Open-Hole Multilateral Well

Introduction

Multilateral wells—those with multiple legs that branch off from a single conduit—can produce oil efficiently because the legs can tap multiple productive zones and navigate around impermeable ones. Unfortunately, drilling engineers must often mechanically stabilize multilateral wells with a liner or casing, which can cost millions of dollars. Leaving the wellbore uncased reduces construction costs, but it runs a relatively high risk of catastrophic failure both during installation and after pumping begins.

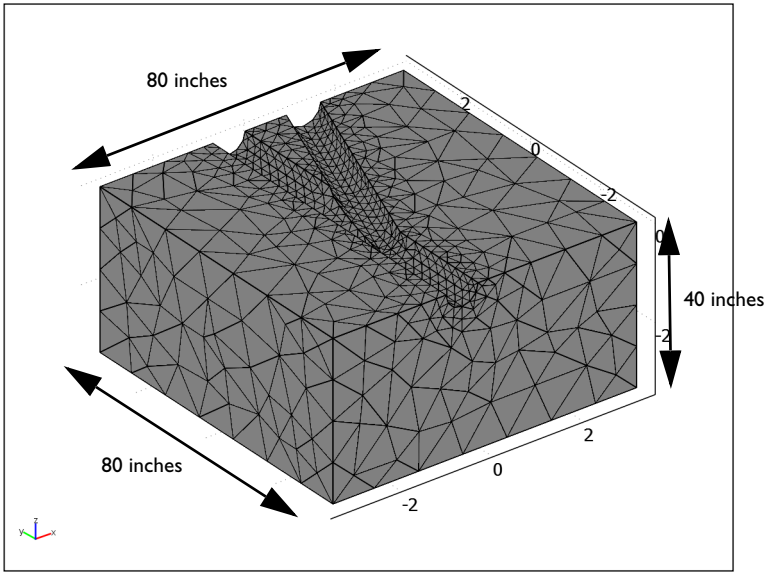


Figure 1: Geometry for an analysis of a horizontal open-hole multilateral well.

The poroelastic simulations estimate 3D compaction related to pumping by taking subsurface fluid flow with Darcy's law and coupling it to structural displacements with a stress-strain analysis. This model focuses on elastic displacements brought on by changing fluid pressures when pumping begins. Related analyses for elasto-plastic deformations are straightforward using material laws automated in the Structural Mechanics Module.

Model Definition: Flow and Deformation Simulation

The modeled geometry (Figure 1) is the lower half of a branching junction, a segment from a larger well network. The junction lies roughly 25 feet from the start of the well. The entire well network extends much further, perhaps hundreds of feet. The well is 8.5 inches in diameter and sits in a cube 80 inches on each side. Pumps move fluid from the reservoir into the well. Fluid exits the geometry only through the well. The displacement at the reservoir edge is constrained. The walls of the well, however, deform freely. The goal is to solve for the change in fluid pressure, stress, strain, and displacement that the pumping causes rather than their absolute values.

FLUID FLOW

To describe fluid flow, you insert the Darcy velocity into an equation of continuity

$$\nabla \cdot \left[-\frac{\kappa}{\mu} \nabla p \right] = 0$$

where κ is the permeability, μ is the dynamic viscosity, and p equals the pressure of the oil in the pore space.

For the flow boundaries, you already know the change in fluid pressure from the well to the reservoir edge. The planar surface adjacent to the well (between the upper and lower blocks) is a symmetry boundary. Because the well is the only exit for the fluid, there is no flow to or from connecting well segments. In summary,

$$\begin{array}{lll} p = p_{\mathrm{r}} & \partial\Omega & \text{reservoir} \\ \mathbf{n} \cdot \left(-\frac{\kappa}{\mu} \nabla p \right) = 0 & \partial\Omega & \text{symmetry face} \\ \mathbf{n} \cdot \left(-\frac{\kappa}{\mu} \nabla p \right) = 0 & \partial\Omega & \text{connecting segments} \\ p = p_{\mathrm{w}} & \partial\Omega & \text{well} \end{array}$$

where $\mathbf{n}$ is the normal vector to the boundary.

SOLID DEFORMATION

The system of equations that describes the solid deformation is

$$-\nabla \cdot \sigma = \mathbf{F}$$

where σ denotes the stress tensor, and the directional components of the gradient in fluid pressure, p, make up a vector forces, $\mathbf{F}$. In this model, the Biot-Willis coefficient

 ©2012 COMSOL

is equal to one. The stress-strain relationship for linear materials relates the strain tensor ε and the stress tensor through the elasticity matrix D, which is a function of Young's modulus, E, and Poisson's ratio, ν.

The elements in the strain tensor depend on the displacement vector $\mathbf{u}$, which has directional components u, v, and w:

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \qquad \varepsilon_{yz} = \frac{1}{2}\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

$$\varepsilon_z = \frac{\partial w}{\partial z} \qquad \varepsilon_{xz} = \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)$$

The tensors ε and σ are linearly related by $\sigma = D\varepsilon$.

For the boundary conditions, the model constrains movement at all external boundaries. The well opening is free to deform. In summary:

$u = v = w = 0$	$\partial\Omega$	reservoir
$w = 0$	$\partial\Omega$	symmetry face
$v = 0$	$\partial\Omega$	connecting segments
free	$\partial\Omega$	well

MODEL DATA

This model uses the coefficients and parameters listed in Table 1.

TABLE 1: MODEL DATA

VARIABLE	DESCRIPTION	VALUE
ρ_f	Fluid density	0.0361 lb/in^3
κ	Permeability	$1\cdot10^{-13}$ in^2
μ	Fluid dynamic viscosity	$1\cdot10^{-7}$ psi·s
E	Young's modulus	$0.43\cdot10^{6}$ psi
ν	Poisson's ratio	0.16
ρ_s	Solids density	0.0861 lb/in^3
p_r	Pressure in reservoir	122.45 psi
p_w	Pressure in well	0 psi

Results: Flow and Deformation Simulation

The following figures show results from simulations for coupled fluid flow and reservoir deformation following a poroelastic approach for the horizontal multilateral well reported in Ref. 1.

The isosurfaces in Figure 2 indicate the fluid pressure throughout the well's lower half. The streamlines show the fluid paths and velocities. Fluid pressure drops from the reservoir toward the well opening. The velocity typically increases toward the well but remains close to zero near the branching legs of the junction.

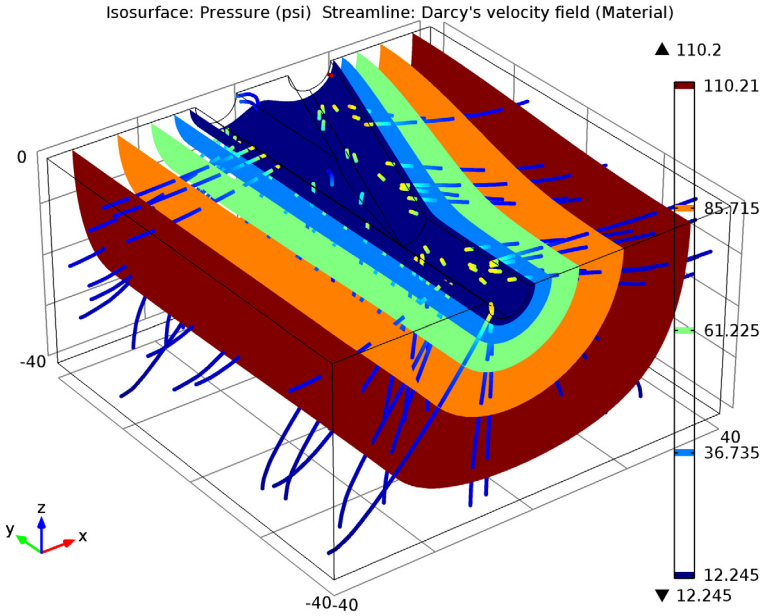


Figure 2: COMSOL Multiphysics poroelastic analysis of a multilateral well. Results are fluid pressure (the isosurfaces), pressure gradient (streamlines), and fluid velocities (streamline shading).

Results for elastic deformation appear in Figure 3. The surface shading denotes total displacement. The plot illustrates directional displacements by shifting the shading relative to an outline of the original geometry. For a clear view, the displacements are

 ©2012 COMSOL

exaggerated. The uncased surface yields slightly because the deformed shading fills in the hollows of the well. The largest displacements occur just above the split in the well.

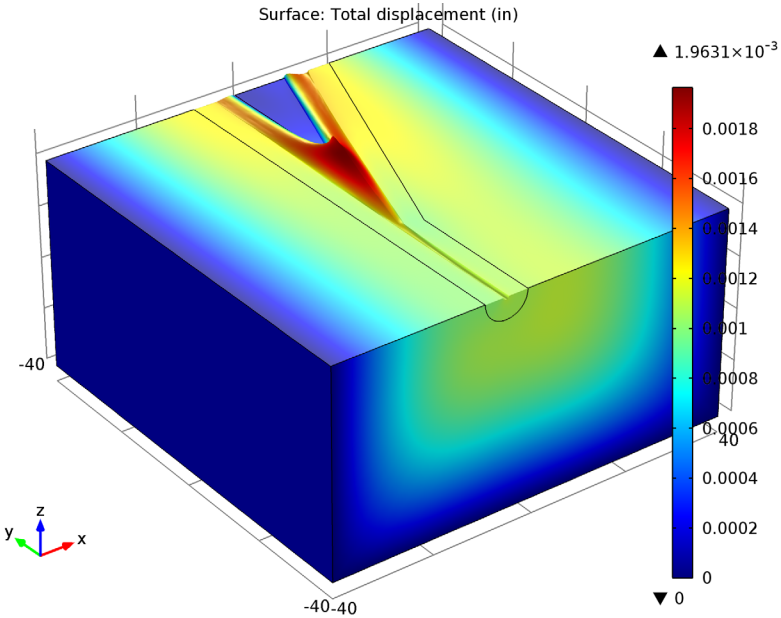


Figure 3: COMSOL Multiphysics estimates of displacement. Shading indicates the total displacement, and the geometry appears as lines. Even as the deformed shape shifts, those lines remain steady; the shaded image shows movement relative to the geometry outlines.

Failure Criterion

This model allows the evaluation of failures during postprocessing using results from the fluid-flow and solid-deformation simulations shown in the preceding figures. This discussion follows calculations that Ref. 1 uses to map calculations indicating where pumping could compact the reservoir enough (see Figure 3) that the well will fail. Refer to Ref. 1 to estimate the critical rock strength required to successfully emplace the well, and also to learn more about calibration to data.

The 3D Coulomb failure criterion in Ref. 1 relates rock failure, the three principal stresses (σ_1, σ_2, and σ_3), and the fluid pressures as follows:

$$\text{fail} = (\sigma_3 + p) - Q(\sigma_1 + p) + N\left(1 + \frac{(\sigma_2 - \sigma_1)}{(\sigma_3 - \sigma_1)}\right)$$
$$Q = \frac{1 + \sin\phi}{1 - \sin\phi}, \qquad N = \frac{2\cos\phi}{1 - \sin\phi}\text{So} \tag{1}$$

Here So is the Coulomb cohesion and ϕ is the Coulomb friction angle. When properly calibrated, fail = 0 indicates the onset of rock failure; fail < 0 denotes catastrophic failure; and fail > 0 predicts stability. Because this model solves for the pressure change brought on by pumping as well as the stresses, strains, and displacements that the pressure change triggers, it calculates the expression just given using the change in pressure than its absolute value.

Results: Failure Criterion

The values for the fail function appear in Figure 4. When the fail values become increasingly negative, the potential for failure is higher. As expected, the fail function estimates show the greatest potential for failure just above the split in the well.

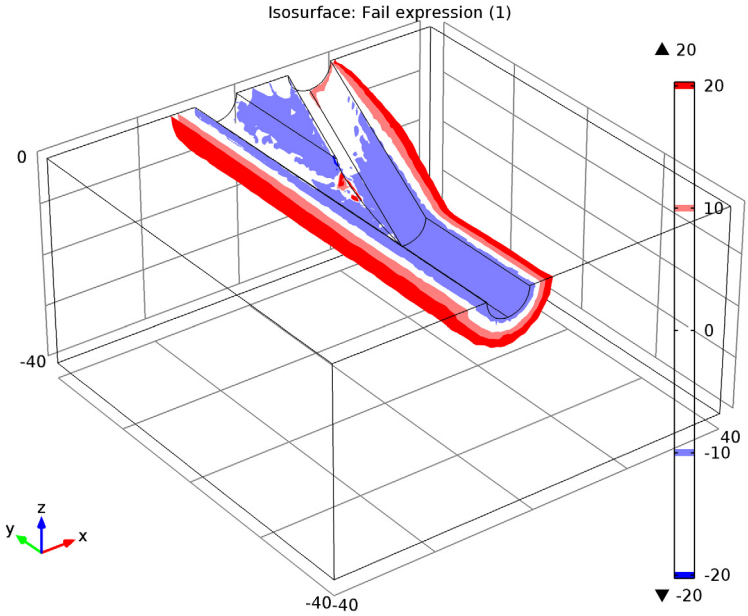


Figure 4: Values of the fail function calculated with results from a poroelastic model for the branching junction in an open-hole multilateral well. A negative value for the fail function denotes greater potential for failure.

 ©2012 COMSOL

Conclusions

This example couples fluid flow and solid deformation for a poroelastic analysis using easy-to-use physics interfaces from COMSOL Multiphysics. The analysis provides estimates of the pressure change brought on by pumping as well as the stresses, strains, and displacements that the pressure drop triggers. Combining the simulation results with a 3D Coulomb failure expression, maps vulnerability to mechanical failure from the pumping. The data and geometry for this model come from petroleum industry analyses by TerraTek (Ref. 1), which in turn use failure criteria to map the potential for failure during emplacement of the well in addition to the potential for failure when the well is pumped.

Reference

1. R. Suarez-Rivera, B.J. Begnaud, and W.J. Martin, "Numerical Analysis of Open-hole Multilateral Completions Minimizes the Risk of Costly Junction Failures," *Rio Oil & Gas Expo and Conference* (IBP096_04), 2004.

Model Library path: `Subsurface_Flow_Module/Flow_and_Solid_Deformation/multilateral_well`

Modeling Instructions

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click **Next**.
3. In the **Add physics** tree, select **Structural Mechanics>Poroelasticity (poro)**.
4. Click **Next**.
5. Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
6. Click **Finish**.

GEOMETRY 1

1. In the **Model Builder** window, under **Model 1** click **Geometry 1**.
2. In the **Geometry** settings window, locate the **Units** section.
3. From the **Length unit** list, choose **in**.

Import 1

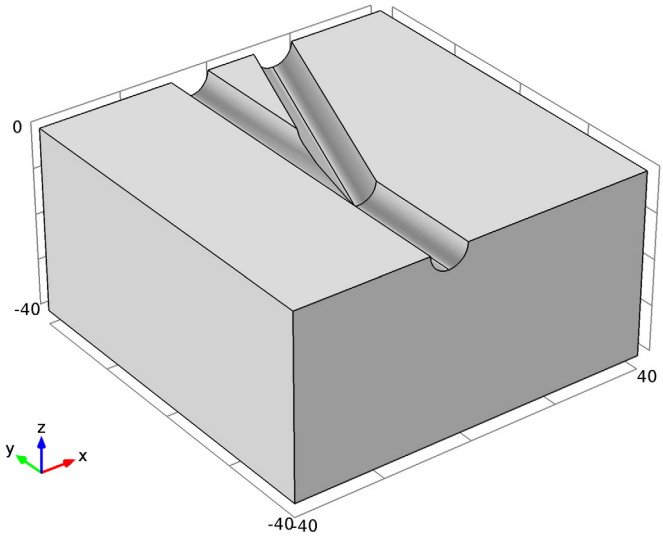
1 Right-click **Model 1>Geometry 1** and choose **Import**.

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `multilateral_well.mphbin`.

5 Click the **Import** button.



GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
p_r	122.45[psi]	Pressure in reservoir
p_w	0[psi]	Pressure in well
So	850[psi]	Coulomb cohesion
phi	31[deg]	Friction angle
C1	14.7	Calibration constant 1

 ©2012 COMSOL

Name	Expression	Description
C2	40	Calibration constant 2
suppX	0	Well support, x-component
suppY	0	Well support, y-component
suppZ	0	Well support, z-component

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
N	2*So*cos(phi)/(1-sin(phi))	Fail parameter 1
Q	(1+sin(phi))/(1-sin(phi))	Fail parameter 2
fail	(((poro.sp3+C1*(p_r-pf))-Q*(poro.sp1+C1*(p_r-pf))+N*(1+(poro.sp2-poro.sp1)/(poro.sp3-poro.sp1)))/C2)[1/psi]	Fail expression

Here, `poro.sp1`, `poro.sp2`, and `poro.sp3` are the principal stresses. Note that the expression `fail` defined in Equation 1 has the dimension of stress. By appending the operator `[1/psi]` to the dimensionful expression enclosed within parentheses you extract its value in psi, which is what Ref. 1 uses. This way, you can analyze the risk for rock failure using the same criterion as in Ref. 1 independently of your choice of base unit system.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Porous matrix` in the **New name** edit field.

4 Click **OK**.

Material 2

1 Right-click **Materials** and choose **Material**.

2 In the **Model Builder** window, under **Model 1>Materials** right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Fluid` in the **New name** edit field.

4 Click **OK**.

POROELASTICITY

Poroelastic Material 1

1 In the **Model Builder** window, expand the **Model 1>Poroelasticity** node, then click **Poroelastic Material 1**.

2 In the **Poroelastic Material** settings window, locate the **Poroelastic Material** section.

3 From the **Porous elastic material** list, choose **Porous matrix**.

4 From the **Specify** list, choose **Young's modulus and Poisson's ratio**.

5 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Fluid**.

MATERIALS

Porous matrix

1 In the **Model Builder** window, under **Model 1>Materials** click **Porous matrix**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	`4.3e5[psi]`
Poisson's ratio	nu	`0.16`
Permeability	kappa	`1e-13[in^2]`
Porosity	epsilon	`0.3`
Density	rho	`0.0861[lb/in^3]`
Biot-Willis coefficient	alphaB	1

Fluid

1 In the **Model Builder** window, under **Model 1>Materials** click **Fluid**.

2 In the **Material** settings window, locate the **Material Contents** section.

 ©2012 COMSOL

3 In the table, enter the following settings:

Property	Name	Value
Density	rho	`0.0361[lb/in^3]`
Dynamic viscosity	mu	`1e-7[psi*s]`
Compressibility of fluid	chif	`4e-10`

POROELASTICITY

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Poroelasticity** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the *pf* edit field, type `p_r`.

Pressure 1

1 In the **Model Builder** window, right-click **Poroelasticity** and choose the boundary condition **Darcy's Law>Pressure**.

2 Select Boundaries 1, 3, and 13 only.

3 In the **Pressure** settings window, locate the **Pressure** section.

4 In the p_0 edit field, type `p_r`.

Pressure 2

1 Right-click **Poroelasticity** and choose the boundary condition **Darcy's Law>Pressure**.

2 Select Boundaries 6–10 only.

3 In the **Pressure** settings window, locate the **Pressure** section.

4 In the p_0 edit field, type `p_w`.

Prescribed Displacement 1

1 Right-click **Poroelasticity** and choose the boundary condition **Solid Mechanics>Prescribed Displacement**.

2 Select Boundaries 1, 3, and 13 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1 Right-click **Poroelasticity** and choose the boundary condition **Solid Mechanics>Prescribed Displacement**.

2 Select Boundaries 2 and 5 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

Prescribed Displacement 3

1 Right-click **Poroelasticity** and choose the boundary condition **Solid Mechanics>Prescribed Displacement**.

2 Select Boundaries 4, 11, and 12 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Boundary Load 1

1 Right-click **Poroelasticity** and choose the boundary condition **Solid Mechanics>Boundary Load**.

2 Select Boundaries 6–10 only.

3 In the **Boundary Load** settings window, locate the **Force** section.

4 In the $\mathbf{F}_A$ table, enter the following settings:

-suppX	x
-suppY	y
-suppZ	z

STUDY 1

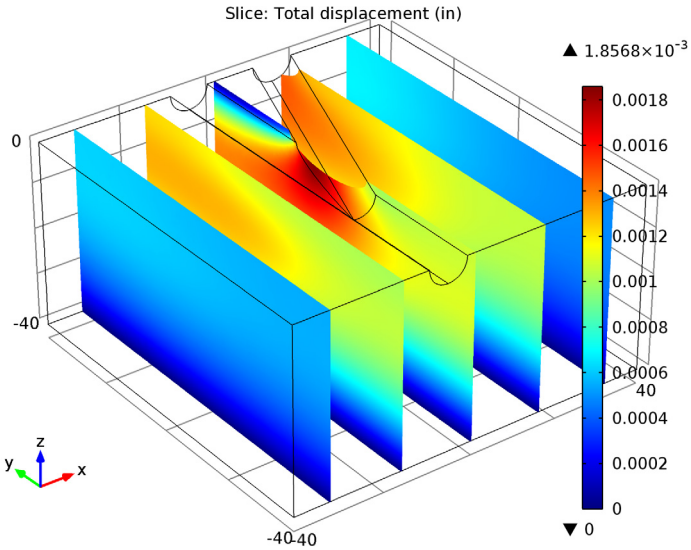
In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

 ©2012 COMSOL

RESULTS

3D Plot Group 1

The default plot is a displacement slice plot.



Add a surface plot of the total displacement with a deformation subnode.

3D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 2** and choose **Surface**.

Keep the default expression, which is the total displacement.

3 Right-click **Results>3D Plot Group 2>Surface 1** and choose **Deformation**.

Again, keep the default expression, which is the displacement field.

4 In the **Settings** window, click **Plot**. Compare the result with that shown in Figure 3.

Next, create a plot that visualizes the pressure and velocity fields.

3D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 3** and choose **Isosurface**.

3 In the **Isosurface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Poroelasticity (Darcy's Law)>Pressure (pf)**.

4 Locate the **Expression** section. From the **Unit** list, choose **psi**.

5 Click the **Scene Light** button on the Graphics toolbar.

6 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Streamline**.

7 In the **Streamline** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Poroelasticity (Darcy's Law)>Darcy's velocity field (Material) (poro.uf,...,poro.wf)**.

8 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.

9 In the **Points** edit field, type `60`.

10 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

11 In the **Tube radius expression** edit field, type `0.4`.

12 From the **Color** list, choose **Custom**.

13 Right-click **Results>3D Plot Group 3>Streamline 1** and choose **Color Expression**.

14 In the **Color Expression** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Poroelasticity (Darcy's Law)>Darcy's velocity magnitude (poro.Uf)**.

15 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

16 Click the **Plot** button.

Finally, investigate the risk for failure by plotting the fail expression.

3D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 4** and choose **Isosurface**.

3 In the **Isosurface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Fail expression (fail)**.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 Click the **Range** button.

6 Go to the **Range** dialog box.

7 In the **Start** edit field, type `-20`.

8 In the **Stop** edit field, type `20`.

9 In the **Step** edit field, type `10`.

10 Click the **Replace** button.

11 In the **Isosurface** settings window, locate the **Coloring and Style** section.

 ©2012 COMSOL

12 From the **Color table** list, choose **WaveLight**.

13 Click the **Plot** button.

 ©2012 COMSOL