

OPENING DISTRIBUTED MODELS WITH SVOFFICE 2009

Presented by: Dr. Murray Fredlund, President of SoilVision Systems Ltd.

Hi, my name is Murray Fredlund and I'm here today to talk to today you about Opening Distributed Models within the *SVOffice 2009 Next Generation geotechnical software suite*.

Getting Started

The first thing when you start this software you may have a particular application in mind that you want to see with the software. Applications are roughly grouped by category in the software. If you look under **Projects** you may want to analyze an EarthDam. By clicking on **Earth Dam** in the **Projects** box a list of models available/distributed are displayed in the models list. This says under **Earth Dams** there are 30 models currently available and as you mouse over each model a thumbnail appears to show the model view and brief description.

Please note **Models** are listed by **2D/3D**, **Type**, the **Application** designed in and the time **Since Last Edit**. The same process can be applied in the **Projects** box. By selecting a **Project Name** you can view the details of each project folder. Of note, there are several slope folders such as Geomembranes, Heap Leach, Mine Stopes, Mine Tailings, Ponds, Pits, Roadways and more than three groups of Slope Stability Models.

Searching and Selecting Models

You can also narrow your search list through the **List Criteria** box. You can select the **Application** (such as only models created in the Slope Stability package). As well you can select by **Model Origin** (note: some models are created as demo models and are too complicated to run in the Student Version but are encoded to run in the full version to demonstrate a certain feature. They cannot be changed but they can be run as examples of how the software would perform under that scenario). **Student Models** adhere to all limitations of the Student Version. Any model can be opened and viewed in the Student Version. This is very useful because you can open models that colleagues send to you. **Category** can also be used to filter the list of models. For example, all models published in our Verification Manuals can be selected through the Verification category and these models exist in the matching manual. **Keywords** search allows you to search for specific words. For example, you can search for models with an Evaporative Component, and they are displayed in the folder.

This gets you to the point where you can search through the software. If we want to select Earth Dam Model and pick any particular model we can double click on the model or select **Open** to open the model. To run the model, select **Solve/Analyze**. The program writes a description file to the finite element solver, the solver grids up the model file and displays **Done** on the solver. To view the results in **AcuMesh**, click on the orange button (top left corner) that displays the results in **AcuMesh**. There will be more about this feature in future presentations.

Starting a New Project

To start and create a new project select **New** (located between Projects and Model) and add a Project Name. For example: AANew Project. The software asks you create a new model within that project. In our example, select the **Application** (SVSlope), **System** (2D), **Unit** (metric),

Slope Direction (left to right) and **World Coordinates**. Once you select **OK**, the model is created and you are prompted to specify the grid spacing (2m:2m). Once you've created your new model, save the model before closing. The next presentation will talk in more detail how to create models.

Models can also be batch run. To select a group of Slope Stability models, hold **SHIFT KEY** use the mouse to select a group. Then select **Models/Batch SVSlope Models/Analyze**. Click **Start** and the solver analyses the models. When the results are displayed select **Graph** to view the graph. This feature is useful for Batch Analysis of these types of models.

<End of transcript>