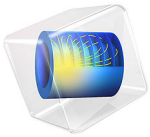

CO2 Storage in a Geologic Formation

Introduction

This example models the underground carbon dioxide storage in a part of the Johansen formation. The Johansen reservoir is a saline aquifer off the coast of Norway. The model is inspired by one of the problems in the benchmark study as reported in Ref. 1. The simulation runs over a period of 50 years. In the first 25 years CO_2 is injected at a constant rate of 15 kg/s. In the subsequent 25 years, the simulation tracks the spreading of the CO_2 due to gravity and capillary forces. This model contains information from the Johansen dataset, which is made available here under the Open Database License (ODbL). Data courtesy of the Norwegian Petroleum Directorate (NPD), the University of Bergen (UiB) and SINTEF.

Model Definition

The part of the reservoir included in the simulation measures approximately 9600 m by 8900 m, with a thickness varying between approximately 90 and 140 m. The geometry, porosity and permeability data are taken from the Johansen dataset (see Ref. 2). The used part of the Johansen formation is shown in Figure 1, with a plot of the porosity distribution. The permeability distribution is shown in Figure 2. The injection well is located at x = 5440 m and y = 3300 m. The total simulation period is 50 years. The CO_2 is injected for the first 25 years at a constant rate of 15 kg/s.

The initial conditions for the simulation are a brine filled reservoir at a hydrostatic pressure distribution (hydraulic head of 0 m). The pressure boundary conditions are a hydraulic head of 0 m at the sides, and no flow at the top and bottom boundaries. CO_2 saturation is 0 at the side boundaries. The (time constant) geothermal gradient is taken to be 0.030 K/m with a temperature of 100°C at a depth of 3000 m.

The pressure and temperature dependent density and viscosity of the supercritical CO_2 are computed using the thermodynamics functionality in COMSOL, using the Peng–Robinson equation of state and the Brokaw model with high pressure correction for the viscosity. The temperature dependent viscosity for the brine is taken to be the viscosity of the Water material from COMSOL Multiphysics' material library. The pressure and temperature dependent density of the brine is approximated using a thermal expansion coefficient of $6{\cdot}10^{-4}$ 1/K, a compressibility of $4{\cdot}10^{-10}$ 1/Pa, and a reference density of 1040 kg/m^3 at $1{\cdot}10^5$ Pa and 360 K. Dissolution of CO_2 into the brine is not taken into account in this model.

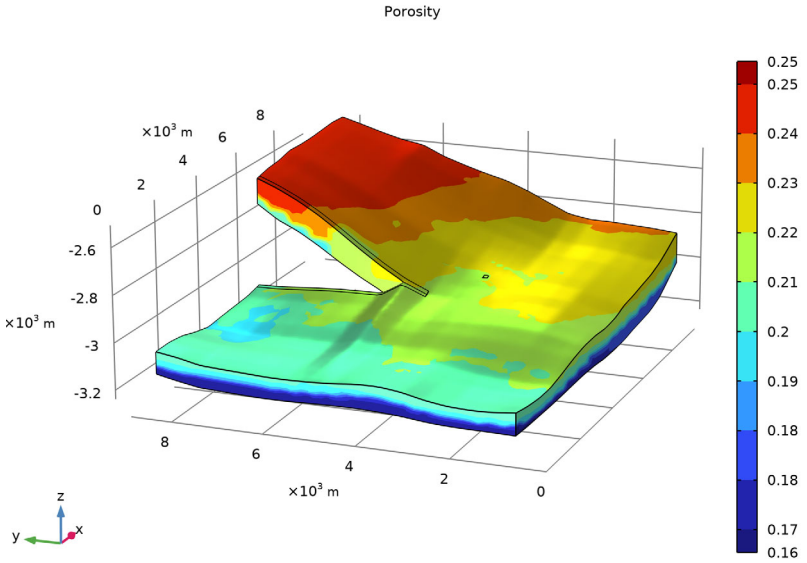

Figure 1: The porosity in the of the part of the Johansen formation that is used for the simulation.

The relative permeabilities and capillary pressure are assumed to be given by the Brooks and Corey capillary pressure model, with the brine being the wetting phase, and with parameters as listed in Table 1.

TABLE 1: BROOKS AND COREY CAPILLARY PRESSURE AND RELATIVE PERMEABILITY PARAMETERS.

Value	Description
0.2	Residual brine saturation
0	Residual carbon dioxide saturation
10^4 Pa	Entry pressure
2.0	Brooks–Corey parameter

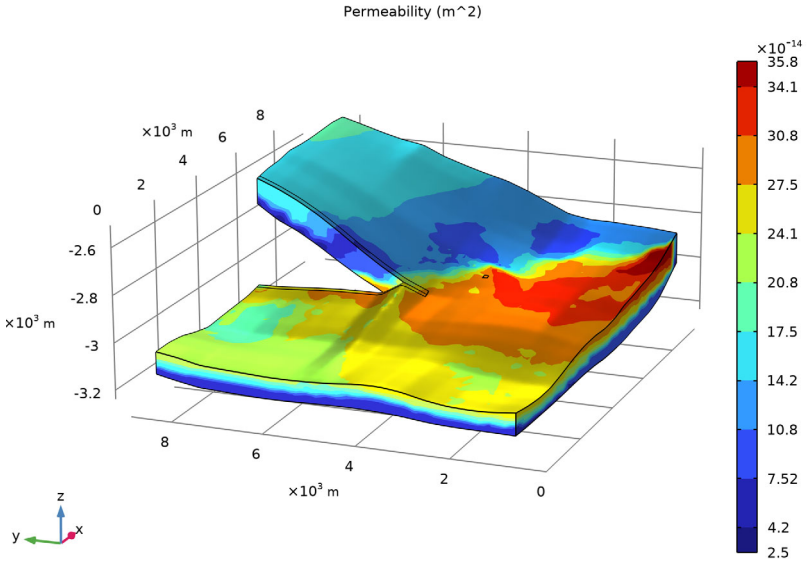

Figure 2: The permeability in the of the part of the Johansen formation that is used for the simulation.

Results and Discussion

As CO_2 is injected, it will start to spread around the well, and it will rise to the top boundary of the formation as the injected CO_2 is lighter than the brine, see for example Figure 3 where the CO_2 saturation is plotted at the end of the injection phase. During the injection phase, the CO_2 plume spreads more or less in all directions, as can been seen in the bottom two plots in Figure 4. Once the injection stops, the CO_2 plume spreads further and it also starts to move to the right and up, as seen from the top in Figure 4, against the slope of the top boundary of the formation, due to gravity. It is to be expected that the CO_2 plume will eventually reach the right and top boundaries (again as seen from the top) of the simulation domain if the simulation would have been over a longer time period. The results for the spreading of the CO_2 plume fit well within the range of the simulation results reported in Ref. 1.

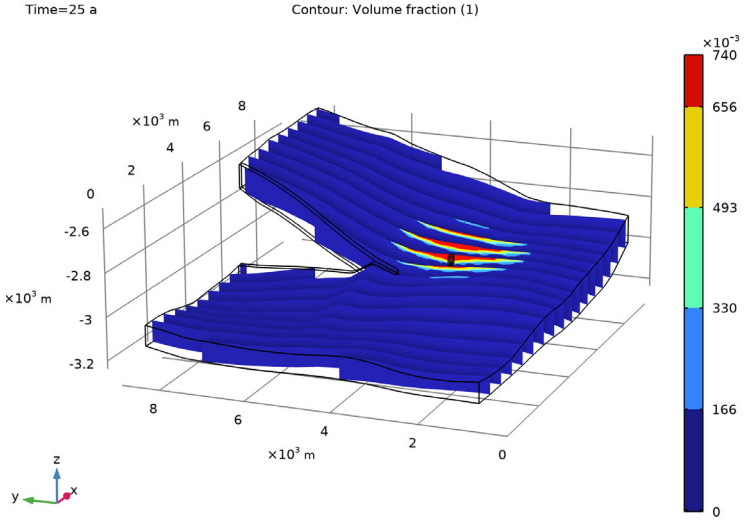

Figure 3: The CO_2 saturation after 25 years of injection.

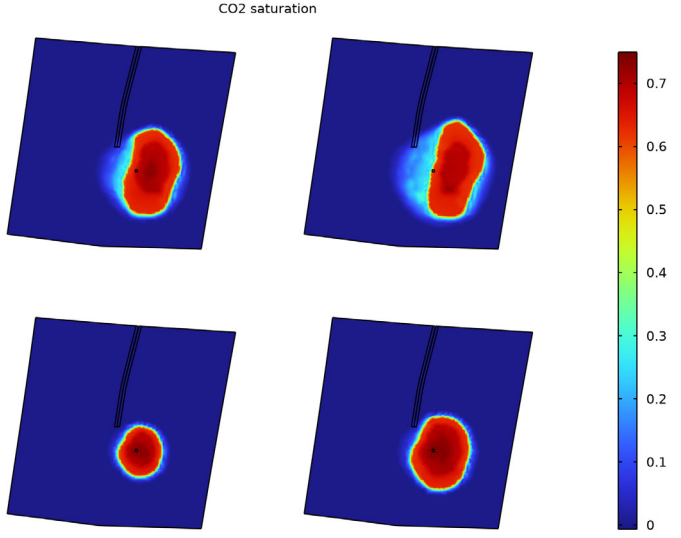

Figure 4: Top view of the formation showing the CO_2 saturation after 12.5 years (bottom left), after 25 years (bottom right,) after 37.5 years (top left), and after 50 years (top right).

References

1. H. Class and others, "A Benchmark Study on problems related to CO_2 Storage in Geologic Formations," *Comput. Geosci.*, vol. 13, no. 4, pp. 409–434, 2009.

2. Available at: https://www.sintef.no/projectweb/matmora/downloads/johansen/. The Johansen dataset is made available under the Open Database License: http://opendatacommons.org/licenses/odbl/1.0/. Any rights in individual contents of the database are licensed under the Database Contents License: http://opendatacommons.org/licenses/dbcl/1.0/.

Application Library path: `Liquid_and_Gas_Properties_Module/Tutorials/carbon_dioxide_storage`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Multiphase Flow in Porous Media**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Time Dependent**.
6. Click **Done**.

GLOBAL DEFINITIONS

The following instructions import parameter values from a file and use the thermodynamics functionality to set up a material with the necessary materials properties for the subcritical carbon dioxide that is injected in the formation.

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `carbon_dioxide_storage_parameters.txt`.

5 In the **Physics** toolbar, click **Thermodynamics** and choose **Thermodynamic System**.

SELECT SYSTEM

1 Go to the **Select System** window.

2 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 In the **Species** list, select **carbon dioxide (124-38-9, CO2)**.

3 Click **Add Selected**.

4 Click **Next** in the window toolbar.

SELECT THERMODYNAMIC MODEL

1 Go to the **Select Thermodynamic Model** window.

2 From the **Gas phase model** list, choose **Peng-Robinson**.

3 Select the **Advanced options** check box.

4 In the table, enter the following settings:

Property	Model
Gas viscosity	Brokaw (with high pressure correction)

5 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Gas System 1 (pp1)

In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Gas System 1 (pp1)** and choose **Generate Material**.

SELECT PHASE

1 Go to the **Select Phase** window.

2 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Next** in the window toolbar.

SELECT PROPERTIES

1 Go to the **Select Properties** window.

2 Click **Next** in the window toolbar.

DEFINE MATERIAL

1 Go to the **Define Material** window.

2 From the **Function type** list, choose **Interpolation**.

3 From the **Maximum number of interpolation points** list, choose **Fine (2500)**.

4 In row **Temperature**, set **High** to 400.

5 In row **Temperature**, set **Number of points** to 30.

6 In row **Pressure**, set **Low** to 1e5.

7 In row **Pressure**, set **High** to 4e7.

8 In row **Pressure**, set **Number of points** to 80.

9 Click **Finish** in the window toolbar.

DEFINITIONS

The following instructions define the geothermal temperature distribution and import the porosity and permeability distributions in the formation and set up interpolation functions for use in the simulation.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Temp	(-3000[m]-z)*0.03[K/m]+100[degC]	K	Geothermal temperature distribution

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `carbon_dioxide_storage_porper.csv`.

6 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
`porosity`	1

7 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
porosity	1

8 In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	`m`
Column 2	`m`
Column 3	`m`

9 Locate the **Definition** section. Click **Import**.

10 Select the **Use spatial coordinates as arguments** check box.

11 From the **Frame** list, choose **Mesh**.

12 Select the **Reinterpolate interpolation data on computational mesh** check box. This makes sure that the interpolation function is evaluated using cached values at Lagrange points on the computational mesh, resulting in a much more efficient evaluation during the computation.

Interpolation 2 (int2)

1 In the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `carbon_dioxide_storage_porper.csv`.

6 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
permeability	2

7 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
permeability	mD

8 In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	m
Column 2	m
Column 3	m

9 Locate the **Definition** section. Click **Import**.

10 Select the **Use spatial coordinates as arguments** check box.

11 From the **Frame** list, choose **Mesh**.

12 Select the **Reinterpolate interpolation data on computational mesh** check box.

Analytic 1 (an1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `density_brine` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `rho0-rho0*alpha*(T-T0)+rho0*beta*(p-p0)`.

4 In the **Arguments** text field, type `T, p`.

5 Locate the **Units** section. In the **Function** text field, type `kg/m^3`.

6 In the table, enter the following settings:

Argument	Unit
T	K
p	Pa

The following instructions add a rectangle function. It will be used at the injection well to gradually ramp up the carbon dioxide injection rate over a period of 0.1 year and to switch it off after 25 years.

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Parameters** section.

3 In the **Lower limit** text field, type `0.05`.

4 In the **Upper limit** text field, type `25`.

GEOMETRY 1

The following instructions import the geometry of the Johansen formation and create an edge at the position of the injection well. Around the well a block is created for meshing purposes.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `carbon_dioxide_storage.mphbin`.

5 Click **Import**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `x_well`.

5 In the **y** text field, type `y_well`.

6 In the **z** text field, type `-3200`.

7 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

8 In the **x** text field, type `x_well`.

9 In the **y** text field, type `y_well`.

10 In the **z** text field, type `-2500`.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `100`.

4 In the **Depth** text field, type `100`.

5 In the **Height** text field, type `500`.

6 Locate the **Position** section. In the **x** text field, type `x_well-50`.

7 In the **y** text field, type `y_well-50`.

8 In the **z** text field, type `-3200`.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Objects**.

2 Select the objects **blk1** and **ls1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Tool objects** subsection. Click to select the  **Activate Selection** toggle button.

5 Select the object **imp1** only.

6 Select the **Keep tool objects** check box.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **par1(1)**, select Domains 1 and 3 only.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 On the object **par1(2)**, select Edges 1 and 3 only.

Partition Edges 1 (pare1)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **del2**, select Edge 1 only.

3 In the **Settings** window for **Partition Edges**, locate the **Positions** section.

4 In the table, enter the following settings:

Relative arc length parameters
0.25
0.75

Delete Entities 3 (del3)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 On the object **pare1**, select Edges 1 and 3 only.

5 Click **Build All Objects**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PHASE TRANSPORT IN POROUS MEDIA (PHTR)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

3 Click **OK**.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Transport in Porous Media (phtr)**.

5 In the **Settings** window for **Phase Transport in Porous Media**, locate the **Gravity Effects** section.

6 Select the **Include gravity** check box.

7 Click to expand the **Quadrature Settings** section. Clear the **Use automatic quadrature settings** check box.

8 In the **Integration order** text field, type 4. This increases the default integration order to guaranty a more accurate evaluation of the nonlinear coefficients in the equations.

Phase and Porous Media Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Phase Transport in Porous Media (phtr)** click **Phase and Porous Media Transport Properties 1**.

2 In the **Settings** window for **Phase and Porous Media Transport Properties**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `Temp`.

4 Locate the **Capillary Pressure** section. From the **Capillary pressure model** list, choose **Brooks and Corey**.

5 In the p_{ec} text field, type `pc_e`.

6 In the λ_p text field, type `lambda`.

7 Locate the **Phase 1 Properties** section. From the **Fluid s1** list, choose **Water, liquid (mat1)**.

8 Locate the **Phase 2 Properties** section. From the **Fluid s2** list, choose **Gas: carbon dioxide(1) 1 (pp1mat1)**.

9 Locate the **Phase 1 Properties** section. From the ρ_{s1} list, choose **User defined**. In the associated text field, type `density_brine(Temp,dl.pA)`.

10 In the s_{rs1} text field, type `s0_b`.

Volume Fraction 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Volume Fraction**.

2 Select Boundaries 1, 2, 5, 12, 14, 21, and 23–25 only.

3 In the **Settings** window for **Volume Fraction**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Sides` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for **Volume Fraction**, locate the **Volume Fraction** section.

8 Select the **Phase s2** check box.

DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, locate the **Gravity Effects** section.

3 Select the **Include gravity** check box.

4 Click to expand the **Discretization** section. From the **Pressure** list, choose **Linear**. Switching to a lower element order reduces the number of degrees of freedom to make the simulation more computationally efficient. Since the used mesh is quite fine, this will not reduce the accuracy significantly.

Porous Matrix 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)> Porous Medium 1** click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_p list, choose **User defined**. In the associated text field, type `porosity`.

4 From the κ list, choose **User defined**. In the associated text field, type `permeability`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Click the **Hydraulic head** button.

Hydraulic Head 1

1 In the **Physics** toolbar, click  **Boundaries** and choose **Hydraulic Head**.

2 In the **Settings** window for **Hydraulic Head**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Sides**.

MULTIPHYSICS

Well 1 (wellmpe1)

1 In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Edge>Well**.

2 Select Edge 38 only.

3 In the **Settings** window for **Well**, locate the **Well** section.

4 In the M_0 text field, type `r_injection*rect1(t/1[a])`.

5 Locate the **Phase 2** section. In the $M_{0,s2}$ text field, type `r_injection*rect1(t/1[a])`.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click  **Boundary** and choose **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Finer**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 120.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 Select Boundaries 4, 8, 9, 11, 13, and 19 only.

Swept 1

1 In the **Mesh** toolbar, click  **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 1 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type 9.

Edge 1

1 In the **Mesh** toolbar, click  **Boundary** and choose **Edge**.

2 Select Edge 38 only.

Distribution 1

Right-click **Edge 1** and choose **Distribution**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click  **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Scale Geometry** section.

3 In the **z-direction scale** text field, type 10.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

6 Click  **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **a**.

4 In the **Output times** text field, type `range(0,2.5,50)`.

5 From the **Tolerance** list, choose **User controlled**.

6 In the **Relative tolerance** text field, type `0.001`.

Solution 1 (sol1)

1 In the **Study** toolbar, click  **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

4 From the **Method** list, choose **Initial value based**.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

7 From the **Jacobian update** list, choose **Minimal**.

8 In the **Tolerance factor** text field, type `0.5`.

9 In the **Study** toolbar, click  **Compute**.

DEFINITIONS

Since the geometry of the formation is thin in the vertical direction, it can be difficult to get a good impression of the distribution of the results in the *z* direction in plots. The following instructions change the view settings such that the plots are scaled in the vertical direction to get a better impression of the results.

Camera

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Camera**.

2 In the **Settings** window for **Camera**, locate the **Camera** section.

3 From the **View scale** list, choose **Manual**.

4 In the **z scale** text field, type `5`.

Create a new plot for the porosity.

RESULTS

Porosity

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Porosity` in the **Label** text field.

Contour 1

1 Right-click **Porosity** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `porosity`.

4 Locate the **Levels** section. In the **Total levels** text field, type `10`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

Porosity

1 In the **Model Builder** window, click **Porosity**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Porosity`.

5 Clear the **Parameter indicator** text field.

6 In the **Porosity** toolbar, click **Plot**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

The previous instructions created the plot of the porosity as shown in Figure 1. Reproduce the plot in Figure 2 by replacing the expression in the **Expression** text field by `permeability`.

The following instructions create the plots in Figure 3 and Figure 4.

Cut Plane 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Cut Plane**.

3 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

4 In the **x-coordinate** text field, type `6000`.

5 Select the **Additional parallel planes** check box.

6 In the **Distances** text field, type `range(-6000,500,9000)`.

CO2 Saturation

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `CO2 Saturation` in the **Label** text field.

3 Locate the **Data** section. From the **Time (a)** list, choose **25**.

Contour 1

1 Right-click **CO2 Saturation** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Expression** section. In the **Expression** text field, type `s2`.

6 Locate the **Levels** section. In the **Total levels** text field, type `5`.

7 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

8 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Ambient Occlusion**.

9 In the **CO2 Saturation** toolbar, click **Plot**.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 Select Boundaries 4, 9, 13, and 19 only.

2D Plot Group 7

In the **Results** toolbar, click **2D Plot Group**.

Surface 1

1 Right-click **2D Plot Group 7** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `s2`.

CO2 Saturation, Top View

1 In the **Model Builder** window, under **Results** click **2D Plot Group 7**.

2 In the **Settings** window for **2D Plot Group**, type `CO2 Saturation, Top View` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Surface 1**.

4 From the **Time (a)** list, choose **12.5**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 Clear the **Parameter indicator** text field.

7 In the **Title** text area, type `CO2 saturation`.

8 Click to expand the **Plot Array** section. Select the **Enable** check box.

9 From the **Array shape** list, choose **Square**.

Surface 2

1 In the **Model Builder** window, under **Results>CO2 Saturation, Top View** right-click **Surface 1** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Surface 1**.

4 From the **Time (a)** list, choose **25**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 3

1 Right-click **Surface 2** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Time (a)** list, choose **37.5**.

Surface 4

1 Right-click **Surface 3** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, click **Plot Last**.

3 Click the **Show Grid** button in the **Graphics** toolbar.

4 In the **CO2 Saturation, Top View** toolbar, click **Plot**.