How To Model Grassed Waterways Using Eagle Point

Table of Contents

Introduction

The following document explains how to model grassed waterways using Eagle Point. The process takes you through the steps of starting an Eagle Point project, importing survey data, making a surface model, defining a waterway alignment, obtaining existing ground cross-sections, defining a proposed grade profile, developing the channel shape, and creating reports and graphics.

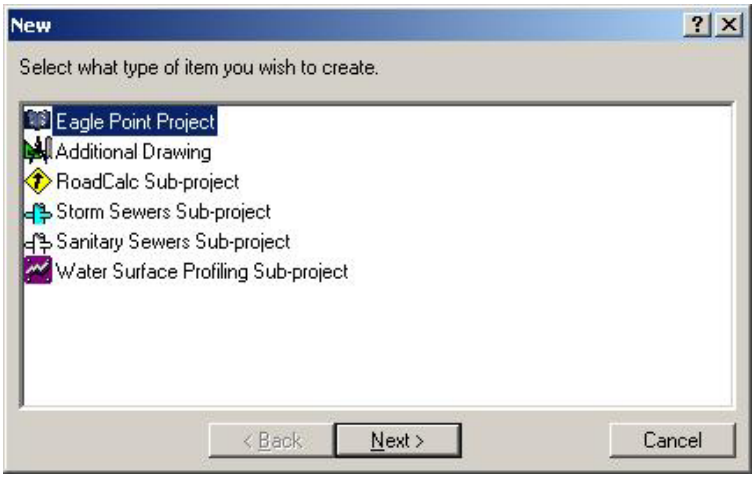
Commands in the ***Data Collection***, ***Surface Modeling*** and ***RoadCalc*** modules are used throughout the process. In particular, the **Define Channel** command (rcdefinechannel) found in ***RoadCalc*** is used to create the design channel of the waterway.

Hydraulic and soil conditions need to be calculated through USDA-NRCS approved applications such as the Ohio Engineering Programs or custom spreadsheets. The **Define Channel** command is then used to create a parabolic or trapezoidal channel shape and define the design conditions and locations. Once the design is processed, you are able to use the Offset/Depth report under the Step Through or Query Cross-section Data commands to query the depth of the channel surface measured from the original ground.

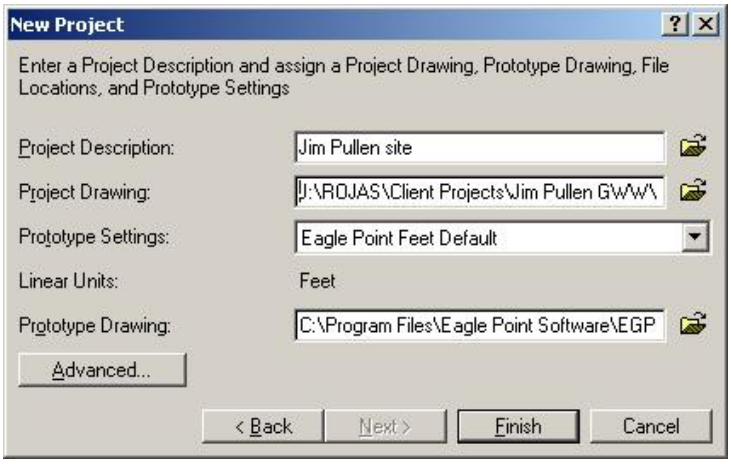
Setting Up an Eagle Point Project

You will start by creating a new Eagle Point project.

1. Open Eagle Point by double clicking on the **Eagle Point AutoCAD** or **Eagle Point Standalone** icon on the desktop.
2. Click on the **New** icon in the **Open** dialog box or select **File → New** from the **Eagle Point** menu.



3. Highlight **Eagle Point Project** and click **Next**.



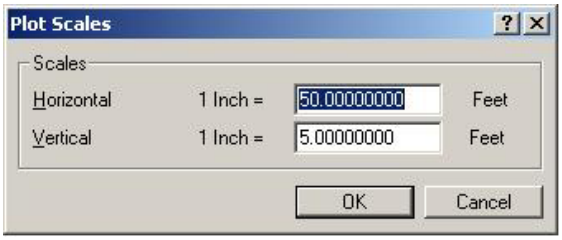
4. Type a **Project Description.**
5. Select a **Project Drawing**.

Note: If you do not have an existing drawing, Eagle Point will create it for you. In this case, you will need to select or create a directory where to store the project files and create a file name.

6. Select the **Prototype Settings**.
7. Click **Finish** when you are ready to continue.
8. In the **Open** dialog box, select the newly created project.

9. Click **OK** to open the project.
10. From the **Eagle Point** menu, select **Tools → Plot Scales**.

Note: All annotation and symbol sizing within Eagle Point is based on these plot scales. It is recommended to verify and/or change the project plot scales.



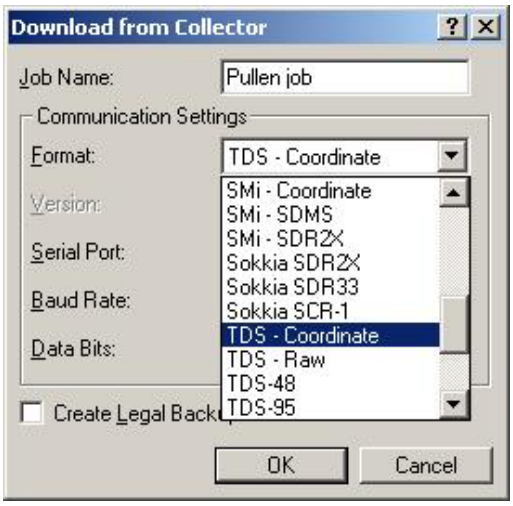
11. Enter the desired **Horizontal** and **Vertical** plot scales and click **OK** to accept changes.

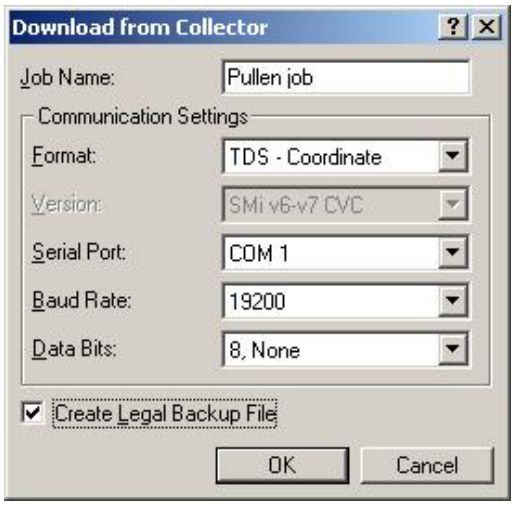
Importing Survey Data

We will assume that electronic methods were used to collect all project data. This allows for all survey data to be imported and processed electronically. We will now use ***Data Collection*** to import and reduce the survey.

1. From the **Eagle Point** menu, select **Products → Data Collection.**

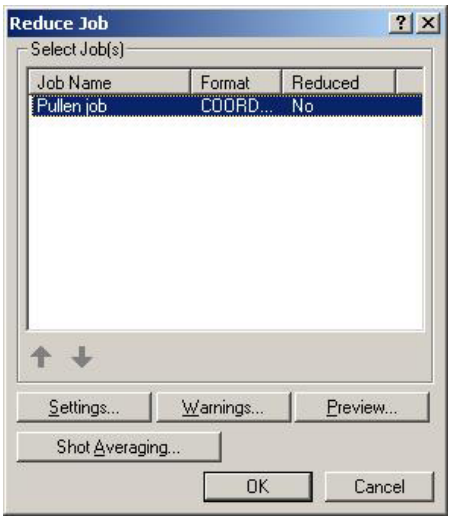
2. Select **Jobs → Download from Collector.**
3. Type in a job name, select a format, select a COM port, and set the transfer rate and parity to match your data collector.





4. Click **OK** and follow the instructions on the screens to transfer the data from your collector to the computer. Steps may differ depending on the type of data collector.
5. Select **Jobs → Reduce**.

*Note: To verify the reduction settings for the default field code or the linework table to be used, click on **Settings**. The survey file can be previewed before committing the data to the drawing. To do this, click on **Preview**.*



6. Select the **Job Name** and click **OK** to reduce the file to the drawing.

*Note: If you previewed the job, click on the **Place** button in the **Preview** window. If necessary, review and correct the errors or problems listed under the **Query Warnings** dialog box. To make corrections to the survey data, you may use the **Edit Formatted File** command found in the **Jobs** menu.*

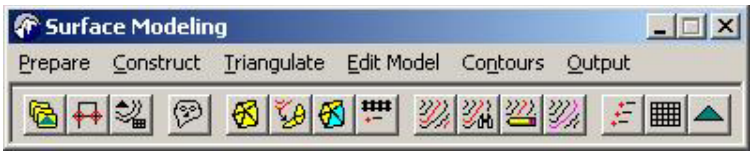
7. Close ***Data Collection***.

*Note: Other methods of importing or plotting survey data through Eagle Point include the use of the **Data Transfer** or **COGO** modules.*

Creating a Surface Model

Surface models (TIN) are the result of the triangulation of the points and breaklines in the drawing that represent existing or proposed conditions. ***Surface Modeling*** will triangulate to all the points, lines, arcs, polylines,and blocks in the drawing. To prevent ***Surface Modeling*** from triangulating to undesired objects, these can be masked or the layers that contain them can be frozen in the CAD layer manager.

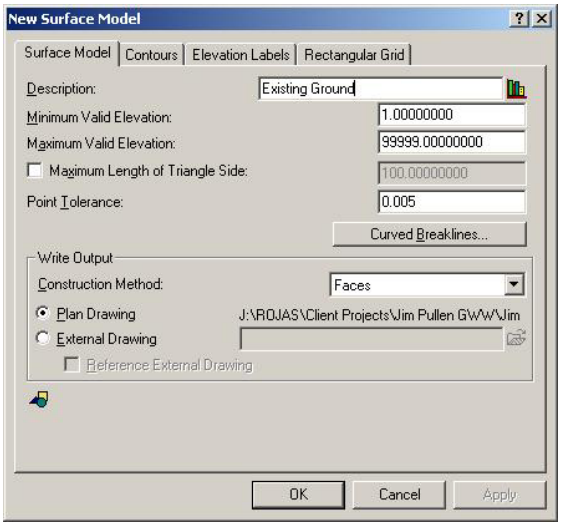
1. From the **Eagle Point** menu, select **Products → Surface Modeling**.

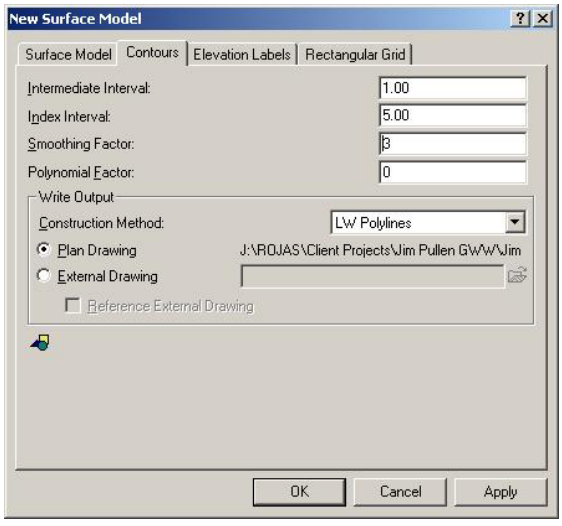


2. Select **Prepare → Manage Surface Models** and click on the **New** button.
3. In the **New Surface Model** dialog box, select the **Surface Model** tab.
4. Type in a **Description** for the surface.

Note: Elevation ranges can also be adjusted to filter out invalid data.

5. Click on the **Contours** tab.
6. Adjust the contour **Intermediate Interval** and **Index Interval** as necessary.
7. Adjust the **Smoothing Factor**. A smoothing factor of 3 to 5 will render good "smooth-looking" contours without increasing the drawing file size too much.
8. Set the construction method to **LW Polylines** to reduce the drawing file size.





9. Click on **OK** to accept all the changes in the **New Surface Model** dialog box.
10. Click **Close** on the **Manage Surface Models** dialog box.

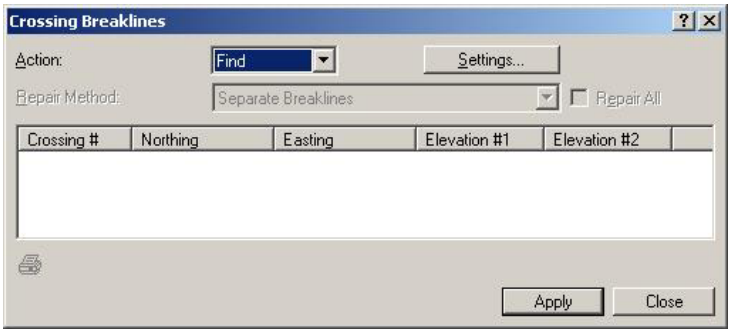
Note: To further restrict the information to be used during the triangulation process, you can draw a closed polyline to serve as a boundary. This boundary polyline can be drawn as a 2D or 3D line and can be selected during triangulation or predefined to save some time if you will triangulate a surface multiple times.

In case of internal features where you want to prevent the software from triangulating, such as buildings, lagoons or others, you may draw closed polylines to be used as void regions.

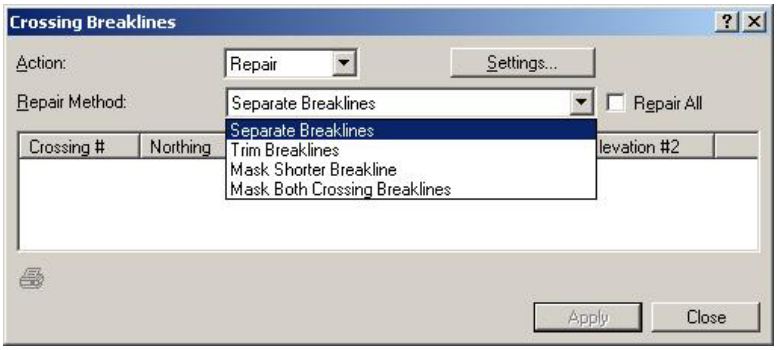
These polylines can also be drawn as 2D or 3D polylines and can be selected during triangulation or predefined to save some time if you will triangulate a surface multiple times.

If any lines that define significant features (*breaklines*) were drawn, it will be necessary to check for conflicting intersections. By finding and fixing any crossing breaklines, ***Surface Modeling*** is able to resolve elevation conflicts and have the TIN diagonals better match the linework. If the crossing breaklines are not fixed, ***Surface Modeling*** will not apply those specific lines to the triangulation.

11. Select **Prepare → Crossing Breaklines.**
12. From the **Action** list, select **Find**.
13. Click **Apply**.



14. If any crossing breaklines were found, use the **Show** and/or **Mark All** options found under the **Action** list to review the linework.
15. The **Repair** options under the **Action** list can be used to resolve the conflicts. Select the **Crossing #** to repair, then select the **Repair Method** from the list and click on **Apply.**

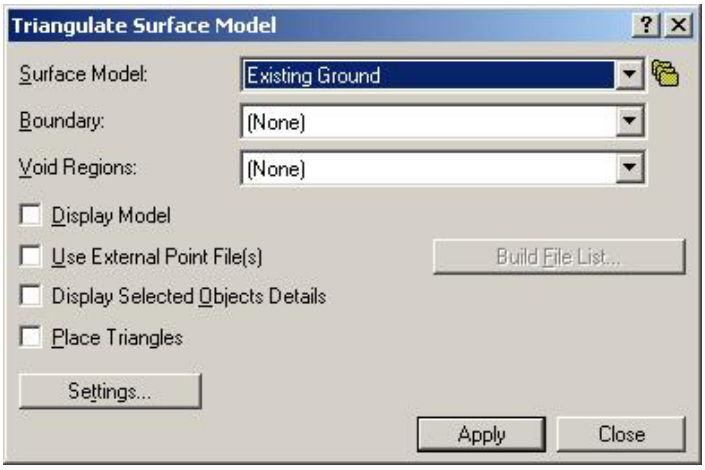


Note: Crossing breaklines can also be fixed using CAD commands.

16. Click on **Close**.

Now you are ready to triangulate the surface model.

17. From the ***Surface Modeling*** menu, select **Triangulate → Surface Model**.



*Note: If a boundary and/or void regions were drawn, select the appropriate option – **Select** or **Predefined** – under the **Boundary** and **Void Regions** pull-downs.*

18. Toggle **Display Model**. ***Surface Modeling*** displays the triangulation using temporary objects. The objects will disappear as soon as the drawing is redrawn by use of the zoom, pan, redraw, or regen CAD commands.

*Note: To place permanent triangles, toggle on **Place Triangles**.*

19. Click **Apply** to perform the triangulation.
20. Select the objects you want included in the triangulation.

Note: Any CAD selection method can be used, e.g., Window, Last, Crossing, ALL, Fence, Wpolygon, Cpolygon, etc.

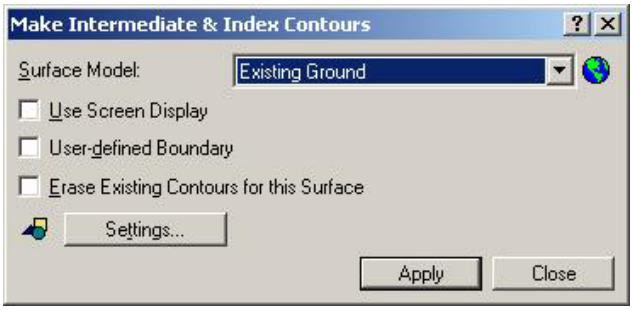
21. Accept the selection group by pressing **Enter**.

Note: Depending on the boundary and void region choices, you may be prompted to select the boundary and void regions.

22. Once the triangulation is done, you are returned to the **Triangulate Surface Model** dialog box. Click **Close**.

With a surface model created, you can now place the contour lines.

23. From the ***Surface Modeling*** menu, select **Contours → Make Intermediate and Index**.



24. Click **Apply**.
25. Once the contour lines are placed, you are returned to the **Make Intermediate and Index Contours** dialog box. Click **Close.**

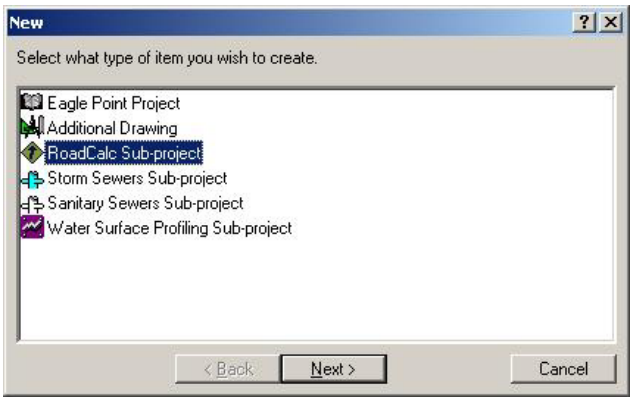
26. You can now close ***Surface Modeling*.**

Note: You can annotate the contour lines by selecting ***Contours → Annotate.*** *You can also annotate the elevation of any point in the surface model by selecting* ***Output → Place Spot Elevation Labels.***

Adding a RoadCalc Sub-project

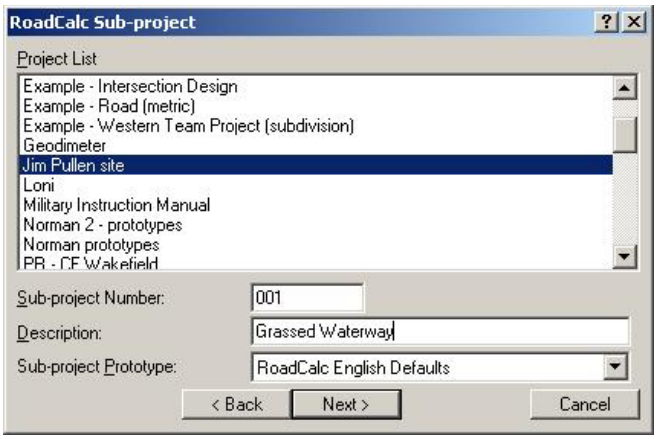
Now that you have defined a surface model, you need to add a ***RoadCalc*** sub-project to develop the alignment, existing ground cross-sections, original/proposed profiles, and channel sections for the waterway. The centerline alignment defined in a ***RoadCalc*** sub-project is the geometry that the program uses to base the cross-sections and profiles from.

1. From the main **Eagle Point** menu, select **File → New.**
2. Highlight **RoadCalc Sub-projec**t and click **Next**.



3. In the **RoadCalc Sub-project** dialog box, your current Eagle Point project is selected in the **Project List** and a proposed **Sub-project Number** is displayed.
4. Type a **Description**. Then click **Next**.

Note: If you have a project with multiple waterways, you may need to define a sub-project for each one of them, e.g., each controlling alignment must have its own sub-project.



5. In the **New Sub-project** dialog box, accept the **Use Project Drawing** selection by clicking **Finish.**
6. In the **Open** dialog box, the newly created **RoadCalc** sub-project should be highlighted. Click **OK** to open it.

Defining an Alignment

Alignments represent the two-dimensional horizontal geometry of a given baseline. The centerline alignment being the geometry from which ***RoadCalc*** bases the cross-sections and profiles.

Alignments can be created numerically or graphically in the plan view graphic. Alignment vertex (PI) and curve data can be entered in the alignment data dialog boxes using nodes, coordinates, or angles and distances. Or, you may use CAD tools to draw lines or arcs for the alignment directly into the plan graphic. The steps that will be followed in this document are based on the assumption that the alignment will be entered graphically.

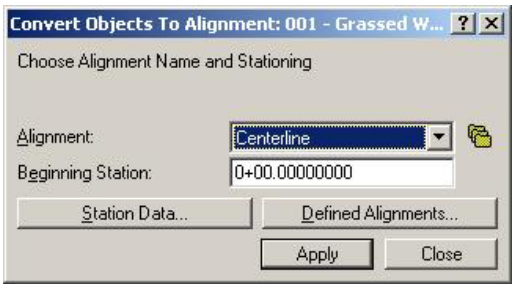
1. You may want to freeze some layers in CAD to have an easier view of the site.
2. In CAD, draw a 2D polyline representing what you will use as the waterway centerline.

Note: If you have proposed centerline shots, you can snap the polyline to them. All distances and angles can be later modified through the ***Edit Data*** *dialog box found under* ***Alignments → Edit Data.***

3. In ***RoadCalc***, select **Alignments → Convert Objects to Alignment.**
4. You are prompted to select the object or objects in CAD. Select the object(s).

Note: Any CAD selection method can be used, e.g., Window, Last, Crossing, ALL, Fence, Wpolygon, Cpolygon, etc.

5. Press **Enter** to accept the selection.
6. You are prompted to '*Pick a point near the beginning of the alignment*'. Select a point close to the end where you want the stationing to begin.
7. In the **Convert Objects to Alignment** dialog box, make sure that *Centerline* is the selected **Alignment.**



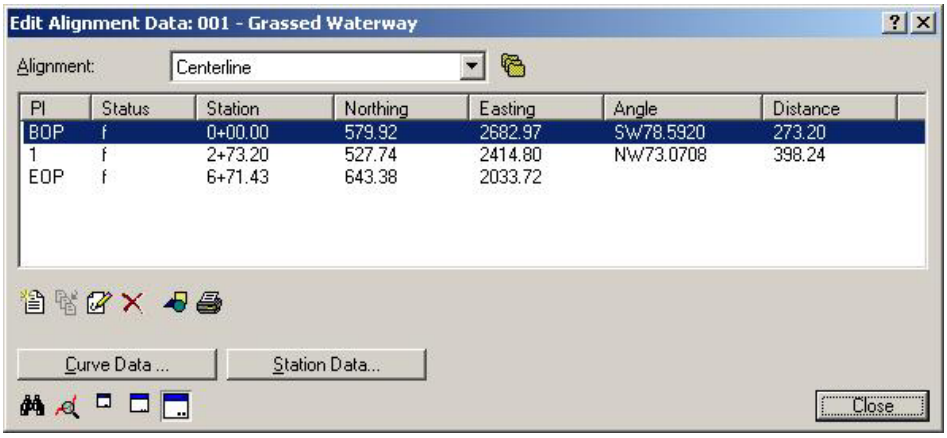
8. Input a **Beginning Station** if it is other than *0+00*.

Note: If you have a known station point along the centerline, click on ***Station Data*** *and then* ***Reference Station*** *to pick that point.* ***RoadCalc*** *adjusts the project stationing accordingly.*

9. Click **Apply**.

Note: As the selected polyline gets converted into a ***RoadCalc*** *alignment, it changes layer and color to match the CAD settings – red by default.*

10. To verify the alignment, select **Alignments → Edit Data**.



To round up station, distance, and/or angular values for the waterway centerline, click on the **Modify PI** button and make the necessary adjustments.

11. Click on **Close** once you are done reviewing the alignment data.

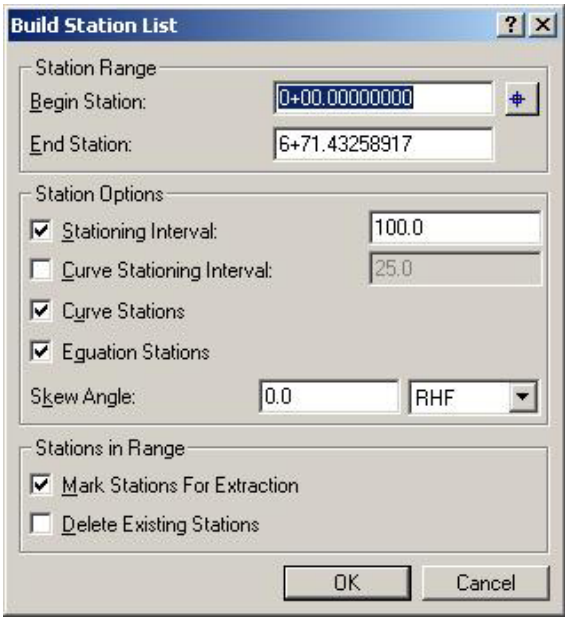
Note: To avoid cross-section overlaps or "blind spots" you can add a 1-foot radius curve to the alignment vertices (PI). This prevents sharp angles from occurring.

Extracting Original Ground Cross-sections

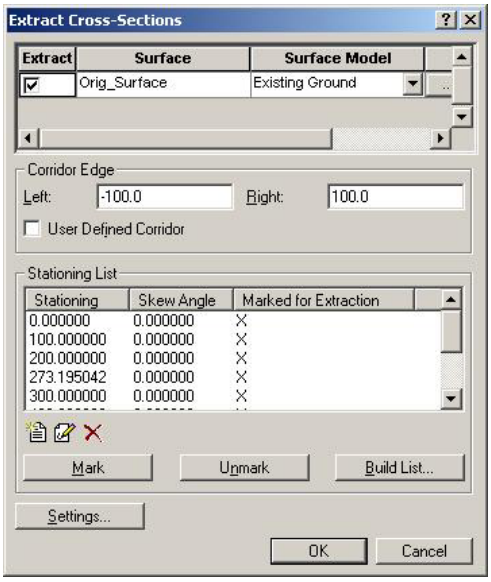
The commands found in the **Cross-Sections** menu allow you to input, edit, and translate the cross-sectional data. Enter and edit the data directly with the edit cross-sections routine, or bring the cross-section data into ***RoadCalc*** from outside sources using the extract cross-sections or the import cross-sections routines.

You will take advantage of the fact that a surface model was created earlier using ***Surface Modeling***.

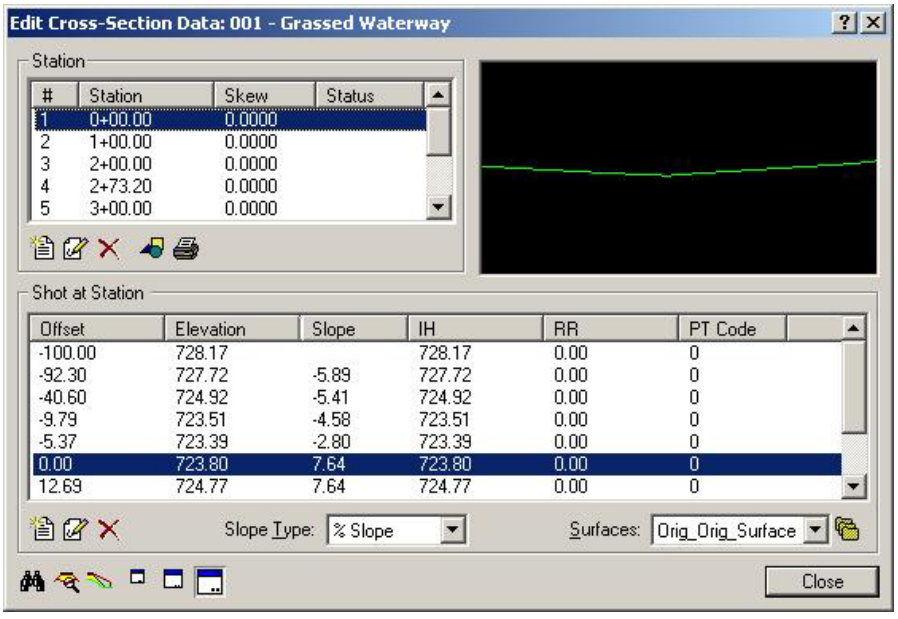
1. In ***RoadCalc***, select **Cross-Sections → Extract Cross-Sections.**
2. The **Build Station List** dialog box displays.



3. By default, ***RoadCalc*** proposes that cross-sections be extracted for the entire project. You can accept the default **Station Range** or adjust it.
4. Input the **Stationing Interval** at which you want to obtain cross-section data.
5. Leave **Curve Stations** toggled for ***RoadCalc*** to extract cross-section at the vertices of the alignment.
6. Make sure that **Mark Stations For Extraction** is toggled on.
7. Click **OK.**
8. The **Extract Cross-Sections** dialog box displays.

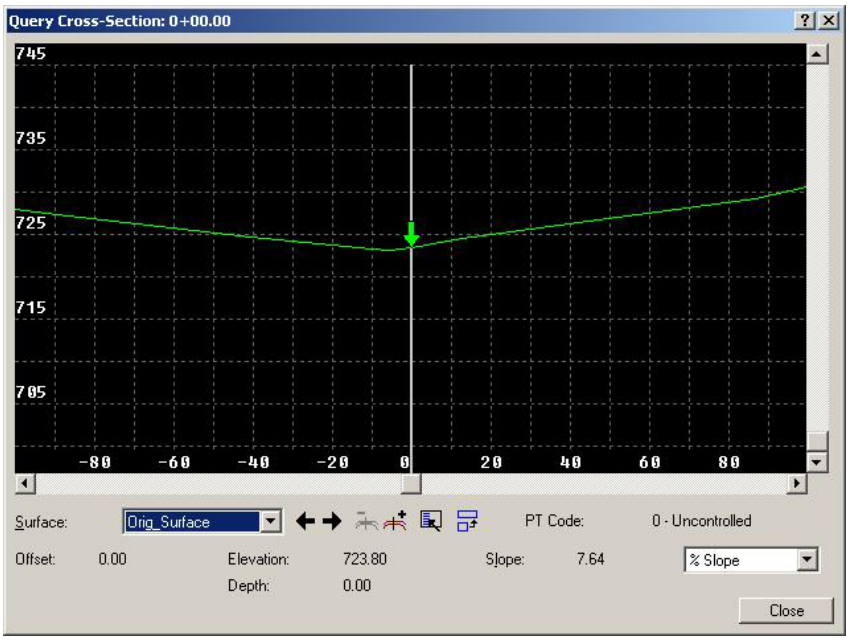


9. Toggle on the surface for which you will **Extract** data.
10. Select the **Surface Model** to be used.
11. If needed, modify the extent to which data will be extracted by entering new values for the **Corridor Edge**.
12. All stations should be **Marked for Extraction** in the **Stationing List**.
13. Click **OK** to extract the cross-sections. As a result, ***RoadCalc*** draws "shadow" lines at each cross-section station.
14. To review the cross-sections in ***RoadCalc***, select **Cross-Sections → Edit Cross-Section Data.**



*Note: You can select the desired station from the **Station** list. The **Shot at Station** list reports the offset, elevation, slope, etc., data for each shot.*

15. To preview the cross-sections, click on the **Query Cross-Section** button on the lower left hand corner of the **Edit Cross-Section Data** dialog box.



The left and right arrows move the cursor along the selected surface to obtain offset, elevation, and slope data. The (**+**) and (**-**) buttons move through the stations. The list button allows you to select a given station. The scale view button adjusts the displaying settings.

16. Click **Close** to close the **Query Cross-Section** dialog box.
17. Click **Close** to close the **Edit Cross-Section Data** dialog box.

Defining a Design Profile

The commands in the ***Profiles*** menu define profiles and specify other design constraints that ***RoadCalc*** uses to generate design cross-sections. ***RoadCalc*** uses the centerline design profile to determine at what elevation to place the typical sections on the original cross-sections.

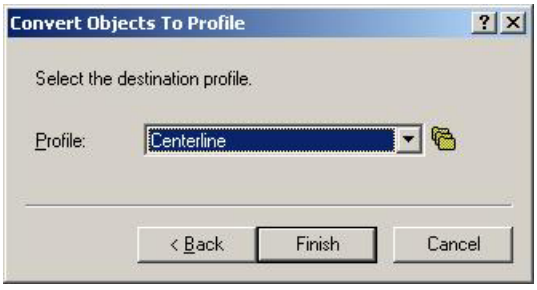
The profile data can be entered graphically or numerically. ***RoadCalc*** automatically supplies you with an original ground centerline profile based on cross-sections, so you may sketch a design profile using CAD and then convert those objects into a profile that can be modified using the **Edit Profile Data** command. If you know the station, elevations, and curve data, that information may be entered using the **Edit Profile Data** dialog boxes.

For the purpose of this document, we will assume that the profile data will be entered graphically.

1. In ***RoadCalc***, select **Profiles → View Profile Graphics**. ***RoadCalc*** creates the original ground profile in a new CAD drawing by default.
2. In CAD, draw a 2D polyline that represents the proposed profile (grade line) of the waterway centerline.

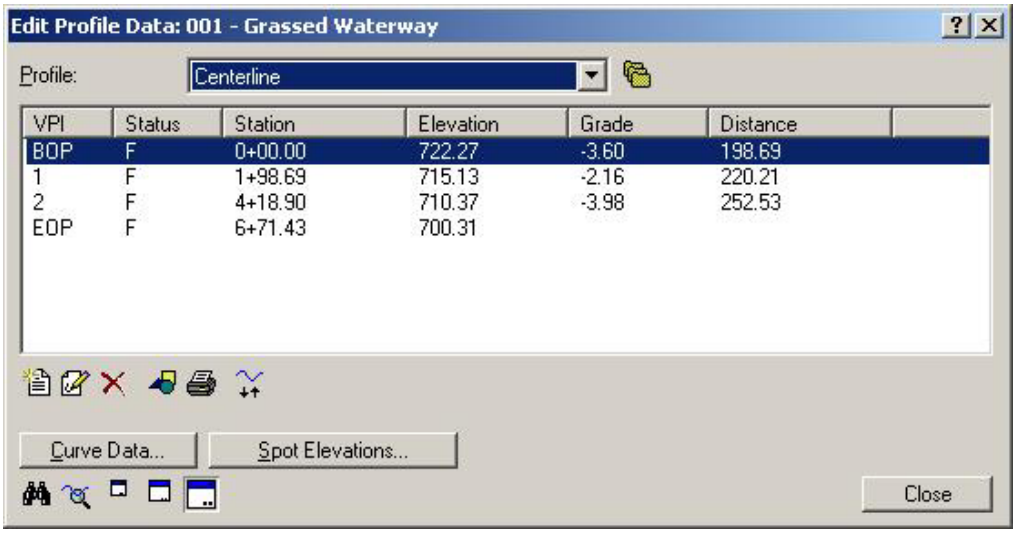
*Note: Grades and distances can be modified using the **Edit Data** dialog box.*

3. In ***RoadCalc***, select **Profiles → Convert Objects to Profile**.
4. In CAD, you are prompted to "*select objects*". Select the polyline that was drawn in the previous step and accept the selection set.
5. Click on **Next** on the **Convert Objects to Profile** dialog box.
6. Select *Centerline* and click on **Finish**.



*Note: The line changes layer and color as it is converted into a **RoadCalc** design profile.*

7. To verify and/or adjust the profile in ***RoadCalc***, select **Profiles → Edit Data**.



8. To adjust the proposed profile station – depth and/or the distance – grade you can modify the profile vertices (VPI). Select the desired vertex and click on the **Modify VPI** button.
9. Click on **Close** once you are done adjusting or reviewing the profile.
10. In CAD, save the drawing.

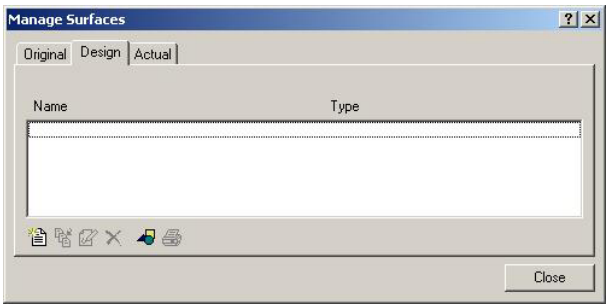
*Note: If you need to report the original ground profile, you can add a new design profile named "Original Ground" through **Profiles → Manage**. Then you can convert the profile that **RoadCalc** automatically created into the "Original Ground" profile or you can extract a profile from the surface model into the "Original Ground." With the profile created, you can now use the **Generate Reports** command found in the **Profiles** menu.*

Creating Channel Shapes

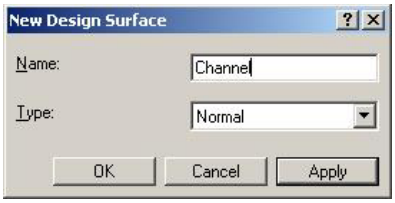
The **Define Channel** command was added to ***RoadCalc***. The key-in command of "**rcdefinechannel**" displays the **Define Channel** dialog box. This command was designed for the NRCS and other conservation engineering organizations to aid in the design of grassed waterways.

The **Define Channel** command allows you to launch an external program to determine the shape for the channel. It also automatically builds parabolic and trapezoidal channel shapes according to the widths, depths, and side slopes specified. Once the channel shapes have been constructed, they can be assigned to profile grade change (vertex/VPI) locations. With the channel shapes specified at the profile grade change locations, a default condition is built and the channel shape and conditions are automatically entered to the design locations so you are ready to process the design.

1. To develop the channel typical section, you need at least one design surface. In ***RoadCalc***, select **Cross-Sections → Manage Surfaces**. Then click on the **Design** tab.

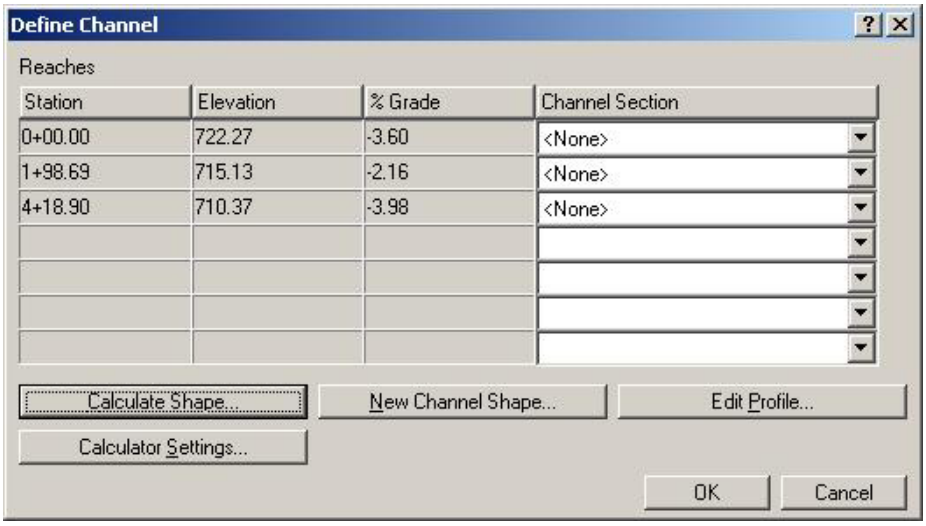


2. Click on the **New Surface** button.



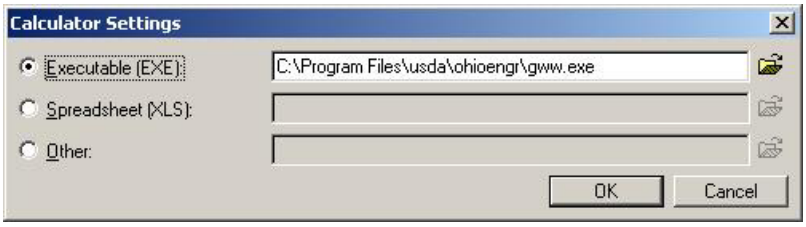
3. Type the new surface **Name** in the **New Design Surface** dialog box. For example, type *Channel* as the design surface name.
4. Click **OK**.
5. Click **Close**.
6. At the CAD command prompt, type **rcdefinechannel** and press **Enter**.

7. The **Define Channel** dialog box displays.



*By default, the **Define Channel** dialog box shows the beginning elevation and grade for each reach. The first time you open the command, no Channel Sections are selected. Once sections have been defined and assigned they are appropriately displayed.*

8. To configure the application to be used to calculate the waterway design shape, click on **Calculator Settings**.



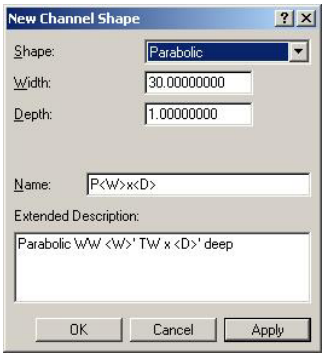
*By default, **RoadCalc** searches for the Vegetated Waterway Design executable (GWW.EXE) from the USDA-NRCS Ohio Engineering program suit. If this executable is not installed in the default program path, you need to adjust the default path. To use a different USDA-NRCS program, spreadsheet, or other program, click on the appropriate toggle for **Executable, Spreadsheet**, or **Other**. Then click on the Browse icon to the right of the field to browse and select the application to be used for this project's calculations.*

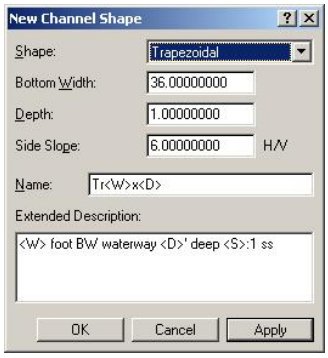
9. Click **OK** to accept the settings.
10. Click on **Calculate Shape**. This opens the application picked for the calculations.

For example:

*The **Define Channel** command supports both **parabolic** and **trapezoidal** channel shapes. Compute the **width** and **depth** of the desired waterway design shape for each reach using your NRCS application.*

11. Once done with the calculations, close the application to return to **Eagle Point.**
12. Click on **New Channel Shape**.

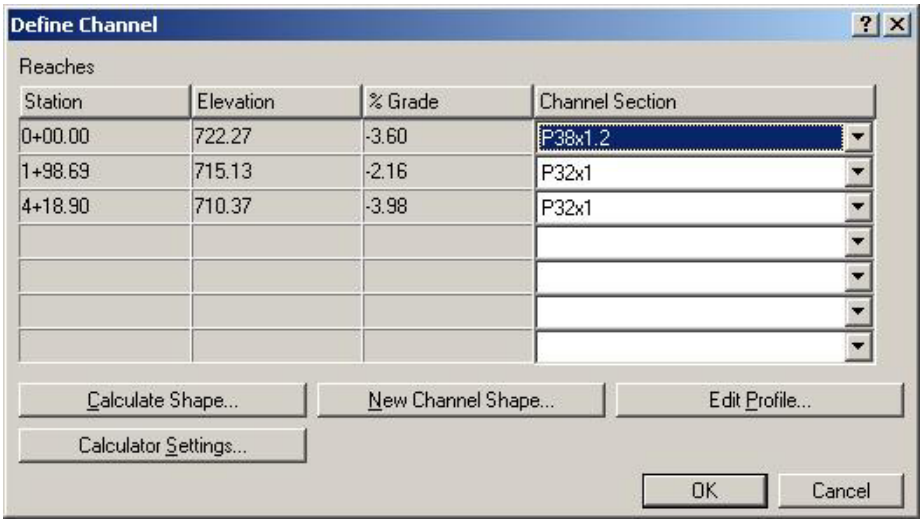




13. Select the desired **Shape** from the list.
14. For **Parabolic** shapes, type in the calculated **Width** and **Depth**. A default **Name** and **Extended Description** are proposed. The <*W*> and <*D*> entries in the Name field are replaced automatically with the values entered once the **OK** or **Apply** button is clicked.
15. For **Trapezoidal** shapes, type in the calculated **Bottom Width**, **Depth**, and **Side Slope**. A default **Name** and **Extended Description** are proposed. The <*W*> and <*D*> entries in the Name field are replaced automatically with the values entered once the **OK** or **Apply** button is clicked.

*Note: If you need to enter more than one shape, click Apply to accept each entry and keep the dialog box open. If you are entering only one shape definition or are at the last entry, click **OK** to accept the entry and close the **New Channel Shape** dialog box.*

16. Click the **Channel Section** list and select the desired shape for each waterway reach.



17. Click **OK** to close the **Define Channel** dialog box.

Define Channel creates and assigns a default Condition Table to the sub-project. The Condition Table is based on the cut and fill slopes defined in the selected ***RoadCalc*** Sub-project prototype and is assigned to all the stations in the waterway.

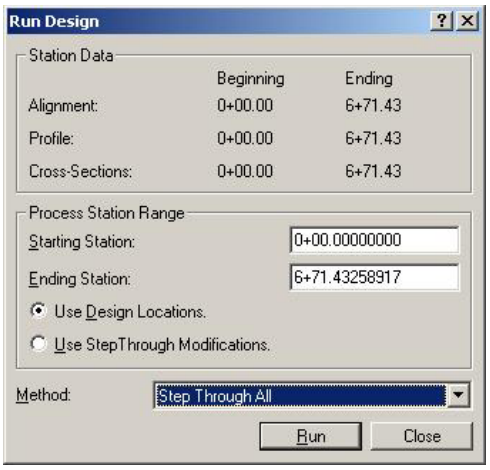
*Note: If you need to adjust the profile data, click on **Edit Profile**. The **Define Channel** dialog box allows you to access the application you usually use to calculate the waterway shape design.*

*Note: You can predefine some of the waterway design settings, including the program or spreadsheet used to calculate the waterway design shape, by creating a **RoadCalc** Sub-project Prototype that includes that information.*

Processing the Design

The **Run Design** command is used to "pull" the individual design elements together to produce design cross-sections. You may choose to have ***RoadCalc*** automatically process all stations in the processed range and then view the designed cross-sections in the Edit Cross-sections Data dialog box. Or you may choose to "step through" all of the processed cross-sections one at a time, or "step through" the cross-sections that produce warning messages during processing. Either "step through" method displays the cross-sections in a window for quick viewing and editing.

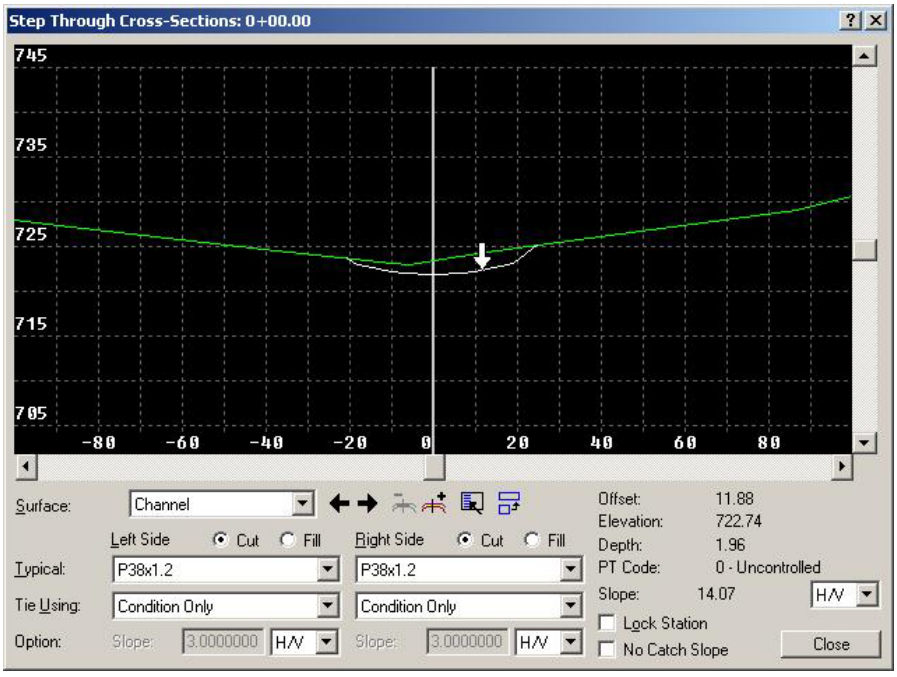
1. In ***RoadCalc***, select **Process → Run Design**. You may enter the station range over which you want to process the design.



2. To view the designed cross-sections as they are processed, select **Step Through All** from the **Method** list.

*Note: If no "step through" method is selected, cross-sections can be reviewed through the **Edit Cross-Section Data → Query Cross-section** command once **RoadCalc** has finished processing the design.*

3. Click on **Run** to process the cross-sections.
4. The **Step Through Cross-Sections** dialog box displays.



*The channel Offset and Depth can be reported at various breakpoints in the cross-section. Select the Channel surface from the **Surface** list. Then click on the left or right arrows to displace the cursor along the surface. The offset and depth values are reported in the respective fields in the dialog box.*

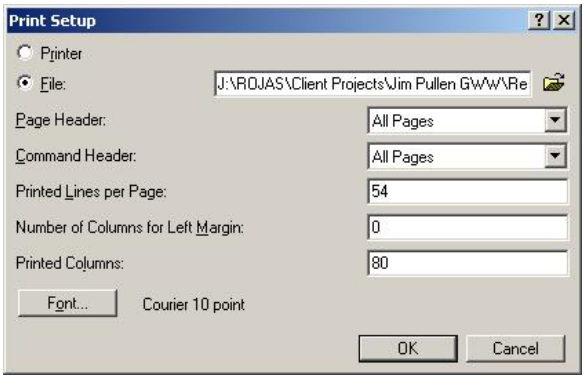
5. Click the **View Previous** or **View Next** cross-section buttons to display different stations.
6. Click **Close** to finish.

Reports

RoadCalc provides multiple output features, including alignment and profile reports, elevation and depth reports, cross-section staking, volumes, and plot sheets (cross-section and plan and profile sheets). Alignment and profile reports are available through the **Alignments** and **Profiles** menus while the **Output** menu provides access to various other commands.

While the printout format is controlled by the **Print Setup** command, plotting will be done and configured in CAD. You can specify if you want the printout to be sent to a printer or file, if you want page headers, command headers, how many lines are printed per page, width to the left margin, and how many columns are printed.

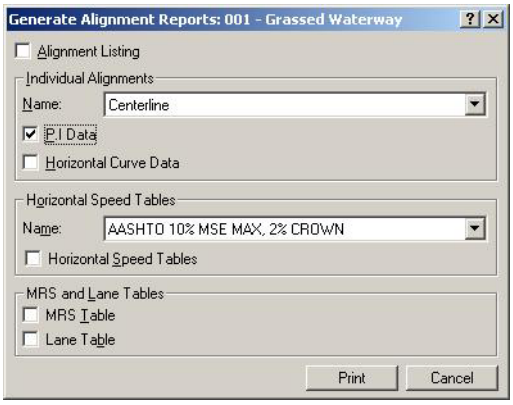
1. From the **Eagle Point** menu, select **File → Print Setup**. Select **Printer** or **File**. You can also modify the header settings.



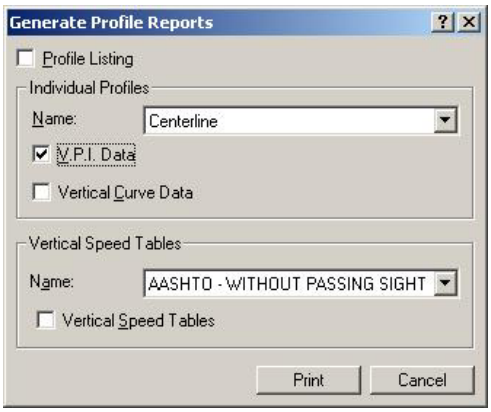
2. Click **OK** to accept changes and close the dialog box.

You will now report the waterway centerline and the design profile.

3. From the ***RoadCalc*** menu, select **Alignments → Generate Reports**.



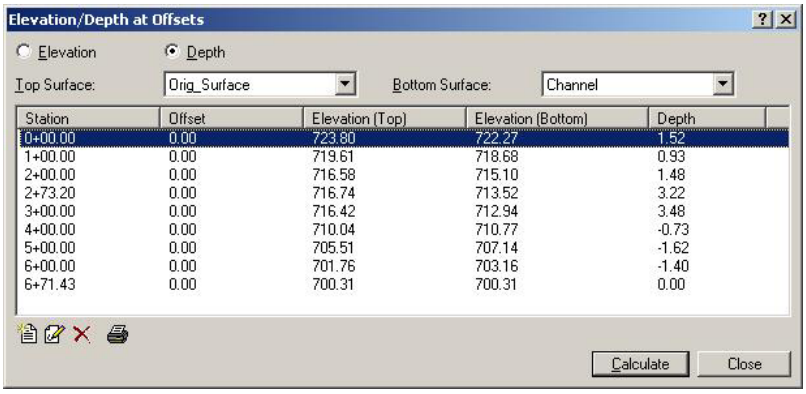
4. Toggle on **P.I. Data.**
5. Click **Print.** ***RoadCalc*** reports the waterway centerline information.
6. From the ***RoadCalc*** menu, select **Profiles → Generate Reports**.



7. Toggle on **VPI Data.**
8. Click **Print**. ***RoadCalc*** reports the waterway proposed profile information.

Follow the next steps to report the cut values at the centerline of waterway.

9. From the ***RoadCalc*** menu, select **Output → Elevation/Depth at Offsets**.



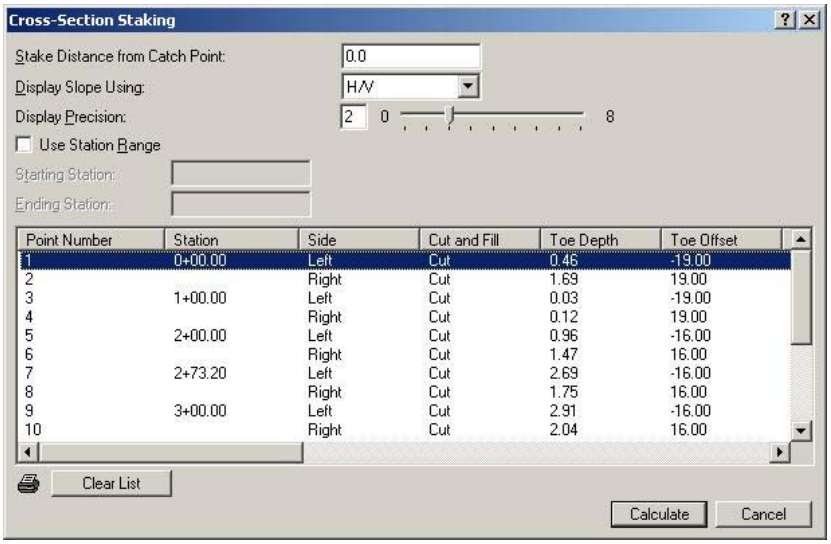
10. Toggle on **Depth**.
11. Select your original surface as the **Top Surface.**
12. Select *Channel* as your **Bottom Surface**.
13. Click on the **New Offset** button.
14. Type *0* and then click **OK**.

Note: By using a zero offset, you will obtain the centerline profile cut values.

15. Click **Calculate**.
16. Click on the **Print Elevation/Depth at Offsets** icon.
17. Click **Close**.

Follow the next steps to report the cut values at the edges of the waterway.

18. From the ***RoadCalc*** menu, select **Output → Cross-Section Staking**.



19. Click **Calculate**.

Note: To view all data, use the scroll bar to display the remaining data.

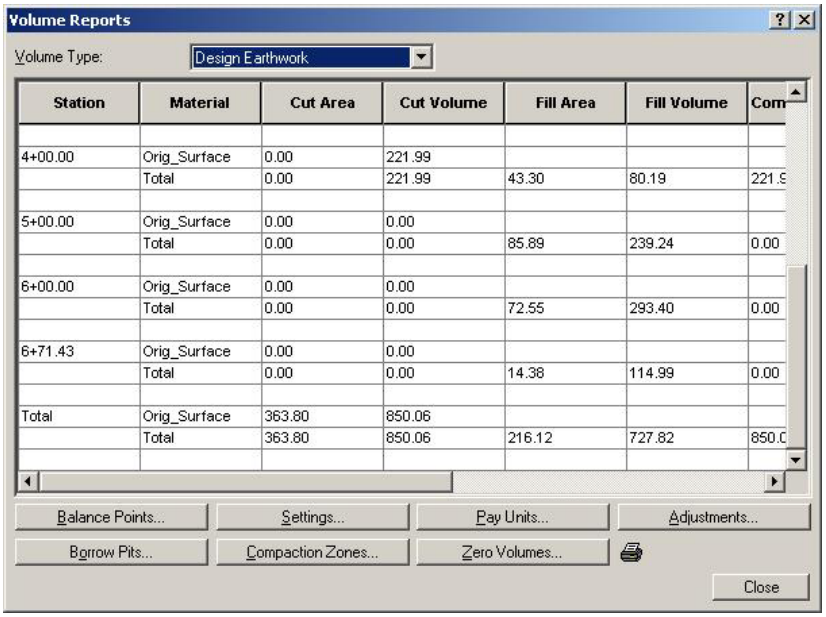
20. Click on the **Cross-Section Staking Print Options** icon.
21. Select which items to include in the printout by toggling on or off the **Print** field.

*Note: The **Width** field controls the number of characters that will fit on the respective fields.*

22. Click on **Print**.
23. Click **Cancel** to close the **Cross-Section Staking** dialog box.

To report the volume (earthwork) calculations, proceed with the following steps.

24. From the ***RoadCalc*** menu, select **Output → Volumes**.



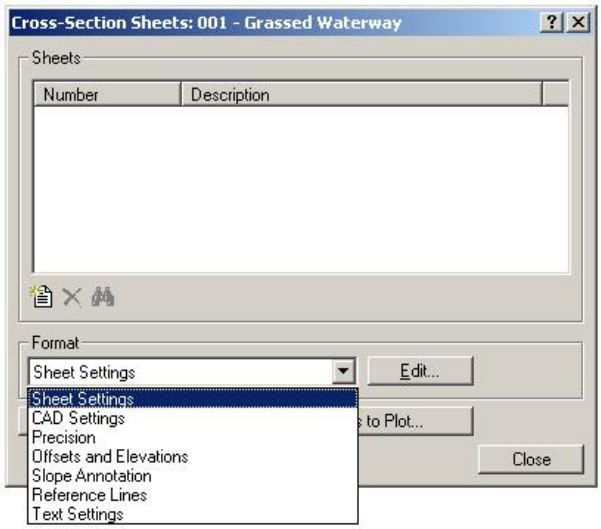
25. Click on the **Print Volume Report** icon.
26. Select **Raw Volumes.**
27. Click on **Print.**
28. Click **Close**.

Graphics

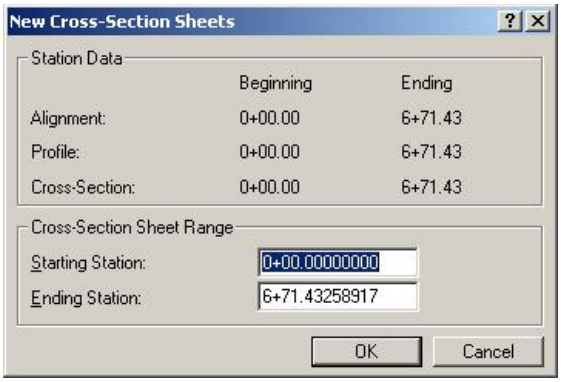
You will now create the plot sheets – cross-section and plan and profile sheets – for the waterway project. For each type of sheet created, numerous parameters controlling sizes, CAD settings, and other text annotation may be set and saved to prototypes that can be used in other ***RoadCalc*** sub-projects.

To obtain cross-section sheets, follow these steps:

1. From the ***RoadCalc*** menu, select **Output → Cross-Section Sheets**. To adjust the sheet or CAD settings or any other annotation, select an item from the **Format** list and then click **Edit**.



2. Click on the **New Cross-Section Sheets** icon. You can adjust the station range.



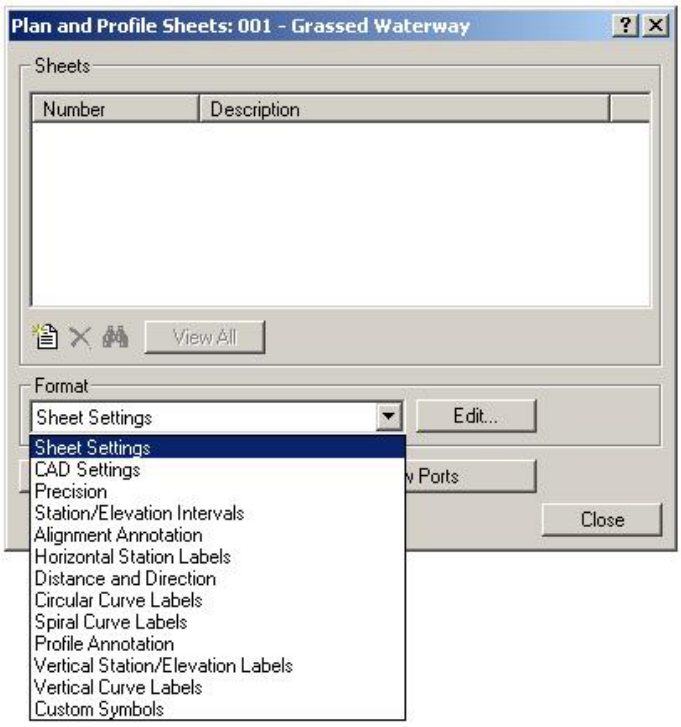
3. Click **OK.**

Note: To review any specific sheet, select the sheet number and then click on the ***View Cross-section Sheets*** *icon.*

4. Click **Close**.

To obtain plan and profile sheets, follow these steps:

1. From the ***RoadCalc*** menu, select **Output → Plan and Profile Sheets**. To adjust the sheet or CAD settings or any other annotation, select an item from the **Format** list and then click **Edit**.



2. Click the **New Plan and Profile Sheets** icon. You can adjust the station range.
3. Click **OK**.

*Note: To review any specific sheet, select the sheet number and then click on the **View Plan and Profile Sheet** button icon.*

*For more information related to the use of **Data Collection, Surface Modeling, RoadCalc** or any other **Eagle Point** programs, refer to the on-line help, documentation, tutorials, or call the **Eagle Point** support line.*

Index

Premium VIP Support: 888.637.6507
Toll-free 800.678.6565
Fax 563.556.5321
4131 Westmark Drive
Dubuque, IA 52002-2627
---
www.eaglepoint.com