FEM VISUAL OF OUTPUT 2D

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

Introduction
Hi, my name is Murray Fredlund. Welcome to the session FEM Visual of Output within the *SVOffice 2009 Next Generation* geotechnical software suite.

Getting Started
What we have here on the screen is a typical finite element model of an EarthDam. You can see there are a number of different regions which make up the EarthDam and make it interesting. The legends can be moved around on the screen and placed wherever you want to see them in the front end. We have essentially an upstream water-head and we have its exit point somewhere downstream. In order to look at the results, once the model has been analyzed by either clicking the **Play** or the **Solve/Analyze**, what we can do is move to the AcuMesh backend. We can do that through either the window command, going to the AcuMesh command, or we can click on the AcuMesh icon and then the results are brought up in the AcuMesh backend. You can see all the contours below the water-table are plotted as varying degrees of the blue contour and anything above the water-table is plotted as a grey. At the start, what we can do is zoom in to see a little more detail about what we're doing. At first you can see that we're not getting enough definition about where our plot regions are. Under the **Plot** menu you can select **Regions**. This allows us to turn on the original coloring for each region and see where the region divisions are. So we can change and set the material color for any particular region in our numerical model. Alternatively we can set the boundaries (another way to see the region differences). We can select a few boundaries and set the weight of those boundaries to be a little higher – *select all> show region boundaries and set them to the number 2*. This way the region boundaries appear a little bolder in the numerical model. It's possible that these regions can be selected one at a time with the CTRL key or a group of them can be selected by holding the SHIFT key and pressing the starting and ending region and then a group of these regions can be selected.

In the current model we have plotted all of the regions as a separate color. The water table still shows up. But we may want to enhance this model a little bit by not plotting the mesh in some regions. This can be accomplished by selecting the **Mesh** dialog and selecting certain regions that will not have any mesh associated with them. Holding the SHIFT key and selecting the various regions turns off the mesh for that region. Mesh display can be selected per region. Then, as well, you may want to select some of those regions and plot contours. We can do that through the **Contours** toolbar item or through the **Contours** on the menu item. We're going to select which variable we want to contour, in this case. For example, we may want to contour pore water pressure and we may want to select that we want to contour that in all regions or just in some regions. So we can select and hide and show various contours in various regions. As well, we can also select the type of contouring, how many levels we have (**Contour Levels**) and there is a fair amount of functionality included with the output modification. It may be of interest to the user to plot streamlines on the numerical model. This can be quite easily done under the **Streamtraces** model menu. Individual ones can be plotted by clicking on the interface and clicking at certain places in the model. **Streamtraces** will then be plotted based on that

location. They will be extended to the front of the model and all the way through the model to the end.

<End of transcript>