and drying, completely coupled sediment transport (not currently supported in the SMS interface), and wind effects.

The area used in the tutorial is where the Cimarron River crosses I-35 in Oklahoma, about 50 miles north of Oklahoma City. The necessary input files are found in the *data files* folder for this tutorial.

## 2 Background Data

---

The first step in building a model with SMS is to import background data:

- Geographic (location) and topographic (elevation) data
- Images of maps and aerial photos
- Land use data
- Boundary conditions

## 2.1 Units

---

The data used in this tutorial are in SI units, so the current projection will need to be set accordingly:

1. Right-click on the “Area Property” coverage and select **Projection...** to open the *Object Projection* dialog.
2. Select **Global Projection** to open the *Select Projection* dialog.
3. Set the following:
  - Projection to “UTM”.
  - Zone to “14 (102°W–96°W – Northern Hemisphere)”.
  - Datum to “NAD83”.
  - Planar Units to “METERS”.
4. Click **OK** to close the *Select Projection* dialog.
5. Set *Units* to “Meters” in the *Vertical* section.
6. Click **OK** to close the *Object Projection* dialog.
7. Select *Display / Projection...* to bring up the *Display Projection* dialog.
8. Follow steps 2–5, above.
9. Click **OK** to close the *Display Projection* dialog.

## 2.2 Topographic Data

---

Topographic data in SMS are managed as triangulated irregular networks (TINs) in the scatter module. The scattered data will be the source of the elevation data for the ADH mesh.

To import the TIN:

1. Select *File / Open* to bring up the *Open* dialog.
2. Select the file “Cimarron Survey.h5” in the *data files* folder and click the **Open** button. The *Open* dialog will close and a scatter set will appear in the Project Explorer.

If the scatter set can't be seen in the Graphics Window, do the following:

3. Select *Display / Display Options...* to bring up the *Display Options* dialog.
4. Select “Scatter” from the list on the left then turn on *Points*.
5. Click **OK** to close the *Display Options* dialog.
6. Click on the **Frame**  macro.

The screen will refresh, showing a set of scattered data points as seen in Figure 1.



Figure 1 Imported scatter set

## 2.3 Background Image

---

An aerial photo or map of the study site is useful when building a numeric model. An image for the study site was generated using Google Earth Pro.

To open this file:

1. Select *File* / **Open...** to bring up the *Open* dialog.
2. Select the file “ge\_highres.jpg” in the *data files* folder and click the **Open** button.  
A map image will appear behind the scatter set in the Main Graphics Window.

## 2.4 Display Options

---

Items loaded into SMS can be hidden and unhidden by toggling on or off the box to the left of each item in the Project Explorer. This can help to reference the location of features or to simplify the display. The display options for the topographic data can also be adjusted.

To do this:

1. Select the *Display* / **Display Options** to open the *Display Option* dialog.
2. Select “Scatter” from the list on the left then turn off *Points* and turn on *Boundary* and *Contours* options.
3. On the *Contours* tab in the *Contour method* section, select “Color Fill” from the drop-down and set the *Transparency* to “50%”.
4. Click **OK** to close the *Display Option* dialog.

## 3 Building a Conceptual Model

---

An ADH model requires a finite element mesh with linear, triangular elements. Feature objects will be used to create a conceptual model. The conceptual model defines the model domain, material properties, and the mesh type. Specific model control and boundary condition data will be added after the mesh is created.

### 3.1 Model Extents

---

To define the model extents, an arc will be extracted from a specific TIN contour that represents a rough estimate of the extents of the flooding:

1. Right-click on Map Data in the Project Explorer and select **New Coverage** to bring up the *New Coverage* dialog.
2. Select “ADH” in the *Coverage Type* section then enter “Boundary” for the *Coverage Name*.
3. Click **OK** to close the *New Coverage* dialog.
4. Right-click on the “Survey 2005” scatter set and select *Convert / Scatter Contours*→**Map** to bring up the *Create Contour Arcs* dialog.
5. Select “Boundary” for the *Destination coverage*. If not already set as the default, click on the button and use the *Select Tree Item* dialog to select the coverage.
6. Enter “271” in the *Elevation* field.
7. Enter “15” in the *Spacing along contour* field.
8. Click **OK** to close the *Create Contour Arcs* dialog.

Arcs will be created along the 271 m elevation contour and the vertex spacing on the arcs will be 15 meters. Additional arcs could be extracted at certain elevations representing other key features. For consistency in completing this tutorial, the newly created coverage will be deleted and a map file with feature objects defined will be imported.

1. Right-click on Map Data in the Project Explorer and select **Clear Coverages**.
2. Select **Yes** to clear all coverages. Only the default “Area Coverage” will remain.
3. Select *File / Open...* to bring up the *Open* dialog.
4. Select the file “ADH\_Model.map” in the *data files* folder and click on the **Open** button.

The imported coverage contains arcs delineating the model boundary as well as the outer river banks, as shown in Figure 2.

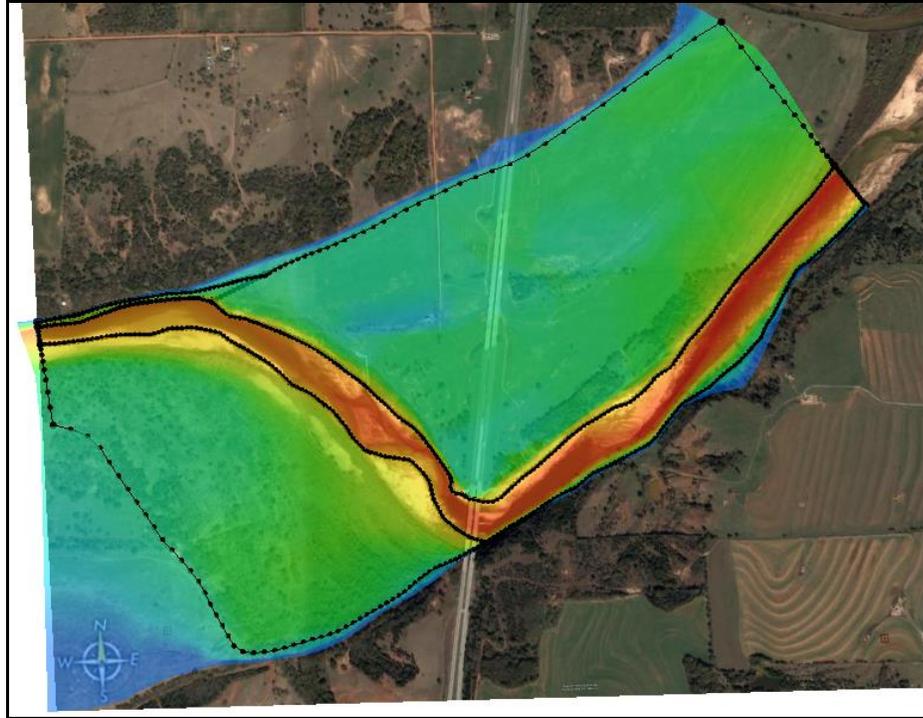


Figure 2 Conceptual model extents with a background image

### 3.2 Area Properties

Feature polygons in an area property coverage are used to define the material zones of the model. Polygons can be digitized manually based on a map or aerial photo, or they can be imported. For this case, land use data will be imported from an ESRI shapefile.

To do this:

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 “Area Property” and enter “materials” for the *Coverage Name*.
3. Click **OK** to close the *New Coverage* dialog.
4. Click on the new “materials” coverage to make it the active coverage. This ensures that when the GIS data are converted to feature objects, the feature objects are added to the “materials” coverage.
5. Select *File* | **Open...** to bring up the *Open* dialog.
6. Select the file “materials.shp” in the *data files* folder and click the **Open** button. This will load the data into the GIS module.
7. Click on “materials.shp” under the GIS Data folder in the Project Explorer to make it active.
8. Select *Mapping* | **Shapes**→**Feature Objects** to bring up a dialog asking if all shapes in all visible shapefiles should be used.
9. Click **Yes** to bring up the *GIS to Feature Objects Wizard* dialog.

10. Select “materials” under the *Use an existing coverage* section and click the **Next** button.
11. Select “Material” from the *MATNAME* drop-down.
12. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.

Notice that the “materials” coverage contains polygons, but the polygons do not cover the entire domain. An additional polygon and material type will be created to cover areas not covered by the shapefile.

Upon startup, SMS automatically creates two materials: Disable and Material 01 with their respective material IDs 0 and 1. ADH requires a material ID of 1 in order for it to run. This material ID will be used as the default.

To do this:

1. Select *Edit | Materials Data* to bring up the *Materials Data* dialog.
2. In the *Materials* section, double-click on “material 01”.
3. Rename this material to “grasslands.”
4. Click **OK** to close the *Materials Data* dialog.

To create the new polygon and assign a material type:

1. Click on the “materials” coverage in the Project Explorer to make it active.
2. Using the **Create Feature Arc**  tool, create a rectangular closed arc that completely encloses the scatter set.
3. Select *Feature Objects | Build Polygons*.
4. Using the **Select Feature Polygon**  tool, click in the newly created polygon to select it.
5. Right-click and select **Attributes...** to bring up the *Land Polygon Attributes* dialog.
6. In the *Polygon Type* section, select *Material* and choose “grasslands” from the drop-down.
7. Click **OK** to close the *Land Polygon Attributes* dialog.
8. Choose *Display | Display Options...* to bring up the *Display Options* dialog.
9. Select “Map” from the list on the left then turn on *Fill* and *Legend*.
10. Click **OK** to close the *Display Options* dialog.
11. Uncheck Scatter Data in the Project Explorer to turn it off.

The display should look similar to Figure 3 showing where the materials occur within the domain. The colors/patterns may be different depending upon the user’s settings. These can be changed by going to *Edit | Materials Data* and adjusting the settings there.

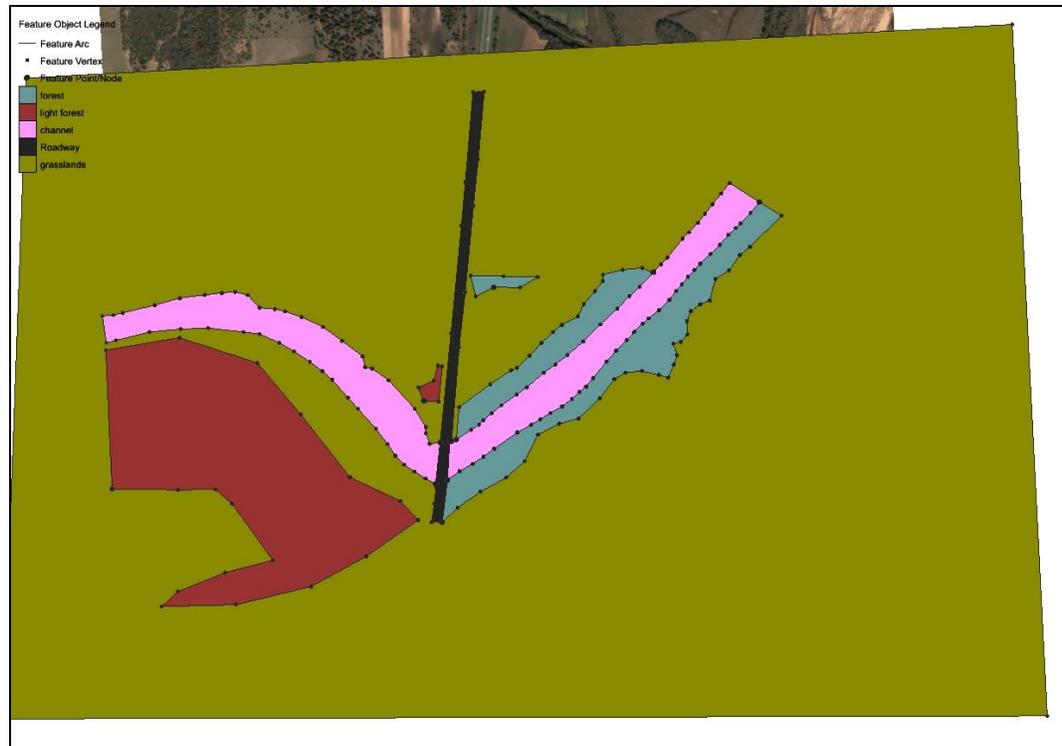


Figure 3 Feature polygons representing material zones

### 3.3 Meshing Properties

The next step in constructing a conceptual model is to define the mesh generation parameters. Meshing properties will be assigned to feature polygons in the “ADH Model” coverage.

To do this:

1. Click on the “ADH Model” coverage to make it active.
2. Using the **Select Feature Polygon**  tool, select all polygons by dragging a box around them (or selecting *Edit | Select All*).
3. Select *Feature Objects | Attributes...* to bring up the *2D Mesh Multiple Polygon Properties* dialog.
4. Turn on *Mesh type* and select “Paving” from the drop-down. Because ADH requires triangular elements, paving is a good option. Paving creates elements based on the vertex distribution on the boundary arcs of the polygons. Resulting mesh nodes are then relaxed to optimize element quality.
5. Turn on *Bathymetry type* and select “Scatter Set” from the drop-down.
6. Click the **Scatter Options...** button to bring up the *Interpolation* dialog
7. In the *Scatter Set To Interpolate From* section, select “elevation”. Leave all other options at the default values.
8. Click **OK** to close the *Interpolation* dialog.
9. Click **OK** to close the *2D Mesh Multiple Polygon Properties* dialog.

## 4 Model Parameters

---

### 4.1 Creating the Mesh

---

With the meshing parameters set, the conceptual model is ready to convert to a finite element mesh for ADH by doing the following:

1. Click on the “ADH Model” coverage to make it active.
2. Select *Feature Objects* | **Map**→**2D Mesh** to bring up the *2D Mesh Options* dialog.
3. Turn on *Use area coverage* and select “materials” from the drop-down.
4. Click **OK** to close the *2D Mesh Options* dialog.
5. Click **OK** if a dialog appears stating how many elevations were extrapolated.
6. Click **OK** to accept the default *Mesh name* of “ADH Model Mesh” in the *Mesh Name* dialog.

A finite element mesh with triangular elements is created. The node elevations are interpolated values from the scatter set survey and element material types are based on the materials coverage.

7. Uncheck Scatter Data, Map Data, and GIS Data in the Project Explorer to make it easier to work with the mesh.

### 4.2 Material Properties

---

Elements in the mesh have been assigned material types, but the parameters and properties associated with each material still need to be specified by doing the following:

8. Select “ADH Model Mesh” to make it active.
1. Select *ADH* | **Material Properties...** to bring up the *ADH Material Properties* dialog.
2. On the *Properties* tab, set *Eddy viscosity* to “Estimated” and enter “0.5” for the *Weighting factor* for each of the five materials listed on the left.
3. For each material in the list, set *Friction* to “Manning’s n” and set the *Manning’s n roughness* values according to the table below.
4. Once done, click **OK** to close the *ADH Material Properties* dialog.

Material	Mannings <i>n</i>
channel	0.03
forest	0.10
grasslands	0.06
light forest	0.08
roadway	0.02

### 4.3 Boundary Conditions

---

Boundary conditions force the model with certain hydrodynamic conditions. For this model, flow vs. time forcing will be specified at the upstream boundary and water surface elevation vs. time downstream.

To set up the boundary conditions for the upstream nodestring:

1. Click on “ADH Model Mesh” in the Project Explorer to make it active.
2. Using the **Create Nodestring**  tool, create nodestrings at the upstream and downstream boundaries as shown in Figure 4. Include all nodes along the boundary in the nodestrings by holding down the *Shift* key while creating it. Create the nodestring from right to left as if facing downstream.
3. Using the **Select Nodestring**  tool, select the upstream nodestring by clicking in the selection box for that nodestring.
4. Right-click and select **Renumber Nodes**.
5. Right-click and select *Boundary Condition* | **Assign...** to bring up the *ADH Boundary Condition Assignment* dialog.
6. On the *Flow* tab, select “Total discharge” from the drop-down.
7. Click on the large **Curve undefined** button below *Discharge 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 “Inflow BC” in the field.
10. Click **OK** to close the *Specify Curve Name* dialog.
11. In the *Curve Data* section, select “hour” from the *Time* drop-down and “m<sup>3</sup>/s” from the *Discharge* drop-down.
12. Open the file “ADH\_bc.xls” in a spreadsheet program.
13. Copy the values from the *Time (hr)* column in the “ADH\_bc.xls” file to the *Time* column in the *Curve Data* section.
14. Copy the values from the *Flow (cms)* column in the “ADH\_bc.xls” file to the *Discharge* column in the *Curve Data* section.
15. Click **OK** to close the *Time Series* dialog.
16. Click **OK** to close the *ADH Boundary Condition Assignment* dialog.

Now set up the boundary conditions for the downstream nodestring:

1. Using the **Select Nodestring**  tool, select the downstream nodestring.
2. Right-click and select *Boundary Condition* | **Assign...** to bring up the *ADH Boundary Condition Assignment* dialog.
3. On the *Flow* tab, select “Water surface elevation” from the drop-down.
4. Click on the large **Curve undefined** button below *Water surface elevation data* to bring up the *Time Series* dialog.

5. In the *Curve Information* section, click the **New...** button to bring up the *Specify Curve Name* dialog.
6. Enter “Outflow BC” in the field.
7. Click **OK** to close the *Specify Curve Name* dialog.
8. In the *Curve Data* section, select “hour” from the *Time* drop-down and “m” from the *WSE* drop-down.
9. Open the file “ADH\_bc.xls” in a spreadsheet program.
10. Copy the values from the *Time (hr)* column in the “ADH\_bc.xls” file to the *Time* column in the *Curve Data* section.
11. Copy the values from the *WSE (m)* column in the “ADH\_bc.xls” file to the *WSE* column in the *Curve Data* section.
12. Click **OK** to close the *Time Series* dialog.
13. Click **OK** to close the *ADH Boundary Condition Assignment* dialog.

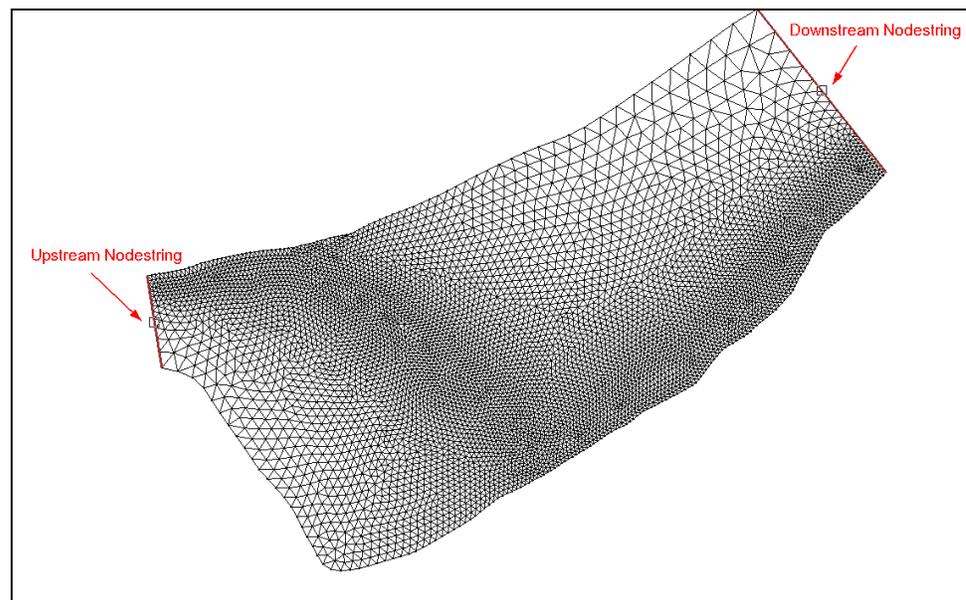


Figure 4 Nodestring locations

#### 4.4 Time Control

For this model, a constant time step is specified and ADH will adjust the time step throughout the run as needed.

Set the time control parameters for the model by doing the following:

1. Select **ADH | Model Control** to bring up the *ADH Model Control* dialog.
2. Click on the *Time* tab.
3. In the *Simulation* section, select the *Dynamic* radio button.
4. In the *Duration* field, enter “36.0” and select “hours” from the drop-down.

5. In the *Time Step Control* section, click on the large button below *Time step size* to bring up the *Time Series* dialog.
6. In the *Time* column, select “hour” from the drop-down.
7. In the *Time step size* column, select “second” from the drop-down.
8. Enter two rows of data:
  - “0.0” in the *Time* column and “600.0” in the *Time step size* column.
  - “36.0” in the *Time* column and “600.0” in the *Time step size* column.
9. Click **OK** to close the *Time Series* dialog.
10. Continue to the next section.

## 4.5 Output Control

---

The ADH model has a lot of flexibility for controlling frequency of model outputs. For this model, solution data will be output every 15 minutes for the entire simulation duration.

11. On the *Output* tab, in the *Output Times* section, select the *Add by specifying a range* radio button.
12. Set the options as follows:
  - *Start at* to “0.0” and select “hours” from the drop-down.
  - *End at* to “36.0”.
  - *Increment* to “15.0” and select “minutes” from the drop-down.
  - Select “hours” from the drop-down for *View output times in*.
13. Click the **Add** button to populate the *Output Times* list on the left.
14. Click **OK** to close the *ADH Model Control* dialog.

## 4.6 Initial Conditions

---

ADH requires initial conditions be specified for a model run. The initial conditions could be interpolated solution data from a previous model run or conditions with simple hydraulics, which are numerically stable. For this model, initial depths that match the starting boundary condition will be specified.

1. Select *ADH | Hot Start Initial Conditions...* to bring up the *ADH Hot Start Initial Conditions* dialog.
2. In the *Depth (required)* section, select the *Constant water surface* radio button.
3. Enter “270.24” for the *Elevation*.
4. Click **OK** to close the *ADH Hot Start Initial Conditions* dialog.

A new “Initial depth” dataset now appears in the Project Explorer under the “ADH Hot Start” folder.

## 5 ADH Model Execution

---

Before running ADH, save an SMS project file containing all data associated with the project:

1. Select *File* | **Save New Project...** to bring up the *Save* dialog.
2. Enter “ADH\_Project” for the *File name*.
3. Select “Project Files (\*.sms)” from the *Save as type* drop-down.
4. Click **Save** to save the file and close the *Save* dialog.

The model contains two separate programs that run in sequence. Pre-ADH examines the model geometry and input file to look for errors. ADH is the numerical engine that generates solution data. Upon successful completion of Pre-ADH and running the model check, ADH is run.

To make sure SMS knows where to find the Pre-ADH and ADH executables, follow these steps:

1. Select *Edit* | **Preferences...** to bring up the *SMS Preferences* dialog.
2. On the *File Locations* tab, in the *Model Executables* section, scroll down to the entry for “ADH”. If a file path is already entered, that path will be displayed and steps 3–5 can be skipped.
3. Click the **BROWSE** button to the right of the entry to bring up the *Open* dialog.
4. Go to the *models\ADH* folder in the folder for SMS (e.g., *SMS 12.0 64-bit\models\ADH* for the 64-bit version) and select “ADH\_v4.5-WIN64.exe”.
5. Click **Open** to close the *Open* dialog.
6. Scroll down to the entry for “Pre-ADH”. If a file path is already entered, that path will be displayed and steps 7–9 can be skipped.
7. Click the **BROWSE** button to the right of the entry to bring up the *Open* dialog.
8. Go to the *models\ADH* folder and select “pre\_ADH\_v4.5-WIN64.exe”.
9. Click **Open** to close the *Open* dialog.
10. Click **OK** to close the *SMS Preferences* dialog.

To run Pre-ADH and ADH, do the following:

1. Select *ADH* / **Model Check...** and click the **OK** button if no model checks have been violated.
2. If there are problems found, go through the steps to fix the problems as outlined in the *Model Checker* dialog before moving on to step 3.
3. Select *ADH* | **Run ADH** to bring up the *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**.

4. Click **Run ADH** to start ADH.

The ADH model run may take a while to complete, up to 1-2 hours, depending on the speed of the computer.

## 6 Viewing the Solution

---

The primary output of ADH are files containing velocity vectors and depth values for each node in the mesh. These files can be automatically imported into SMS for viewing. Upon successful completion of ADH, the **Abort** button in the model wrapper will change to **Exit**.

Automatically import the files by doing the following:

1. Toggle on *Load solution* and click the **Exit** button. An “ADH Model Mesh Solution” folder will appear in the Project Explorer containing the ADH solution datasets.
2. In the Project Explorer, turn off all Map Data and Scatter Data and turn on Mesh Data (if not already done).
3. Click on “ADH Model Mesh” to make it active.
4. Select *Display* | **Display Options** to bring up the *Display Options* dialog.
5. Select “2D Mesh” from the list on the left then turn on *Contours* and *Vectors*.
6. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the drop-down.
7. Click **OK** to close the *Display Options* dialog.

Solution data for each output time can be visualized in the graphics window. For more information on visualization options, consult the “Data Visualization” tutorial.

## 7 Conclusion

---

For practice, experiment with changing various model parameters and observing the effects on the model output. Adjust roughness values and/or experiment with different levels of element refinement. Be sure to save new project files in each instance to avoid overwriting the original tutorial files.

This concludes the “ADH Hydrodynamics” tutorial.