SVAIRFLOW™

2D / 3D AirFlow Modeling Software

Tutorial Manual

Written by:
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D., P.Eng.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVDYNAMIC ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

1 SVAirFlow Tutorial Manual

SVAIRFLOW™

2D / 3D Airflow Modeling Software

Tutorial Manual

Written by:
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own models.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE ™ is a trademark of SoilVision Systems, Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVDYNAMIC ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

1.1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVAIRFLOW software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

The Tutorial Manual serves as a guide by: i) assisting the user with the input of data necessary to solve the boundary value problem, ii) explaining the relevance of the solution from an engineering standpoint, and iii) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVAIRFLOW.

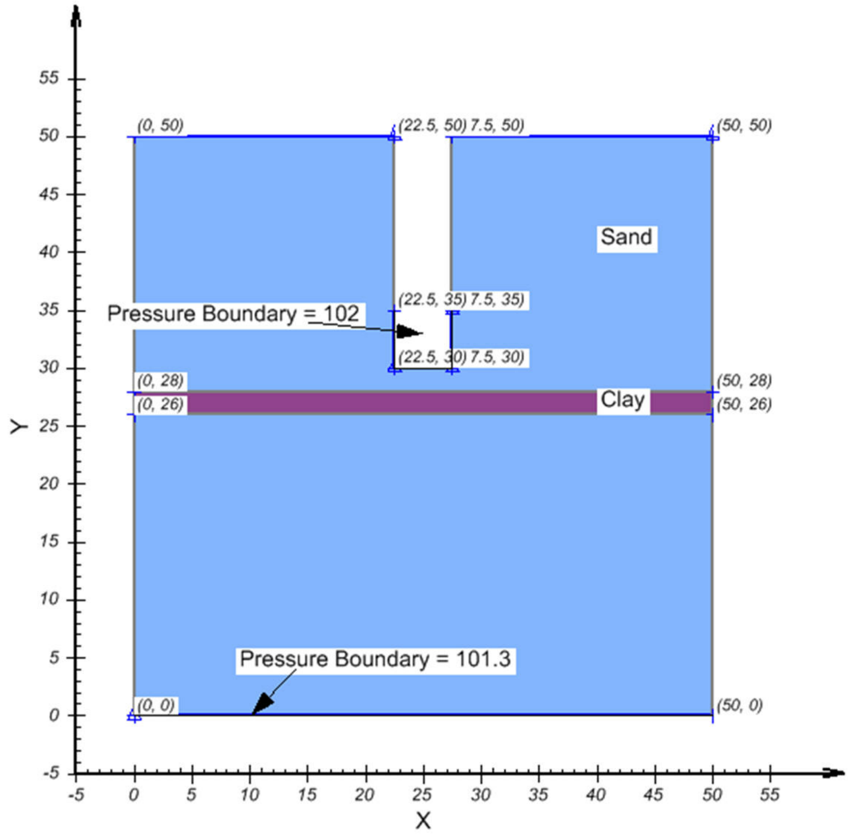
It should be noted that some models presented in this manual can be run with the free STUDENT authorization of the software. Other models require a purchased LITE, STANDARD, or PROFESSIONAL authorization level to run through the tutorial. The authorization level required for each model is specified at the start of the model.

1.2 A Two-Dimensional Example Model

The following example will introduce some of the features included in SVAIRFLOW and will set up a model of a simple air injection well. The purpose of this model is to determine the effects of a clay layer on the air pressure contours around an injection well. The well dimensions have been exaggerated for simplicity and viewing purposes. The model dimensions and material properties are provided below.

Project:	BoreHolePumping
Model:	SingleWellwClay
Minimum authorization required:	STUDENT

Model Description



Material Properties

Sand: air conductivity, k_a = 2.18E-04 m/s
Clay: air conductivity, k_a = 2.18E-05 m/s

1.2.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify model output
g. Run model

h. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the SVOFFICE Manager dialog.
2. Select *"ALL"* under the Applications combo box and *"ALL"* for the Model Origin combo box.
3. Create a new project called *UserTutorial* by pressing the *New* button next to the list of projects. If the project is already present select it.
4. Create a new model called *User_SingleWellwClay* by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created *UserTutorial* project.
5. Select the following:
    - *SVFlux* for Application
    - *2D* for System,
    - *Steady-State* for Type,
    - *Metric* for Units, and
    - *Seconds (s)* for Time Units.

Before entering any model geometry it is best to set the World Coordinate System to ensure that the model will fit in the drawing space. The user may access the *World Coordinates System* dialog by selecting the *World Coordinates System* tab on the Create New Model dialog.

1. Access the *World Coordinate System* tab on the Create New Model dialog.
2. Enter the *World Coordinates System* coordinates shown below into the dialog.

    x-minimum = -5
    y-minimum = -5
    x-maximum = 55
    y-maximum = 55

3. Click *OK* to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The SandyRegion region of the model has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum of 0.5.

1. The *View Options* dialog should automatically appear.
2. Enter *0.5* for both the horizontal and vertical spacing.
3. Click *OK* to close the dialog.

b. Enter Geometry

This model will be divided into two regions, which are named *SandyRegion* and *ClayRegion*. Each region will have separate material properties. To add the necessary region node points follow these steps:

1. Open the regions dialog by selecting *Model > Geometry > Regions* from the menu.
2. Change the first region name from R1 to *SandyRegion*. To do this, highlight the name and type new text.
3. Press the *New* button to add a second region.
4. Change the name of the second region to *ClayRegion*.
5. Click *OK* to close the dialog.

The shapes that define each material region will now be created. Note that when drawing geometry shapes, the region that is current in the region selector is the region the geometry will be added to. The Region Selector is at the top of the workspace.

SandyRegion

X	Y	Boundary Condition
0	0	BN_R1_758278157
50	0	
50	50	
27.5	50	
27.5	35	WellScreen
27.5	30	
22.5	30	
22.5	35	End
22.5	50	
0	50	

ClayRegion

X	Y	Boundary Condition
0	28	BN_ClayRegion_707530074
50	28	
50	26	
0	26	

Geometry in the SVOFFICE software may be i) typed in manually, ii) cut and pasted from Excel, iii) imported from AutoCAD, or iv) drawn graphically using the CAD window. In this example we will provide instruments for drawing the geometry using the CAD window. Prior to drawing geometry shapes it is good to ensure that snappy is turned on by clicking the "Snap On" at the bottom of the drawing space.

- **Define the SandyRegion**

1. Ensure the SandyRegion region is current in the region selector drop-down.

2. Select *Draw > Model Geometry > Region Polygon* from the menu or press the toolbar button. 
3. Move the cursor near (0,0) in the drawing space and left-click the button to initiate drawing the shape. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space.
4. Now move the cursor to subsequent points on the region, clicking once on each

region point.

5. For the last point, move the cursor near the point (0,50) and double-click on the *point to finish the shape*. A line is now drawn from (22.5,50) to (0,50) and the shape is automatically finished by SVAIRFLOW by drawing a line from (0,50) back to the start point, (0,0).

If the SandyRegion geometry has been entered correctly the shape should look like the model diagram at the beginning of this tutorial.

NOTE:

If a mistake was made entering the coordinate points for a shape, edit the shape using the Region Properties dialog (menu item *Model > Geometry > Region Properties*). Please also see the following sections in the user's manual to undo changes, move a point, move multiple points, delete points, or subdivide a line segment.

- **Define the ClayRegion**

6. Ensure that "ClayRegion" is current in the region selector.

7. Select *Draw > Model Geometry > Region Polygon* from the menu.
8. Move the cursor near (0,28) in the drawing space and left click.
9. To select the point as part of the shape left click on the *point*.
10. Now move the cursor near (50,28) and left click on the *point*. A line is now drawn from (0,28) to (50,28).
11. Repeat the process for the rest of the points in the region.
12. For the final point, move the cursor near the *point (0,26)* and double-click on the *point to finish the shape*. A line is now drawn from (50,26) to (0,26) and the shape is automatically finished by SVAIRFLOW by drawing a line from (0,26) back to the start point, (0,0).

NOTE:

At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on *"OSNAP"* in the status bar to alleviate this problem.

c. Specify Initial Conditions

A temperature of 15°C is required for the specification of initial conditions. Initial conditions can be specified through the following steps:

1. Select *Model > Initial Conditions > Settings* from the menu.
2. Move to the *Temperature* tab.
3. Select the *Constant/Expression* Temperature Option.
4. Enter a temperature value of *15*.
5. Click *OK* to close the Initial Conditions dialog.

d. Specify Boundary Conditions

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user cannot define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. An atmospheric pressure of 101.2 *kPa* is applied at the ground surface and a pressure of 101.3 kPa is applied at the base of the model to provide a small gradient. The injection well pressure is 102 *kPa*.

X	Y	Boundary Condition
0	0	Pressure Expression = 101.3 kPa
50	0	Zero Flux
50	50	Pressure Expression = 101.2 kPa
27.5	50	Zero Flux
27.5	35	Pressure Expression = 102 kPa
27.5	30	Continue
22.5	30	Continue
22.5	35	Zero Flux
22.5	50	Pressure Expression = 101.2 kPa
0	50	Zero Flux

The steps for specifying the boundary conditions are thus:

- **SandyRegion**

1. Select the *SandyRegion* region in the drawing space.
2. From the menu select *Model > Boundaries > Boundary Conditions*. The boundary conditions dialog will open. By default the first boundary segment will be given a Zero Flux condition. The user may also press the  toolbar button.
3. Select *Pressure Expression* from the Boundary Condition drop-down.
4. Select the *point* (0,0).
5. Enter 101.3 in the *Expression* field.
6. Select the point (50,0) from the list.
7. Select the Zero Flux condition from the drop-down.
8. Apply the remaining boundary conditions referring to the list above.
9. Click the *OK* button to close the dialog.

NOTE:

The Pressure Expression boundary condition for the point (27.5,50) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. In this case the line segments from (27.5,50) to (22.5,35) are all given a zero flux boundary condition.

- **ClayRegion**
  By default a Zero Flux boundary condition is set for the ClayRegion region and this boundary condition is appropriate so no specifications are required.

e. Apply Material Properties

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. A sand is defined for the majority of the model and a clay layer extends horizontally through the middle. This section will provide instructions on creating the sand material. Repeat the process to add the clay material.

1. Open the Materials dialog by selecting *Model > Materials > Manager* from the menu.
2. Click the *New* button to create a material.
3. Enter *Sand* for the material name in the dialog which appears.
4. Press *OK* to close the dialog. The Material Properties dialog will open automatically.

NOTE:

When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

5. Move to the *Conductivity* tab and select *Constant/Expression* from the drop-down.
6. Enter the k_a value of 2.18E-04 m/s.

7. Click *OK* to close the Material Properties dialog.
8. Repeat these steps to create the clay material; refer to the data provided in the A Two Dimensional Example Model section at the beginning of this tutorial.
9. Press *OK* to close the Materials Manager dialog.
10. Open the *Regions* dialog by selecting Model > Geometry > Regions from the menu.
11. Select the *Sand* Material for the SandySoil using the material drop-down.
12. Select the *Clay* Material for the ClayLayer using the material drop-down.
13. Press the *OK* button to accept the changes made and close the Regions dialog.

f. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

The plot manager dialog is first opened to display appropriate solver graphs. Boundary Flux specifications are used to report the rate of flow across a boundary for a steady state analysis and the rate and volume of flow moving across a boundary in a transient analysis. Boundary Names must be assigned using the Region Properties dialog and then Reports selected using the Boundary Flux dialog.

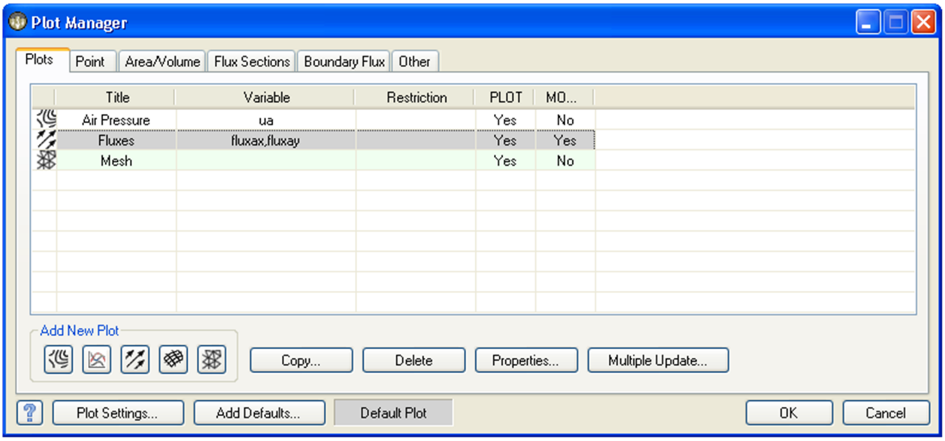
1. Click on the *SandyRegion* region shape in the workspace with the mouse.
2. Select *Model > Boundaries > Boundary Conditions* from the menu to open the Boundary Conditions dialog. By default the point (0,0) has a Boundary Name defined.
3. Select the point (27.5,35).
4. Enter the boundary name WellScreen.
5. Select point (22.5,35) from the list.
6. Enter *End* as the boundary name.
7. Press *OK* to close the dialog.

Once the Boundary Names have been defined, the plots can be defined.

1. Select *Model > Reporting > Plot Manager* from the menu to open the *Plot Manager* dialog.
2. Select the *Boundary Flux* tab and press the *Report* button.
3. Select *WellScreen* as the boundary and enter *WellScreen* as the title.
4. Select the *Output Options* tab and click the checkbox beside PLOT.

5. Press *OK* to return to the Plot Manager dialog.
6. Press the *History Plot* button on the Boundary Flux tab.
7. Select *WellScreen* as the boundary, enter *BF: WellScreen - X Component of Flow* as the title, and select *X Component of Flow* as the variable.
8. Select the *Output Options* tab and ensure both checkboxes are unchecked.
9. Press *OK* to return to the Plot Manager dialog.
10. Press the *History Plot* button on the Boundary Flux tab.
11. Select *WellScreen* as the boundary, enter *BF: WellScreen - Y Component of Flow* as the title, and select Y *Component of Flow* as the variable.
12. Select the *Output Options* tab and ensure both checkboxes are unchecked.
13. Press *OK* to return to the Plot Manager dialog.
14. Press the *History Plot* button on the Boundary Flux tab.
15. Select *WellScreen* as the boundary, enter *BF: WellScreen - Normal Flow* as the title, and select *Normal Flow* as the variable.
16. Select the *Output Options* tab and ensure both checkboxes are unchecked.
17. Press *OK* to return to the Plot Manager dialog.
18. Click *OK* to close the *Plot Manager* dialog and return to the workspace.

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example model including a plot of the solution mesh, air pressure contours, and flux vectors. These plots may be automatically generated by pressing the Default Plots button.



OUTPUT MANAGER

1. Open the Output Manager dialog by selecting *Model > Reporting > Output Manager* from the menu.
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *ACUMESH* button to begin adding the output file. The Output File Properties dialog will open.

3. Click *OK* to close the dialog and add the output file to the list.
4. Click *OK* to close the Output Manager and return to the workspace.

g. Run Model

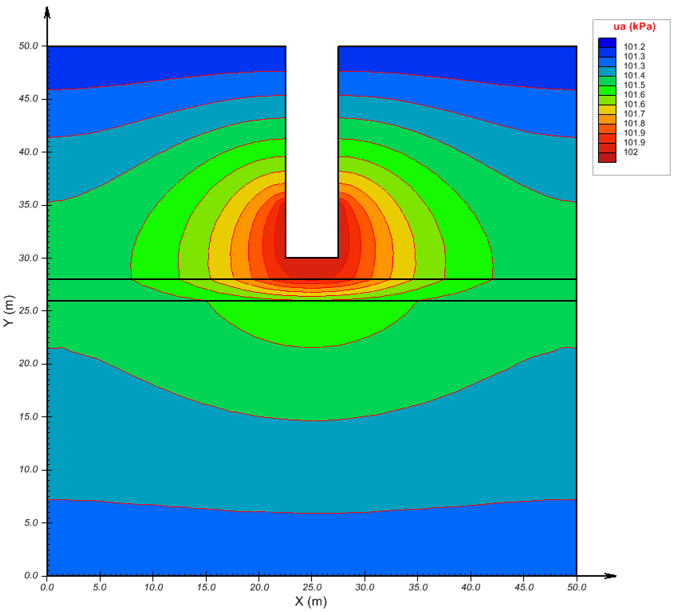
The next step is to analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

h. Visualize Results

The visual results for the current model may be examined by selecting the *View* > *ACUMESH* menu option or clicking on the ACUMESH button  on the processes toolbar.

1.2.2 Results & Discussion

A contour plot of the completed model results can be seen below. All outputs previously specified can now be visualized using ACUMESH. It can be seen from the following results that the clay layer inhibits the flow of air. Lateral flow in the air is therefore unlikely in this instance.



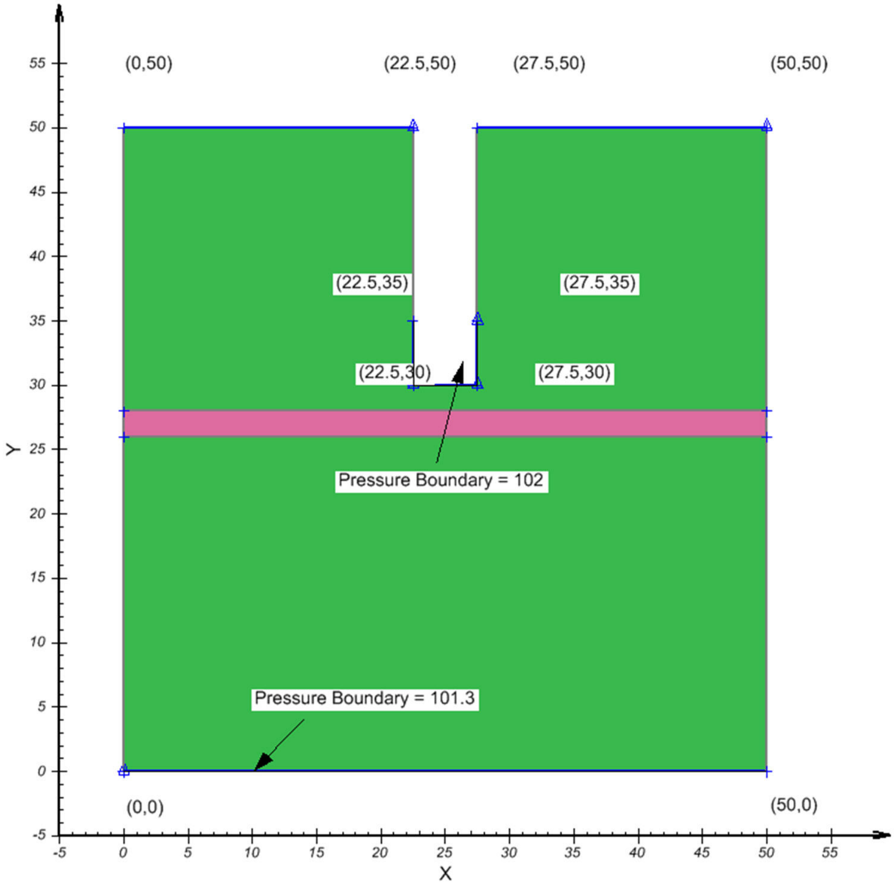
1.3 2D Stochastic Example

A single run of a numerical model is often not sufficient. SVAIRFLOW implements the ability to incorporate statistical uncertainty into the input of material properties. This tutorial section guides the user through the setup and execution of a model implementing the stochastic or statistical features available in SVAIRFLOW.

The air injection well example presented in Section 1 is extended to incorporate a variation in the air conductivity, k_a of the clay layer. Instead of specifying a single value for this parameter an incremental analysis will be used to generate 50 runs of the same model varying the "k_a" parameter over 6 orders of magnitude.

Why do we want to do this? In this example we want to determine the influence of the k_a on the difference in pressure distributions around the injection well. The sensitivity of the k_a value may be significant in determining injection rates.

Project: BoreHolePumping
Model: SingleWellwClaySto



1.3.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

1. Create model
2. Enter geometry
3. Specify boundary conditions
4. Apply material properties

5. Specify model output
6. Run model
7. Visualize results

1. Create Model

The first step in creating the stochastic model is to save a copy of the example model created in section 1. This is accomplished through the following steps:

1. Open the *Tutorial* > *SingleWellwClay* model,
2. Select *File* > *Save Model As…*
3. Type the name *SingleWellwClaySto* and click *OK*.

Now a new model has been created and loaded into the workspace that will be modified to include stochastic analysis.

The second step is to describe the number of stages/runs that will be used in the analysis. Stages are controlled in SVAIRFLOW by opening the *Model* > *FEM Options* dialog. The *STAGES* field should be set to *50* under the Stage Controls area of the dialog. Click *Yes* for the warning message and then close the FEM Options dialog.

2. Enter Geometry

The geometry has already been specified for this model.

3. Specify Boundary Conditions

The boundary conditions have already been specified for this model.

4. Apply Material Properties

The next step that must occur is that the k_a value will be staged. In this example the parameter will be set to have a mean value of 5 and a standard deviation of 2. This staging is accomplished through the following process:

1. Select *Model* > *Materials* > *Manager…* to open the list of current materials,
2. Select the *"ClayRegion"* material and click the *"Properties…"* button,
3. Click the *Stage Parameters* button located in the lower left of the dialog,
4. Select the *k* parameter and click the *Include* button,
5. Click the *Stage Values* button,
6. Click the *Increment Stage Values* button,
7. Enter a start of 0.0000001 and an increment of 0.001 and click the *OK* button,
8. A list of 50 generated values will now be displayed in the Stage Values list box.
9. Press *OK* on all remaining dialogs to close them.

The parameter has now been staged.

5. Specify Model Output

Model Outputs have already been defined for this model.

6. Run Model

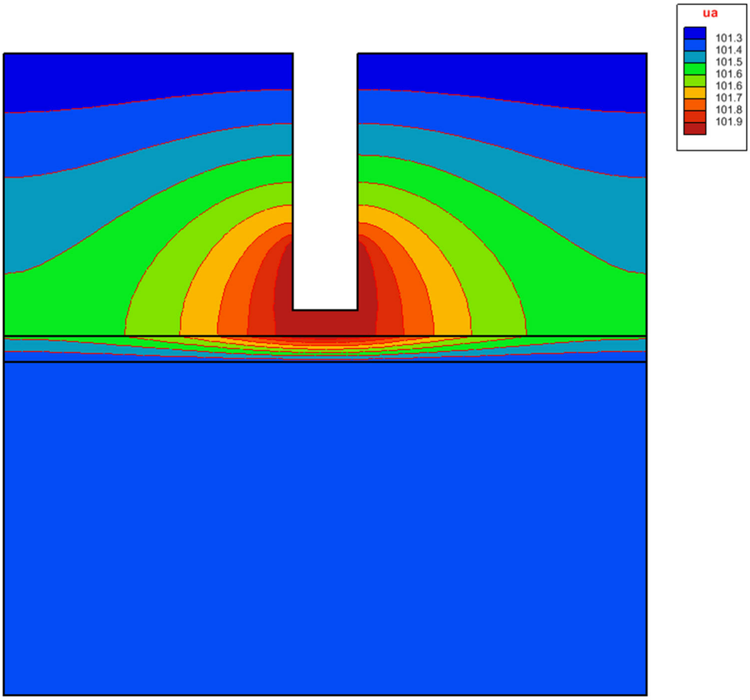
Selecting *Solve > Analyze* from the menu will initiate model solution. Attention should be directed to the stage display in the finite element solver. Once a particular stage is complete the next stage will automatically be initiated. The differential flows through the sandy material and clay layer may be seen in the gradient plot as the air conductivity in the clay layer varies.

7. Visualize Results

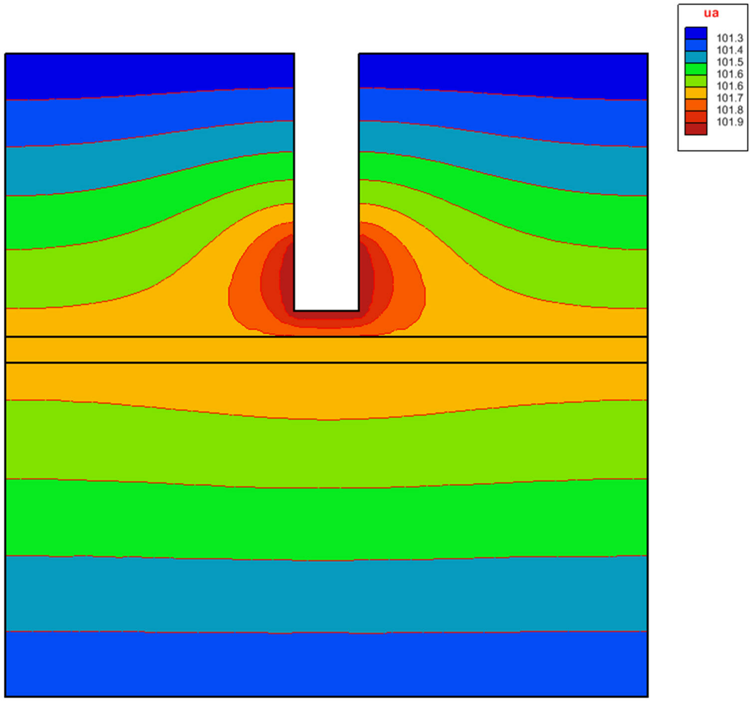
The visual results for the current model may be examined by selecting the *View > ACUMESH* menu option.

1.3.2 Results & Discussion

Contour plots of the completed model at stages 1 and 20 can be seen below. All outputs previously specified can now be visualized using ACUMESH.



Contour Plot at Stage 1

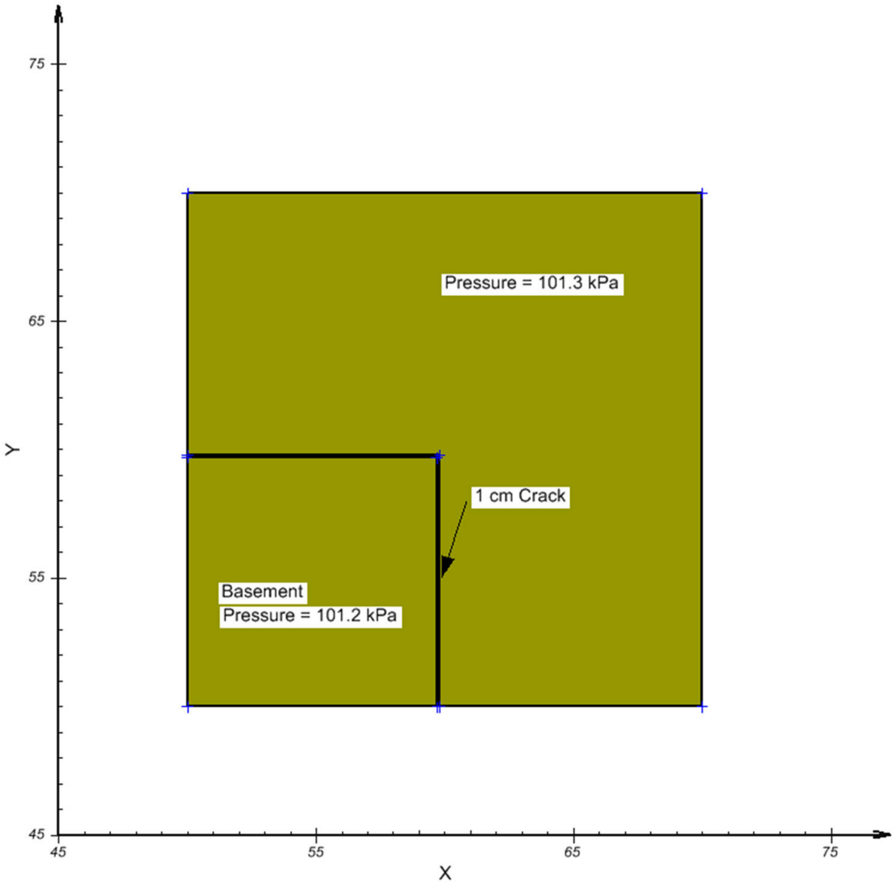


Contour Plot at Stage 20

1.4 A Three-Dimensional Example Model

The following example will introduce you to the three dimensional SVAIRFLOW modeling environment. The purpose of this model is to calculate the quantity of air-flow into the basement. A 1 cm crack exists between the floor slab and the basement walls. The intent of the current model is to calculate the volume of contaminated air which will enter the basement in these specific conditions.

Project: Foundations
Model: Floorleak
Minimum authorization required: FULL



1.4.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a) Create model
b) Enter geometry
c) Specify initial conditions
d) Specify boundary conditions
e) Apply material properties
f) Specify model output
g) Run model
h) Visualize results

a. Create Model

Since FULL authorization is required for this tutorial, the user must perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File > Authorization* dialog on the SVOFFICE Manager.
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the SVOFFICE Manager dialog.
2. Select the project called *UserTutorial* by pressing the *New* button next to the list of projects.
3. Create a new model called *UserFloorLeak* by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created *UserTutorial* project.

The first step in defining the model is to specify the settings that will be used for the model. To open the Settings dialog select *Model > Settings* in the workspace menu. Select the following settings:

1. Select *3D* for System,
2. *Steady-State* for Type,
3. *Metric* for Units, and
4. *Seconds* for Time Units.

Before entering any model geometry it is best to set the World Coordinate System to ensure that the model will fit in the drawing space. The user may access the *World Coordinates System* dialog by selecting the *World Coordinates System* tab on the Create New Model dialog.

1. Access the *World Coordinate System* tab on the Create New Model dialog.
2. Enter the *World Coordinates System* coordinates shown below into the dialog.

   x-minimum = 45
   y-minimum = 45
   x-maximum = 75
   y-maximum = 75

3. Click *OK* to close the dialog.

b. Enter Geometry

A region in SVAIRFLOW is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVAIRFLOW CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, text, and line art can be defined on any given region.

This model will be defined by three regions, which are named Outer, Basement, and Crack. To add the necessary regions follow these steps:

1. Open the regions dialog by selecting *Model > Geometry > Regions* from the menu.
2. Change the first region name from *R1* to *Outer*. To do this, highlight the name and type new text.
3. Press the *New* button to add a second region.
4. Change the name of the second region to *Basement*.
5. Press the *New* button to add a third region.
6. Change the name of the third region to *Crack*.
7. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes the region that is current in the region selector is the region the geometry will be added to. The Region Selector is at the top of the workspace.

Outer			**Basement**			**Crack**	
X	**Y**		**X**	**Y**		**X**	**Y**
50	50		50	50		50	50
70	50		59.8	50		59.7	50
70	70		59.8	59.8		59.7	59.7
50	70		50	59.8		50	59.7

- **Define the Outer region**

1. Ensure the *Outer* region is current in the region selector.
2. Select *Draw > Model Geometry > Polygon Region* from the menu.
3. Move the cursor near (50,50) in the drawing space.
4. When the cursor is near the point, click the left mouse button. The cursor should automatically snap to the point (50,50) as long as the SNAP and GRID options in the status bar are both bold.
5. Now move the cursor to subsequent region points and repeat the left-click procedure.
6. For the last point (50,70), double-click on the point to finish the shape. A line is now drawn from (70,70) to (50,70) and the shape is automatically finished by SVAIRFLOW by drawing a line from (50,70) back to the start point, (50,50).

NOTE:
Select a shape with the mouse and select *Edit > Delete* from the menu if a mistake was made entering the coordinate points for a shape. This will remove the entire shape from the region. To edit the shape use the Region Properties dialog.

- **Define the Basement**
  In the instructions for defining the Outer shape the command line was used. To draw the Basement region the instructions below explain the use of the drawing tool to create the Basement shape.

7. Select *Model > Geometry > Regions* from the menu.
8. Select the Basement region and click the *Properties* button.
9. Click the *New Polygon* button.
10. Enter the region points as shown in the above table.
11. Click *OK* to save the region geometry and close the Region Properties dialog.

- **Define the Crack**
  Follow the above method to define the Crack shape, referring to the table of points above.

  After all the region geometry has been entered it will appear like the diagram at the beginning of this tutorial.

This model consists of three surfaces defined by constant elevations. By default every model initially has two surfaces.

- **Define Surface 1**
  This surface will be defined by providing a constant elevation.

1. Select *Surface 1* in the *Surface Selector* located at the top of the workspace.
2. Select *Model > Geometry > Surface Properties* in the menu to open the *Surface Properties* dialog.
3. For the Surface Definition option, select *Constant*.
4. Enter a *Surface Constant* of *0.*
5. Click *OK* to close the dialog.

- **Define Surface 2**
  This surface will be defined by providing a constant elevation.

6. Select *Surface 2* in the *Surface Selector* located at the top of the workspace.
7. Select *Model > Geometry > Surface Properties* in the menu to open the Surface Properties dialog.
8. For the Surface Definition option, select *Constant* .
9. Enter a *Surface Constant* of 4.
10. Click *OK* to close the dialog.

- **Insert and Define Surface 3**
  Surface 3 is not present by default and must be created using the Insert Surface dialog.

  1. Open the Surfaces dialog by selecting *Model > Geometry > Surfaces* from the menu. Press the *New* button to add surfaces.
  2. 1 is selected in the Number of New Surfaces dialog and Place New Surfaces *At The Top* is the default selected, so press *OK* to add Surface 3 and close the dialog.
  3. Click *OK* to close the Insert Surface dialog.
  4. Select *Surface 3* in the Surfaces dialog and click the *Properties* button.
  5. For the Surface Definition option, select *Constant*.
  6. Enter a *Surface Constant of 5.5.*
  7. Click *OK* to close the dialog.

c. Specify Initial Conditions

A temperature of 20°C is required.

1. Select *Model > Initial Conditions > Settings* from the menu.
2. Move to the *Temperature* tab.
3. Select the *Constant/Expression* Temperature Option.
4. Enter a temperature value of *20*.
5. Click *OK* to close the Settings dialog.

d. Specify Boundary Conditions

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions, surfaces, and the materials have been successfully defined, the next step is to specify the boundary conditions on the region shapes. The ground surface will be set at a pressure of 101.3 kPa while the basement will be set at a pressure of 101.2 kPa. The steps for specifying the boundary conditions include:

1. Select the *Outer region* in the drawing space.
2. Select *Surface 3* in the surface selector.
3. From the menu select *Model > Boundaries > Boundary Conditions*. The boundary conditions dialog will open and display the boundary conditions for Surface 3.

4. Move to the *Surface Boundary Conditions* tab.
5. From the *Boundary Condition* drop-down select a *Pressure Expression* boundary condition. This will cause the Expression box to be enabled.
6. In the Expression box enter a pressure of *101.3*.
7. Click the *OK* button to save the boundary condition to the list.

Now, to set the Crack regions Surface 2 Boundary Condition to 101.2 kPa:

1. Select the *Crack* region in the drawing space.
2. Select *Surface 2* from the surface drop-down.
3. From the menu select *Model > Boundaries > Boundary Conditions*.
4. Click the *Surface Boundary Conditions* tab.
5. Select the *Pressure Expression* boundary condition and enter a value of 101.2 in the expression box.
6. Close the dialog using the *OK* button.

NOTE:
The remaining Surfaces are by default set to the None boundary condition, which is treated as a Zero Flux condition. The remaining Segments are by default set to the No *BC* boundary condition, which also is treated as a Zero Flux condition.

e. Apply Material Properties

The next step in defining the model is to enter the material property for the material that will be used in the model. It will be defined for all the regions.

1. Open the Materials Manager dialog by selecting *Model > Materials > Manager* from the menu.
2. Click the *New* button to create a material. Enter *Soil1* for the name.
3. The Material Properties dialog will open automatically; or, select the new material and click *Properties* to open the Material Properties dialog.
4. Move to the *Conductivity* tab.
5. Enter the k_a expression of 7E-08*(z/5.5) *m/s*. This expression will cause the air conductivity to vary with depth, *z.*
6. Press *OK* on the Materials Manager dialog to close both dialogs.

Each region will cut through all the layers in a model creating a separate “block” on each layer. Each block can be assigned a soil or be left as *void*. A void area is essentially air space. In this model all “blocks” will be assigned a material.

1. Select *Outer* in the Region Selector.
2. Select *Model > Materials > Region Materials* from the menu to open the Region Materials dialog.
3. Select the *Soil1* material from the drop-down for Layer 1.

4. Select the *Soil1* material from the drop-down for Layer 2.
5. Select *Basement* in the Region Selector using the right arrow at the top of the Region Materials dialog.
6. Select *Model > Materials > Region Materials* from the menu to open the Region Materials dialog.
7. Select the *Soil1* material from the drop-down for Layer 1.
8. Select *Crack* in the Region Selector.
9. Select *Model > Materials > Region Materials* from the menu to open the Region Materials dialog.
10. Select the *Soil1* material from the drop-down for Layer 1.
11. Close the dialog using the *OK* button.

f. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with AcuMesh software. Output is specified in the following two dialogs in the software:

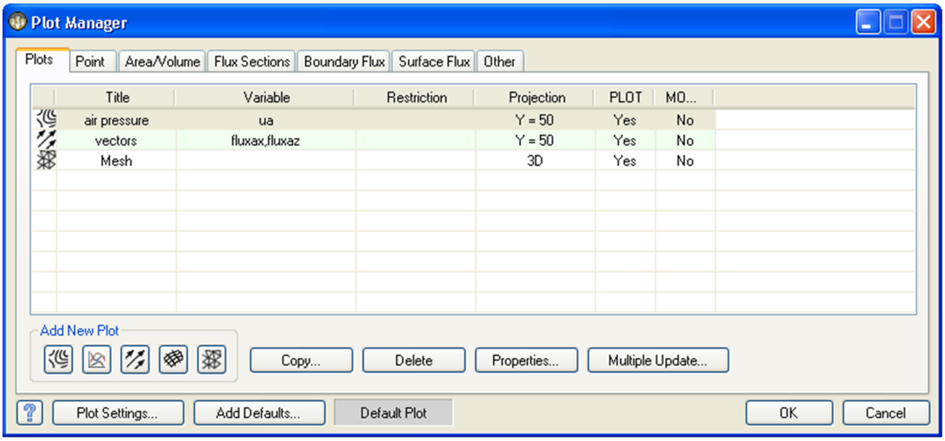
i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in AcuMesh or for inputting to other finite element packages.

PLOT MANAGER

The plot manager dialog is first opened to display appropriate solver graphs. There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example model including a plot of the pressure contours, solution mesh, and gradient vectors.

1. Open the Plot Manager dialog by selecting *Model > Reporting > Plot Manager* from the menu.
2. The toolbar at the bottom left of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open.
3. Enter the title *air pressure*.
4. Select *ua* as the variable to plot from the drop-down.
5. Select the *PLOT* output option.
6. Move to the *Projection* tab.
7. Select *Plane* Projection option.
8. Select *Y* from the Coordinate Direction drop-down.
9. Enter *50* in the Coordinate field. This will generate a 2D slice at y = 50m on which the air pressures will be plotted.
10. Click *OK* to close the dialog and add the plot to the list.
11. Repeat these steps 2 to 10 to create the remaining plots. Note that the Mesh plot does not require entry of a variable.

12. Click *OK* to close the Plot Manager and return to the workspace.



In order to find out the amount of air flowing into the basement, we must define a surface flux across the Surface 2 (the basement floor) and restrict it to the BasementIn region. Follow these steps to define this:

13. Open the Plot Manager dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu.
14. Select the *Surface Flux* tab and press the *Summary Plot* button from the Add New Plot area.
15. The Plot Properties - Surface Flux dialog will appear. Select *Surface 2* from the Surface drop-down.
16. Type in *Crack Flow* as the name of the Surface Flux and Restrict to Region *Basement.*

Additional plots may be defined by pressing the Default Plots button on the Plot Manager.

OUTPUT MANAGER

Two types of output files will be generated for this tutorial example model: a transfer file of air pressure, and a .dat file to transfer the results to ACUMESH.

1. Open the Output Manager dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu.
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *ACUMESH* button to begin adding the output file. The Output File Properties dialog will open.
3. Enter the title *Basement2*.
4. Click *OK* to close the dialog and add the output file to the list.
5. Press the Transfer File button to create a new transfer file.
6. Enter the title *uaTransfer*.
7. Select the variable *ua* in the available variables list.

8. Press the *Single Right Arrow* button.
9. Click *OK* to close the dialog and add the output file to the list.
10. Click *OK* to close the Output Manager and return to the workspace.

g. Run Model

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the SVAIRFLOW solver. The solver will automatically begin solving the model.

When the Regrid Limit message appears click *No* and the solver will begin generating the plots.

h. Visualize Results

Once you have analyzed the model, the output plots can be visualized using ACUMESH. In order to view plots in ACUMESH, select *Window > ACUMESH* from the menu.

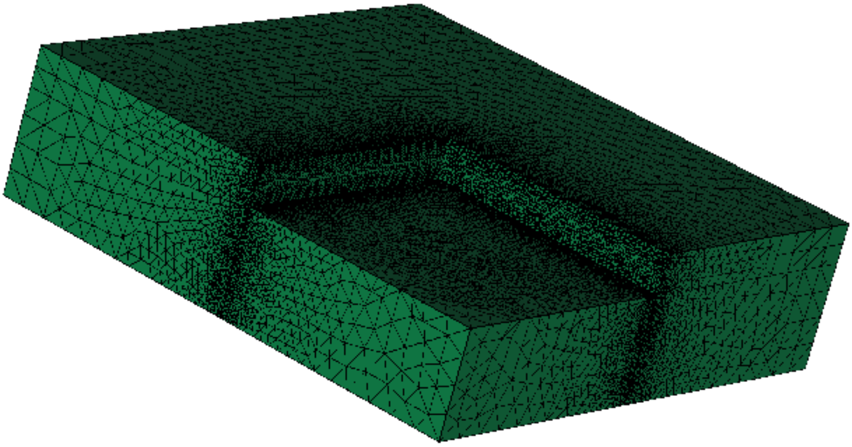
ACUMESH will prompt you to select the Output file to view. Choose *ACUMESHOut.dat* and click *OK* to open this file.

Plots can visualized by selecting the desired process under *Plots* in the menu.

1.4.2 Results & Discussion

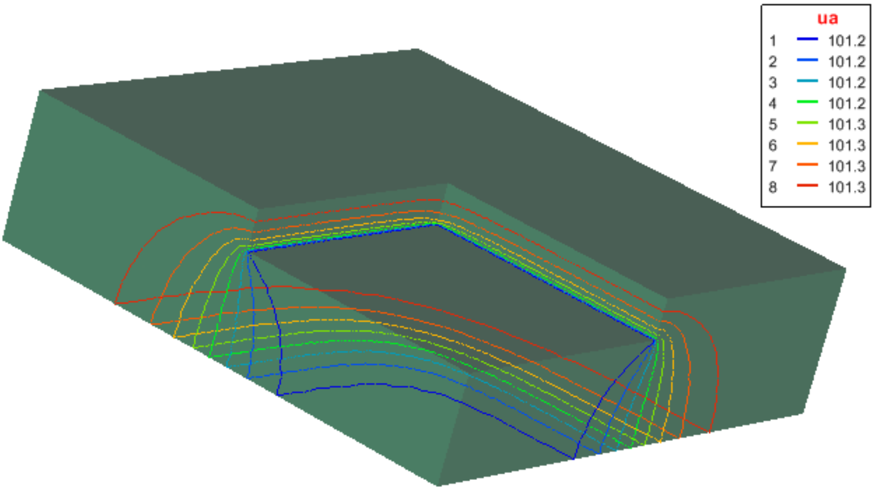
After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVAIRFLOW solver. These plots can also be examined in details using ACUMESH. This section will give a brief analysis for each plot that was generated.

- **Solution Mesh**



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas.

- **Pressure Contours**



The plot indicates a pressure gradient causing flow of air into the basement. The results of the flux section indicate the steady-state flow of air of 4.80e-7 m^3/second, equivalent to 0.0415 m^3/day is entering the basement.

1.5 References

Baehr, A.E., and Hult, M.F. 1991. Evaluation of Unsaturated Zone Air Permeability Through Pneumatic Tests. Water Resources Research, 27(10): 2605-2617.

Ba-Te (2004). "Flow of Air-phase in Soils and Its Application in Emergent Stabilization of Soil Slopes," Hong Kong University of Science and Technology, Hong Kong.

Fredlund, D. G., and Rahardjo, H. 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, New York

FlexPDE 3.x Reference Manual, 1999. PDE Solutions Inc. Antioch, CA 94509

FlexPDE 4.x Reference Manual, 2004. PDE Solutions Inc. Antioch, CA 94509

This page is left blank intentionally.