Abaqus/Standard Simulation of Ground Subsidence due to Oil and Gas Extraction

Abaqus Technology Brief

Summary

Extraction of oil and gas from an underground reservoir alters the pore pressure distribution in the reservoir strata. Modifications in pore pressure can lead to plastic deformation in the strata, which can in turn modify the pore fluid flow and lead to ground subsidence.

Since subsidence can affect the operability of equipment on the ground as well as the safety of buildings and structures, it needs to be accurately predicted and kept within limits. In this Technology Brief, we present a method for predicting the rock deformation and ground subsidence resulting from oil and gas extraction.

Background

The oil and gas fluids in an underground reservoir are under pressure, and this pressure is released or altered when the fluids are extracted. The fluid pressure can be affected by the permeability and compaction properties of the rock, the locations of the bore holes, and the specific extraction and recovery procedures employed.

As the fluids are extracted, the reservoir rock deforms and undergoes a change in its porosity, which then leads to modifications in the permeability. The phenomena of fluid flow and rock deformation are thus coupled; the fluid flow leads to deformations, which in turn can affect the fluid flow. Simulation of a coupled reservoir geomechanics system would therefore require one to take into account the movement of the different fluids as well as the inelastic deformations in the rock.

The simulation of the flow of different fluids within the reservoir is itself a complex problem, and sophisticated tools that are specifically designed to address flow complexities such as phase changes and miscibility are available. However, these tools may not directly include deformations in the rock and their effect on the fluid flow. Abaqus, on the other hand, is well suited for modeling inelastic deformations in the rock, but is not able to simulate the multiphase complexities in the fluid flow regime.

We can, however, combine the strengths of different tools. For example, using a reservoir flow simulation code, the flow problem can be solved to compute the pore pressure depletion history; Abaqus can then be used to compute the inelastic deformation in the reservoir rock. A methodology based on this approach is described in [1], wherein the flow simulation is performed using ECLIPSE (available from Schlumberger, Ltd.), and the geomechanics simulation is performed using Abaqus/Standard.

The approach taken in [1] involves considerable manual effort to create and modify the finite element mesh and transfer the pore pressures from ECLIPSE to Abaqus. However, as explained in [2], it is now possible to automate several of these tasks using more recent capabilities available in Abaqus. In this Technology Brief we describe a representative reservoir geomechanics application that uses these new capabilities.

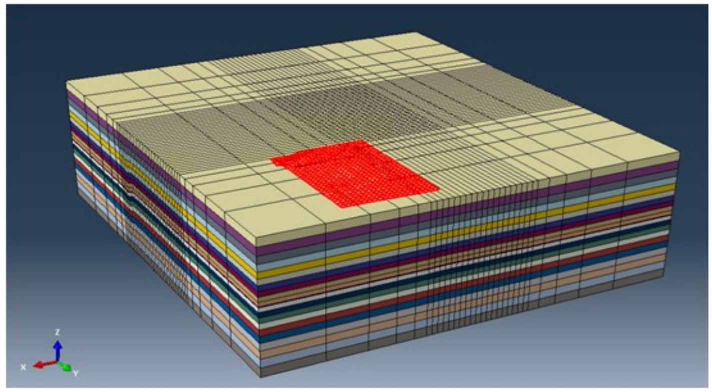

Figure 1: A representative oil and gas reservoir model. The reservoir region is shown in red and the surrounding geological layers are shown with independent colors.

Key Abaqus Features and Benefits

- Fully coupled displacement-pore fluid flow solution method in Abaqus/Standard
- Geomechanics material models
- Submodeling technique to drive detailed local models from global model results
- Python scripting for automation of modeling and job submission tasks

Finite Element Model

We assume that a flow simulation has been performed using ECLIPSE and the pore pressure depletion history is available. For this Technology Brief we use the data for the PUNQ reservoir [1]. Using a translator, an Abaqus output database is created from the ECLIPSE flow simulation results. This output database contains the finite element mesh data, the initial values of porosity and void ratio, as well as the pore pressure history.

Abaqus can represent only a single fluid within each finite element. Therefore, even though multiple fluids exist in each flow simulation grid cell, the appropriate fluid to be used for each finite element must be selected. Using the fluid saturation values and the grid cell volume data, the translator automatically locates and designates regions that are principally gas-bearing, oil-bearing, or water-bearing for each reservoir layer. The specific weight val-

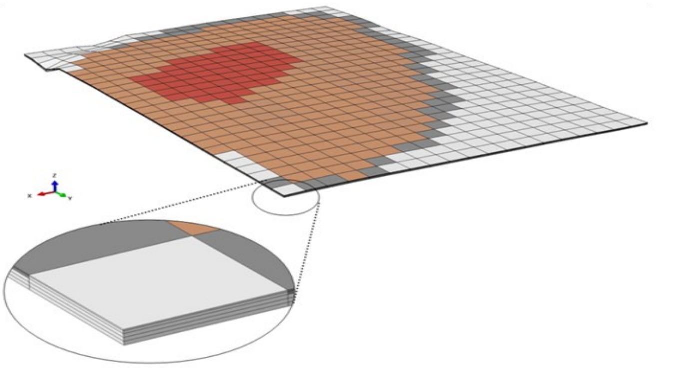

Figure 2: Finite element mesh of the PUNQ reservoir obtained from the flow simulation grid. Gas-filled regions are colored red, oil-filled regions brown, and water-filled regions dark grey. The reservoir consists of five layers, as shown in the zoomed view.

ues of the fluid are also derived and written to the output database. The output database is then imported into Abaqus/CAE. Figure 2 shows the finite element mesh output by the translator, highlighting the regions determined to contain gas, oil, and water.

A script is used in Abaqus/CAE to manage the sequence of operations used to modify the mesh and to specify density, specific weight, material properties, initial conditions, and boundary conditions. The mesh is first edited, mainly to merge extremely thin layers in the reservoir. For example, layer 2, which is the second layer from the top of the reservoir, is found to be very thin in some regions and is hence merged with layer 3. Figure 3 shows a zoomed view of the unedited mesh on the left and the edited mesh on the right.

Layers of elements are then added below, on the sides, and on top of the reservoir mesh to represent the under, side, and over burden regions, respectively. The grid for these regions is not present in the original flow simulation model and hence no corresponding mesh is present in the output database created by the translator. Figure 4 shows the mesh after adding elements to represent the burden regions. The burden regions are added such that the top surface of the model is located at ground level and the bottom surface at a depth of 5.0 km. The final dimensions of the model then become 15.0 km x 13.5 km in the horizontal plane. Material density is specified using distributions that are created by the script based on an analytical function. Gravity loading is then applied.

<u>*Initial and Boundary Conditions*</u>

Initial stress, pore pressure, and void ratio conditions are specified. The initial void ratios and pore pressures are read directly from the output database created by the translator. Initial stresses are specified as a piece-wise linear distribution through the model depth.

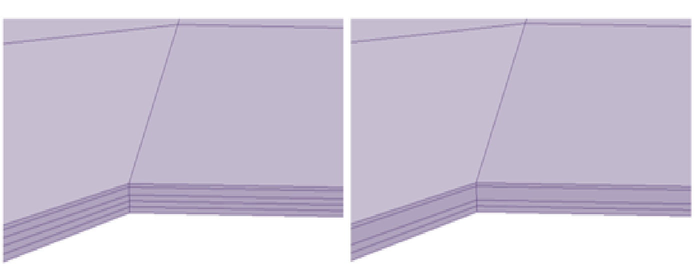

Figure 3: Zoomed views of the finite element mesh before editing (left) and after layers 2 and 3 from the top are merged (right).

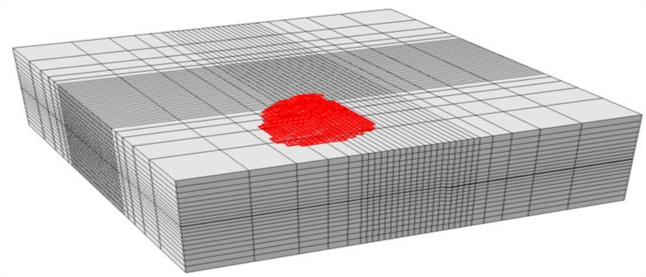

Figure 4: Finite element mesh after the addition of elements representing the under, side, and over burden regions. The fluid-containing region of the reservoir is colored red.

Displacements normal to the side and bottom boundary surfaces are specified as zero. The submodeling technique is used to specify pore pressure boundary conditions in the reservoir region. Pore pressure values within the reservoir are driven by the values computed from the flow simulation; these are available in the output database created by the translator. As pore pressures are specified at all nodes that have pore pressure degrees of freedom, the permeability values used for the analysis are inconsequential.

<u>*Material Properties and Analysis*</u>

The geomechanics simulation involves two distinct analyses. An elastic geostatic analysis is first performed to obtain the vertical stress distribution. For this analysis the porous reservoir rock is modeled as porous elastic. Nonporous regions, including the under, side, and over burden regions are modeled as linear elastic. The analysis is allowed to iterate until the displacements obtained for the applied gravity load are nearly zero. This analysis thus provides the stress distribution that equilibrates the applied loading and boundary conditions.

For the subsequent analysis the reservoir rock is modeled as elastoplastic, using the clay plasticity material model with exponential hardening. The vertical stress values obtained from the elastic analysis are used to compute the compressibility values of the reservoir rock; these values are then used to compute the logarithmic plastic bulk modulus, and also update the logarithmic elastic bulk modulus for the porous region. The compressibility values are also used to update the linear elastic properties of the nonporous regions including those of the under, side, and over burden regions. A steady-state coupled pore fluid flow displacement analysis using these elastoplastic material properties is then performed for 30 time increments representing 30 time-history pore pressure depletion data sets obtained from the flow simulation. Displacements and plastic strain values for the reservoir and the surrounding burden regions are obtained from this analysis.

Results and Discussion

The mesh used for the geomechanics analysis is different from the mesh derived from the flow simulation grid. In particular, layers 2 and 3 in the flow simulation grid have been merged in the mesh used for the geomechanics analysis. Variables, including pore pressures and initial void ratios, hence, need to be accurately transferred from one mesh to the other. Mapping functionalities available in Abaqus can be used to transfer these data values. Figure 5 shows a comparison of the initial void ratio distribution in the two meshes. The void ratio values are consistently mapped from one mesh to the other.

Figure 6 shows contours of pore pressure at the end of the flow simulation and at the end of the elastoplastic

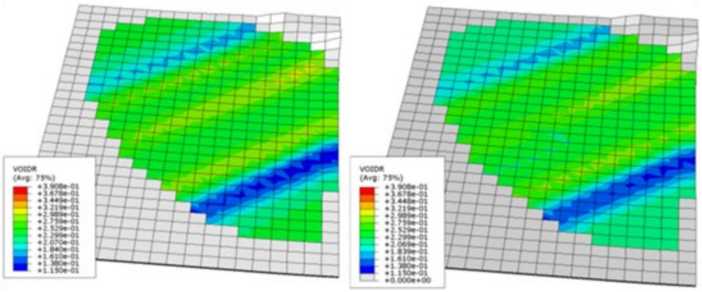

Figure 5: Initial values of void ratio from flow simulation (left) and after mapping onto the finite element mesh (right)

geomechanics analysis. The pore pressure values in the geomechanics simulation are consistent with the submodeling-driven boundary conditions used for the analysis.

Figure 7 shows contours of displacement in the vertical direction in the reservoir region at the end of the analysis. The top surface of the reservoir settles downwards and the maximum value is about 4 cm.

Figure 8 shows the contours of plastic strain in the vertical direction at two time instances, at 50% of the total depletion time, and at the end of depletion. These are plotted on plan views of the fluid-containing region of the reservoir. The plastic strain magnitude increases with time, indicating that the fluid-containing region of the reservoir gets compacted as fluids are extracted.

Figure 9 shows the contours of vertical displacement of the ground. A maximum subsidence of about 2.5 cm is obtained just above the reservoir. The subsidence values decrease with distance away from this location.

Figure 10 shows contours of vertical displacement of the ground displayed on the model in plan view. The fluid-containing region of the reservoir is colored red. Displacement contour lines of –0.5 mm and –2.5 mm have been highlighted in this figure. The ground subsidence decreases rapidly with distance away from its maximum location just above the reservoir. The subsidence is less than a millimeter at a horizontal distance of approximately the horizontal span of the fluid-containing region of the reservoir.

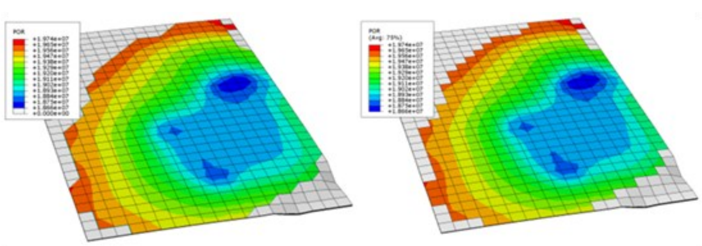

Figure 6: Pore pressure (Pa) at the end of the flow simulation (left) and at the end of the elastoplastic geomechanics analysis (right)

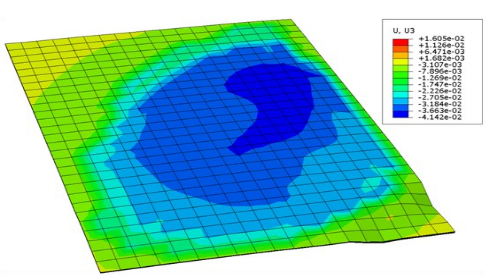

Figure 7: Vertical displacement (m) in the reservoir region

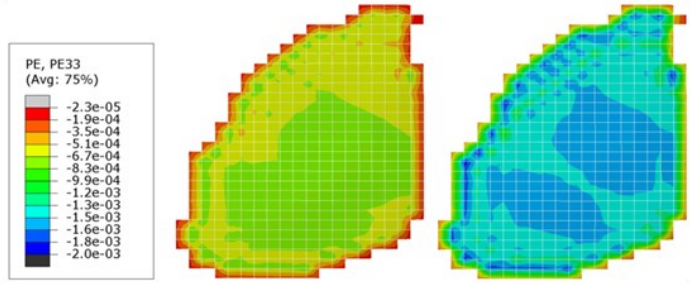

Figure 8: Plastic strain in the vertical direction at 50% depletion (left), and at the end of depletion (right)

Figure 11 shows contours of vertical displacement of the ground surface from the analysis presented in [1]. The results show a strong correlation to the displacements in Figure 10.

The methodology presented in this Technology Brief can be used to compute reservoir compaction and surface subsidence of the ground. Additionally, the geomechanics model can be used as a global model to drive submodels for smaller scale applications such as bore-hole stability. The method presented in this Technology Brief can also be used as one of the components of an iteratively coupled technique for reservoir geomechanics simulation. In such a technique the modifications in void ratios arising from reservoir compaction are transferred to the flow simulation code, which would then be able to modify the flow permeability values for an incremental flow simulation.

Acknowledgements

DS SIMULIA is thankful to Eni, SpA for providing the flow simulation data and results files for the PUNQ model.

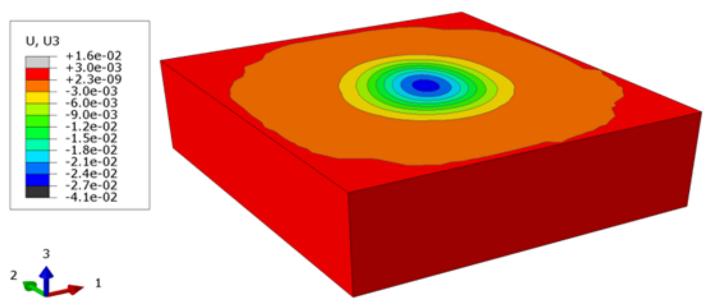

Figure 9: Vertical displacement of the ground (m)

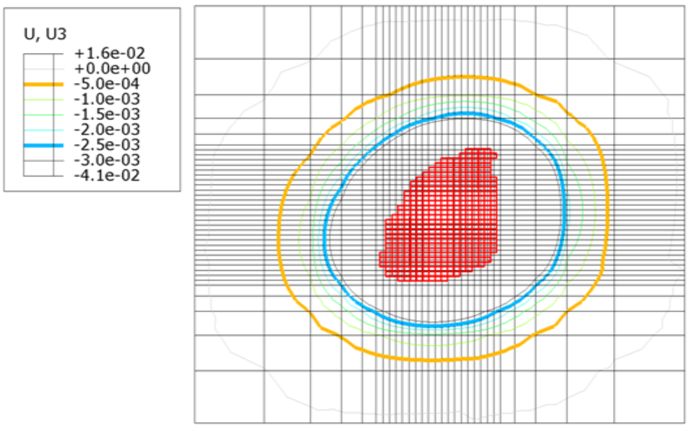

Figure 10: Vertical displacement of the ground (m)

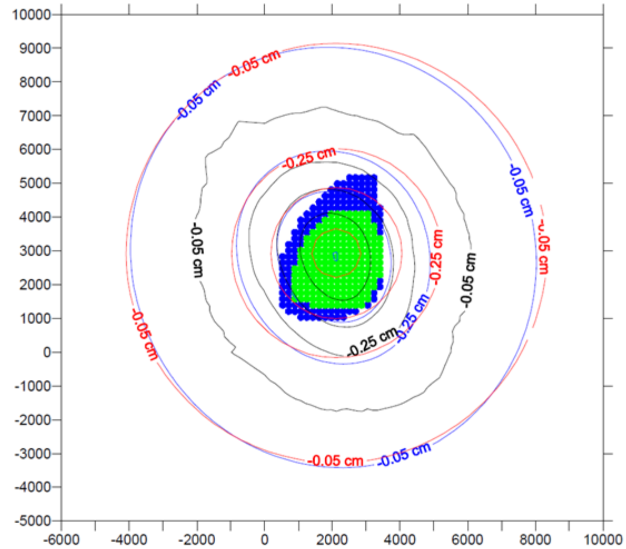

Figure 11: Vertical displacement of the ground from the analysis presented in [1]

References

1. "Numerical Simulation of Compaction and Subsidence using Abaqus," G. Capasso and S. Mantica, Abaqus Users' Conference, 2006, Boston, USA.

2. "Field Scale Geomechanical Modelling using a New Automated Workflow," S. Monaco, G. Capasso, S. Mantica, D. Datye, and R. Vitali, International Petroleum Technology Conference, 7-9 February 2012, Bangkok, Thailand.

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors.

DASSAULT SYSTEMES

Delivering Best-in-Class Products

CATIA
Virtual Product Design

EXALEAD
Information Intelligence

SOLIDWORKS
3D for Professionals

NETVIBES
Dashboard Intelligence

SIMULIA
Realistic Simulation

3DSWYM
Social Innovation

DELMIA
Virtual Production

3DVIA
Online 3D Lifelike Experiences

ENOVIA
Global Collaborative Lifecycle Management

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes in all industries in more than 80 countries. For more information, visit www.3ds.com.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA