

## FEM VISUAL OUTPUT OF 3D

---

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

### Introduction

Hi, my name is Murray Fredlund. Welcome to the session 3D FEM Visual of Output within the *SVOffice 2009 Next Generation* geotechnical software suite.

### Getting Started

The purpose of this video is to illustrate to the user how to do standardized visualization of a 3D finite element model. In this case we've pulled up an example model out of the Ponds project (**Model: pstr01**) and, as you can see, this is a very simple model defined with two surfaces and flow from an upper reservoir down to a lake. We want to see what the flow regime looks like down towards the lake. This model has already been run. Normally you would solve it, analyze it first, once you've analyzed it you'd go to the AcuMesh backend visualization software. So we'll proceed there right now. What we're going to look at is typical means for producing visualization aspects of this model that can be utilized in a consulting report. Here is the default contours that come up in SVFlux. The contour pattern that comes up by default highlights the location of the water table so everything beneath the water-table is colored in various shades of blue and everything above the water-table (that is unsaturated) is colored in various shades of brown. The user should know, first of all, that the contouring levels can be changed. There are a lot of different types of contours that can be changed and visualized in the software. Contour settings are currently set to blue and brown. They can be changed to Autumn, pastels and a variety of other color schemes that the user can use. It's also possible to change the total number of levels such that more default can be seen in the contour levels. The visualization of the mesh can also be set for different regions. In other words, the user may desire to show mesh in certain areas (rather than show it all). So the mesh can be turned off in certain specific areas and enabled. Here, in this case, if we turn off the mesh in this middle zone, and leave it translucent, then we can also have a look at the Iso-Surfaces.

### Plotting 3D Iso-Surfaces

In this case we're going to plot Heads and plot Iso-Surfaces at every Head contour. Plotting Iso-Surfaces are found under **Plot>3D Iso-Surfaces** and we're going to plot them at each contour level throughout the model and in this case we're not going to make them translucent at all, we're going to make them solid. So here you can see, as you're plotting the Iso-Surfaces, the user can get a very good idea for how the model is behaving and where the energy drops are in the model. As well, with the backend, one thing you can do is do Value Blanking. If you want to see the interior parts of the model you can turn on **Plot>Value Blanking** and pick a **Constraint** (in this case we're going to cut the model mid-way, where y is less than 75). What that does is blank out a section of the model where all values of the y coordinate are less than 75. You can see if you twist the model, those values are all blanked out and are no longer plotted. You can get a good idea for how the interior of the model is displayed. These are all translucent. There are two general views: you can look inside the model or you can turn off the translucency and plot contours over the surface of the model. In that case you can get a slightly different visualization of the model. Here we're going to turn off **Value Blanking** and just turn off the **Translucency** and you get a very good idea for how the contours of the model are behaving. There are many more features associated with the Acumesh software which will be covered in a



future presentation. This covers the basic ideas of how to visualize basic results of any numerical model.

*<End of transcript>*

For more information on SVOOffice 2009 products, feel free to contact us directly at (306) 477-3324, or download our student version from [www.soilvision.com](http://www.soilvision.com) and try it out for yourself!