

Field Scale Geomechanical Modelling Using a New Automated Workflow in Abaqus

S. Monaco¹, G. Capasso¹, D. Datye², S. Mantica¹ and R. Vitali³

1. eni e&p, S. Donato Milanese (MI), Italy
2. Dassault Systèmes Simulia Corporation, Providence (RI), USA
3. Simulia Italia, Lainate (MI), Italy

Abstract: Abaqus has been used for many years in eni as the main stress/strain simulator for studying the geomechanical behaviour of reservoirs both at field and well scale. In the work presented in this paper, the new capabilities developed in Abaqus/CAE are applied with reference to a field scale geomechanical study performed on a realistic test case. As shown in the example application, the capabilities developed in Abaqus/CAE are particularly suitable for reservoir geomechanical simulations, and allow one to perform accurate analyses that overcome the limitations of the previous workflow and simultaneously provide significant automation.

Keywords: Cam-Clay, Constitutive Model, Faults, Geomechanics, Hydrocarbon Production, Reservoir Compaction, Surface Subsidence.

1. Introduction

Up to now, the study of geomechanical effects due to field scale reservoir depletion has been performed following the steps of the workflow developed internally by eni and described in (Capasso and Mantica, 2006).

However, this workflow includes non-automated procedures as well as simplifications related to the geometry description, such as smearing of faults and simplified treatment of collapsing layers. The new features implemented in Abaqus 6.11 for soils analyses definitely change the approach to geomechanical reservoir simulation by allowing a complete automated workflow that can be managed through Abaqus/CAE.

A reservoir modeler plug-in, which covers all the steps of the standard workflow for subsidence studies, has been implemented. Each task of the workflow from the creation of the Eclipse odb using the translator (built in to establish a link between the flow-dynamic simulator Eclipse and the stress simulator Abaqus) to the final step of computing the plastic properties is listed in the GUI. The Upscaling and Burdens steps are performed outside of the plug-in, but their statuses are documented within the plug-in and they are properly executed in Abaqus/CAE. All information entered by the user is stored on the custom data base on a model-by-model basis and is persistent across sessions. This allows for providing significant automation and makes it easier to the user to complete all the steps for the creation of a reservoir model.

2.2 Reservoir Modeler Tree overview

A convenient, model tree-based approach is adopted for the Eclipse reservoir modeler GUI. A new tab named “Reservoir Modeler” (RM) appears in the model tree when the plug-in is enabled; this tab is in addition to the standard Abaqus/CAE model tree tabs Model and Results (and any other custom tabs) as shown in Figure 2.

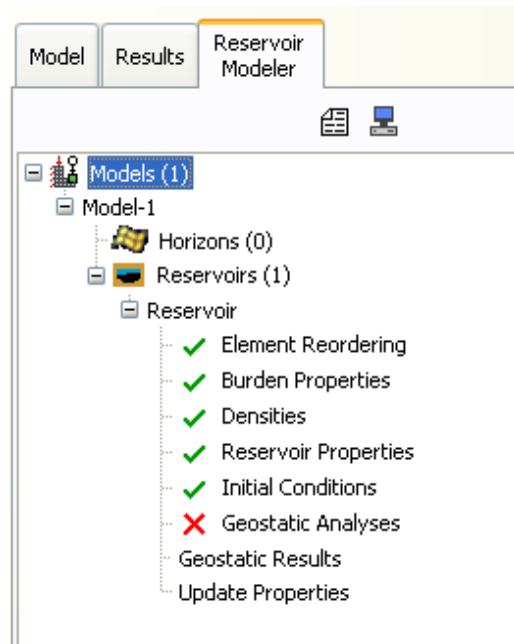


Figure 2. Expanded Tree of the Reservoir Modeler in Abaqus/CAE.

The creation of a model using the RM involves the completion of a sequence of steps which will be described in detail in the next section. In order to perform any particular step, all of the previous steps must have already been completed. The RM provides feedback on the progress of each step by modifying the icons associated with each step. No icon indicates that the user has not yet entered this information. A green check indicates that the user has entered this information and any necessary tasks have been executed successfully. A red check indicates that the information has been entered, but the step failed execution or is out-of-date due to changes to a previous step. If changes are made to a given step, all the subsequent steps must then be regenerated.

This is the workflow and the GUI which have been adopted to perform the study of the synthetic Brugge field. All the steps passed through to create the geomechanical model and to perform elastic and elasto-plastic simulations are described in detail in the next section.

3. Reservoir geomechanical model generation workflow applied to the Brugge field

3.1 General description of the Brugge field

The Brugge field is a complete synthetic oil field whose structure consists of an E-W elongated half-dome with a large boundary fault at its northern edge and one internal fault with a modest throw.

The Eclipse model is composed of 139x48x9 cells in directions I, J and K respectively, adding up to a total of 60048 cells among which 44464 are active. Figure 3 provides a top view of the first reservoir layer where the intra-field fault is evident.

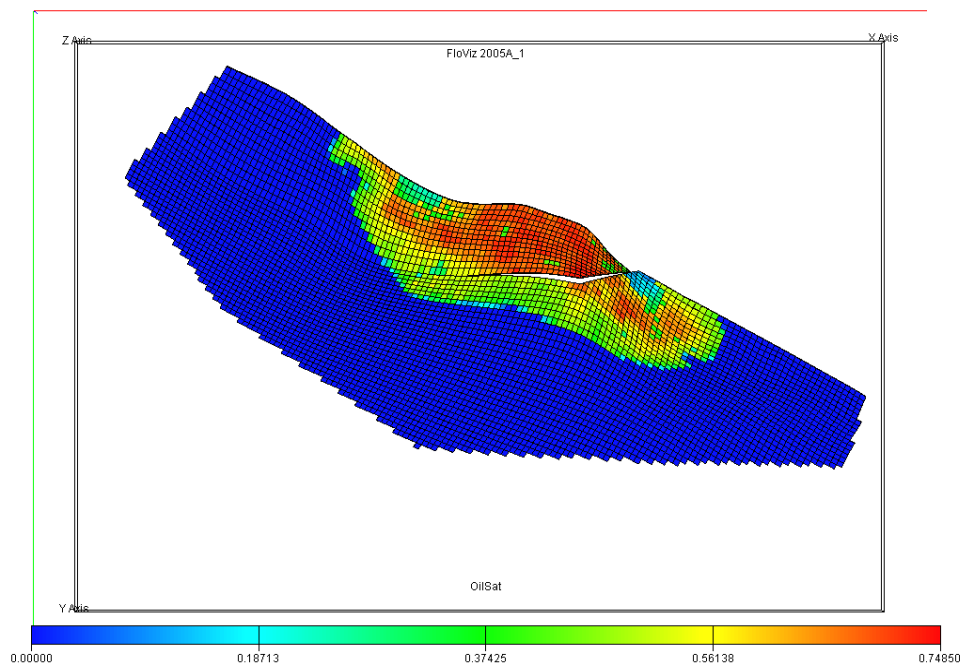


Figure 3. Brugge Field Eclipse discretization: top view of Layer 1.

The field has 10 injector and 20 producer wells, but for this study we consider the injectors to be deactivated. The pressure controls on the bottom hole pressure has also been lowered in order to obtain higher depletion and associated subsidence problems.

The production start is set for 1st March 2016 and the end on 1st January 2028.

3.2 Eclipse output file translation

The first step in the creation of the reservoir model is performed by using the fromreservoir translator to obtain an Abaqus output database (odb) file equivalent to the Eclipse model. The Eclipse reservoir simulation data translation converts cell data to equivalent Abaqus element definitions, and pressure history descriptions to pore pressure field output frames. The created odb

contains mesh data, initial void ratio data, initial pore pressure values, pore pressure history data, and specific weight values. It also contains other data, such as element set information for the material regions.

By double-clicking on the  icon from within the RM, a dialog is opened up. Figure 4 shows how it has been filled to run the translator for the Brugge field.

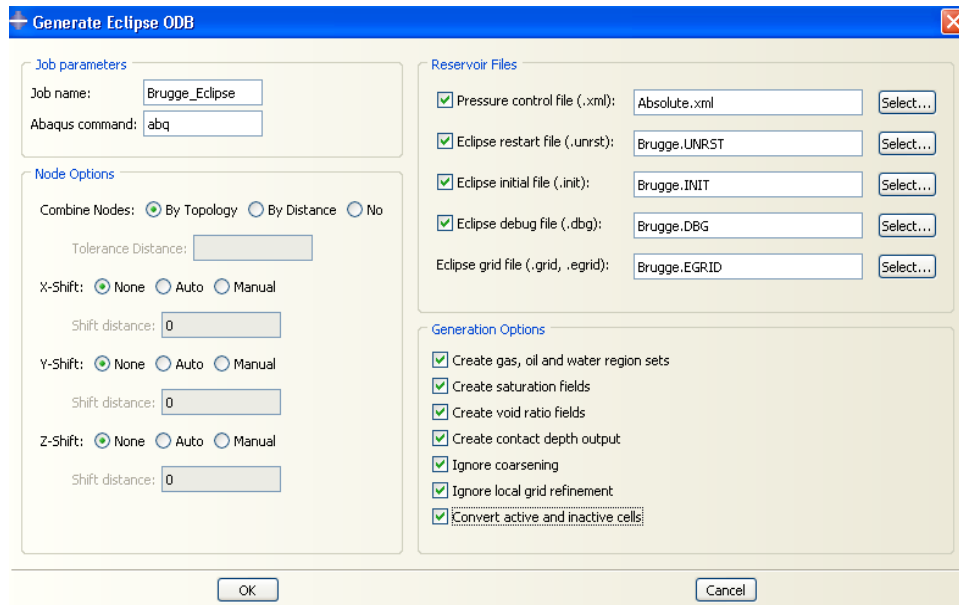


Figure 4. GUI for invoking the fromreservoir translator.

The translator converts Eclipse model cells into Abaqus elements. Cells labeled as active in the Eclipse model are converted to C3D8P elements, which then enable definitions of pore pressure history at the element nodes. Eclipse inactive cells are converted to C3D8 elements.

The Eclipse grid layout is regular in all three coordinate directions and each finite difference hexahedral cell is defined with eight vertex points that are not explicitly connected to neighboring cells. This definition format implies cell adjacency rather than explicitly defining it. To facilitate efficient mapping, the translator infers cell adjacency and creates element connectivity using either topology-based or distance-based point consolidation method.

Since the Eclipse model of the Brugge field is characterized by a big gap due to the fault, the topology-based method is chosen and Eclipse grid points are combined to create output database nodes based on their topological adjacency. Eclipse vertex coordinate discrepancies at adjacent elements are thus ignored.

According to the kind of simulation (elastic or elasto-plastic), a different .xml configuration file has been used in order to perform the pressure conversion in terms of relative or absolute pressure.

From the .DBG Eclipse file, the specific weight is computed for each equilibration region and then used to calculate the initial pressure for the elasto-plastic simulation, as described in §3.11.

All the generation options in the GUI are flagged in order to convert Eclipse model cells into Abaqus elements, ignoring local grid refinement and coarsening, and taking into consideration the void ratio and saturation fields. The element sets generated by the fromreservoir translator follow a specific naming convention. These element sets are maintained on the part upon reservoir creation and are subsequently used by the plug-in for section assignments, load assignments, and more. The set naming convention is as follows:

Layer_n:
Layer_n_Gas_Region_m
Layer_n_Oil_Region_m
Layer_n_Water_Region_m
Layer_n_Inactive

where n = Eclipse layer number
 m = equilibration region number

Since Abaqus is a single phase simulator, it is necessary to choose whether a cell belongs to gas, oil or water region according to the percentage of fluid saturation. In particular, the following criterion has been adopted and implemented in the translator in order to preserve as much gas and oil saturation as possible: an (i,j) cell is assigned to the water region if and only if its water saturation is equal to 1, otherwise it is gas or oil saturated depending on the biggest value between them.

3.3 Model and Reservoir Creation

Once the Eclipse odb is generated from the Eclipse output files, the reservoir model can be imported into Abaqus/CAE. Upon submission of the dialog shown in Figure 5, the RM imports the orphan mesh part from the Eclipse odb into the current model and creates an instance of the part into the assembly.



Figure 5. GUI for reading in the output database created by the translator.

3.4 Element Renumbering

Element renumbering is then performed in order to assist in the upscaling process. The merging of elements, either vertically or horizontally requires a decision to be made as to which element properties the merged element will inherit. The meshing tools in Abaqus/CAE always retain the lowest element number when merging elements. The variable that has been chosen as representative in the merging process is the fluid volume (FV), which is defined as follows:

$$FV = PV \cdot Sat$$

where PV = Pore Volume of the cell
 Sat = Gas / Oil / Water Saturation

The plug-in rennumbers elements so that elements with the most fluid volume are assigned the lower element numbers and will thus have their material properties assigned to the merged element.

3.5 Upscaling

A process of upscaling is necessary to improve the quality of the elements of the mesh. Upscaling involves the merging of entire rows of elements in the horizontal direction while maintaining the vertical discretization. This process is conducted based on the scheme specified in Table 1:

	J direction			
start	1	33	35	37
end	32	34	36	48
merges	4	2	1	2

Table 1-Horizontal upscaling.

Upscaling is performed using the Merge Layers item in the mesh edit tools. Selecting the icon  brings up the Edit Mesh dialog shown in Figure 6.

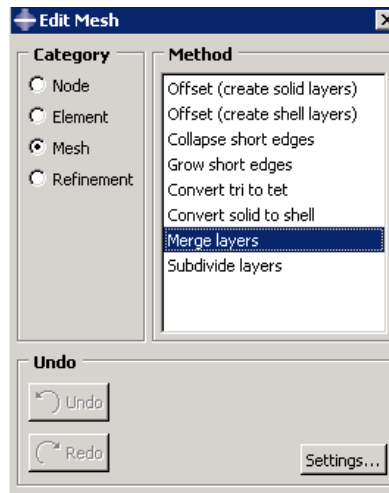


Figure 6. Snapshot of the GUI for performing mesh editing.

Using the same icon, the method 'Grow short edges' is used for adjusting some elements whose thickness was less than 2.2 cm.

The quality of the mesh is then verified; it is confirmed that no elements get flagged as erroneous. It is then possible to carry on with the creation of the burden regions.

3.6 Burden regions creation

The creation of burden regions is performed entirely outside of the plug-in. The Extrude method of the Create Bottom-Up Mesh dialog is used to extrude layers of elements from the element faces on the sides of the reservoir. The elements are extruded in a direction provided by the user until they intersect chosen planes. The chosen planes can be either datum planes or horizons. The number of layers comprising the burden regions and the associated bias can also be chosen.

For the Brugge field, datum planes are created to generate the side burden regions, and the option of extending the original sets is selected as shown in Figure 7. This option modifies any sets that have elements bordering the sides. The plug-in uses these modified sets to determine which parts of the burden regions are associated with each reservoir layer. It will then generate additional new sets which are then used for material property assignments. For this model, five layers have been created along the four directions with a bias of 5; this bias value leads to creation of elements whose length increases with increasing distance from the reservoir region.

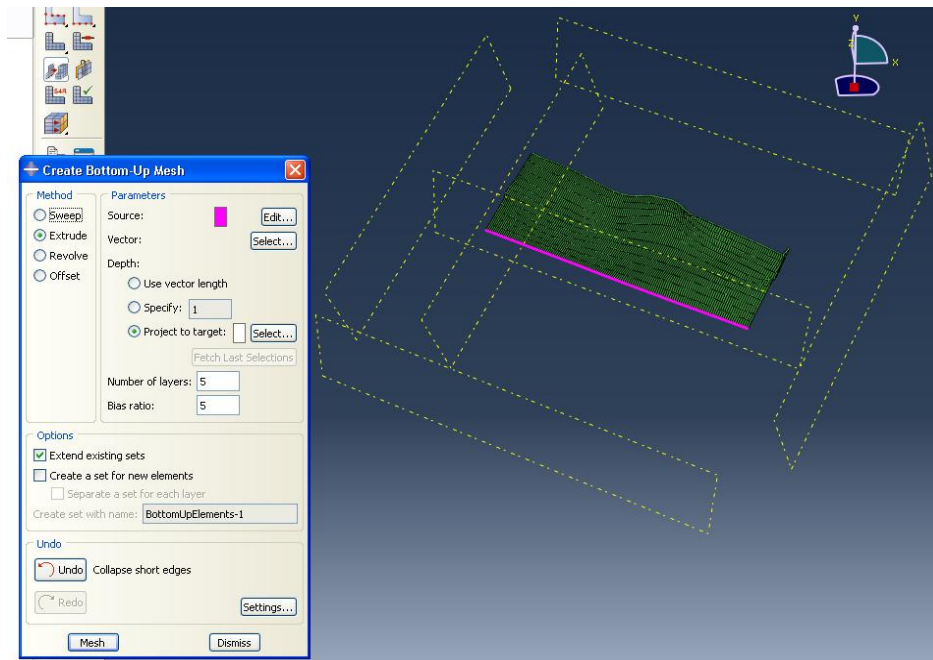


Figure 7. GUI for creating side burden regions.

The original Eclipse model covers an area of approximately $3 \times 17 \text{ km}^2$, while the final geomechanical model extends to $31 \times 31 \text{ km}^2$.

The original grid is then extruded vertically upwards and downwards so as to cover the sea-bed region at 50 m and up to a depth of 5000 m. The model is divided into 4 layers from the top of the reservoir up to the top surface (over-burden) and into 4 layers from the bottom of the reservoir up to the base (under-burden) using the same Create Bottom-Up Mesh dialog. In this case, the 'Extend existing sets' is turned off, and 'Create a set for new elements' is turned on. Using these

options, new element sets get created for the over-burden and under-burden layers. These sets will be used later to assign material properties to the over- and under-burden regions (§3.8).

The final geomechanical model results in 83425 elements and 91792 nodes for a total of 304576 degrees of freedom.

3.7 Boundary Conditions

The external nodes belonging to the side-burden and under-burden regions have been restrained against motion normal to the boundary surface.

Submodel boundary conditions have been used to constrain the pore pressure during the steps created for the soils analysis.

3.8 Burden Property Assignment

The next step after the grid creation is performed by selecting the Burden Properties option which brings up the dialog shown in Figure 8. This box has been filled in to assign properties to the over-, under-, and side-burden regions. The layer names associated with the side-burdens are automatically listed in the table. The sets associated with the over- and under-burdens have been entered by right-clicking in the table and selecting the Add Sets option. Upon submission of the burden dialog, new elements sets beginning with *RSV_* get created for all of the burden layers and reservoir regions.

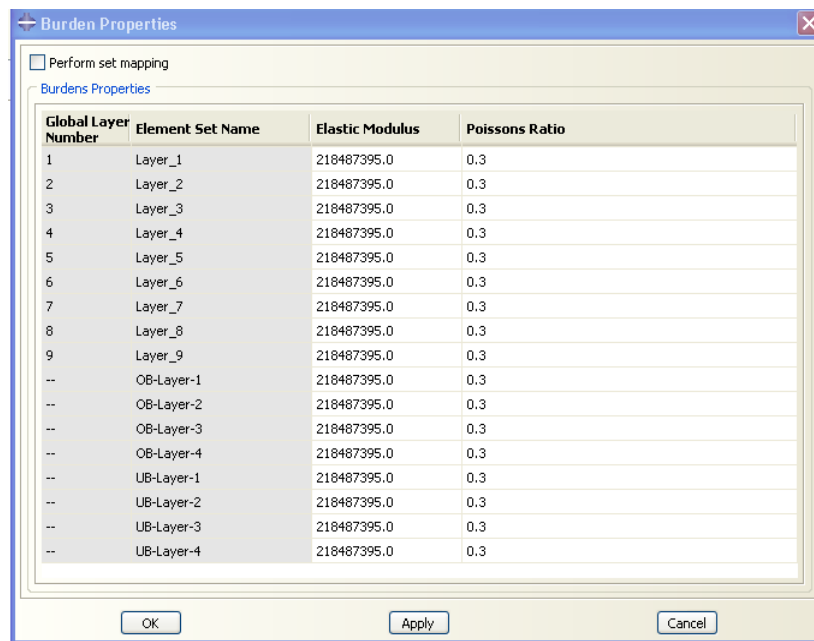


Figure 8. GUI for specifying elastic properties for the burden regions.

For the elastic simulation, a Poisson's ratio (ν) of 0.3 and a constant uniaxial compressibility (cm) of $3.40\text{e-}4 \text{ bar}^{-1}$ is considered. The elastic modulus (E) is then computed using the following formula:

$$E = \frac{(1 + \nu) \cdot (1 - 2 \cdot \nu)}{cm \cdot (1 - \nu)}$$

The value of $2.185\text{e}8 \text{ Pa}$ for E is then assigned to all the burden regions.

Considering plasticity, a compressibility value depending on the depth is computed for each layer and a resulting elastic bulk modulus is obtained. These initial values are reported in Table 2 and have been used to run the geostatic simulation. The new stress field obtained after the initialization will then be used to automatically update these moduli as explained in §3.18.

3.9 Density Assignment

The density required for an analysis within Abaqus is referred to as dry density and is defined as follows:

$$\rho_d = \rho(z) - \rho_f$$

where ρ_d = Dry density used in *Density definition
 ρ_f = Fluid density
 $\rho(z)$ = Bulk density

The density of the fluid, necessary to determine the overall density (dry density) that Abaqus requires for an analysis, is calculated as follows:

$$\rho_f = \frac{\phi \cdot \gamma_f}{g}$$

where ρ_f = Fluid density (varies element-by-element)
 ϕ = Porosity (varies element-by-element)
 γ_f = Specific weight of fluid (constant within an equilibration region)
 g = Gravitational constant

The specific weight of the fluid is stored on the Eclipse odb file on an element-by-element basis but with the same value for each element in an equilibration region. It is computed from the .DBG Eclipse file specified in the translator dialog box. The porosity, on the other hand, is stored on an element-by-element basis, consistent with the Eclipse values.

We assume the bulk density to vary based on depth only; it is supposed to be represented by the mathematical expression reported in Figure 9 where 50 m represents the depth of the seabed.

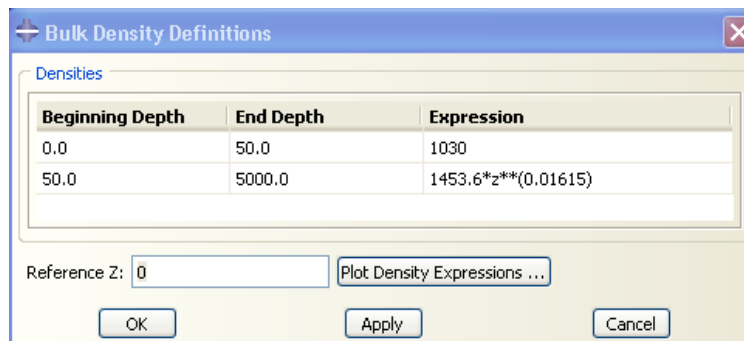


Figure 9. Bulk Density Definitions.

Upon submission of the Bulk Density dialog, a discrete field gets calculated using the bulk density values minus the fluid density values that is pulled from the previously described region-level discrete fields. The discrete field will thus contain densities defined on an element-by-element basis, and will be referenced in all of the material assignments through distributions.

3.10 Reservoir Region Properties

Selecting the Reservoir Properties item in the tree brings up the dialog shown in Figure 10.

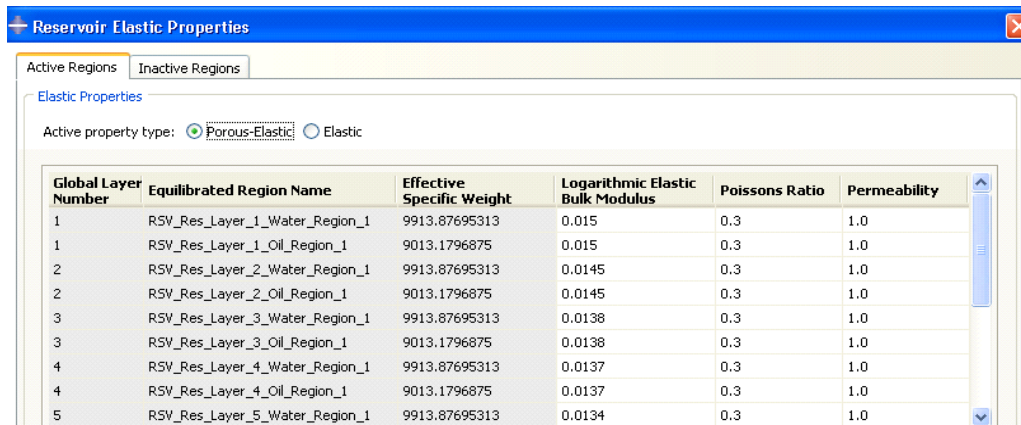


Figure 10. GUI for specifying porous elastic or elastic properties for the active regions.

Both for active and inactive regions, it is possible to choose whether to assign elastic properties or porous elastic ones according to the choice of the material behaviour.

An elastic simulation is run before the elasto-plastic one; the *ELASTIC keyword is chosen for both active and inactive regions. These regions are now characterized by an elastic modulus of 2.185e8 Pa, analogously to what is performed in the assignation of properties for burden regions described in §3.8.

For the plastic simulation, the *POROUS ELASTIC keyword is instead selected for the active regions. The logarithmic elastic bulk modulus is computed for each layer according to its average depth and vertical effective stress. The Young's modulus referring to each reservoir burden region is similarly calculated and inputted. Table 2 reports the values used for the elasto-plastic simulations.

It is important to underline that these properties will be recalculated after the reading of the geostatic results (§3.16) in order to compare the guessed moduli with the ones obtained after equilibrium is reached.

3.11 Pressure field computation

As already mentioned in §3.2, the translator converts finite volume results to an Abaqus output database file. This output database file can be used to apply Eclipse generated pore pressure field history to an Abaqus Standard analysis for performing a soils consolidation study.

To enable use in the submodeling procedure, the translator associates cell pressures with nodes. The nodal values of the pore pressure, $\hat{p}$, are obtained by averaging and combining pore pressures, p_i , in N cells adjacent to each node. Adjacency is determined according to the method specified for the grid translation and various methods can be chosen as well for controlling the averaging.

For the Brugge Field, the poreweight averaging method has been specified. With this method, the pore pressure in the output database is determined according to the expression:

$$\hat{p} = \sum_{m=1}^N p_m PV_m / \sum_{m=1}^N PV_m$$

where PV_m are the adjacent cell pore volumes.

Moreover, depending on the .xml configuration file, either relative or absolute pressure can be computed and written to the odb file as pore pressure history data. Relative values are used in the elastic simulation. They refer to the first frame field and so they are computed at time step t as follows:

$$\Delta \hat{p}_t = \hat{p}_t - \hat{p}_0$$

Absolute pressures are instead used in the plastic simulation.

In the present study, the initial pore pressure field is specified as the one obtained from the flow model initialization. It is also adjusted to make it consistent with the fluid contacts and the specific weight of the saturating fluid, both of which are computed from the data values in the .DBG Eclipse file for each equilibration region.

The pore pressure at any time step t is then calculated taking into consideration the initial pore pressure and the relative pressure at the time step specified.

3.12 Elastic simulation

The first FE calculations are performed under the hypotheses of a linear elastic behaviour of the material with homogeneous and isotropic mechanical properties.

Choosing the elastic as active property type, the GUI removes all the following items with the exception of the assignation of initial void ratio which is directly read from the odb file.

In the analysis, the pore pressure history is defined through submodeling techniques. The input file is written out, and the simulation is then run.

The results obtained in terms of iso-subsidence curves are shown in Figure 11 together with the results obtained with a semi-analytical solution (Geertsma and Van Opstal, 1973) using the same homogeneous and isotropic mechanical properties. It is evident that a very good agreement is achieved.

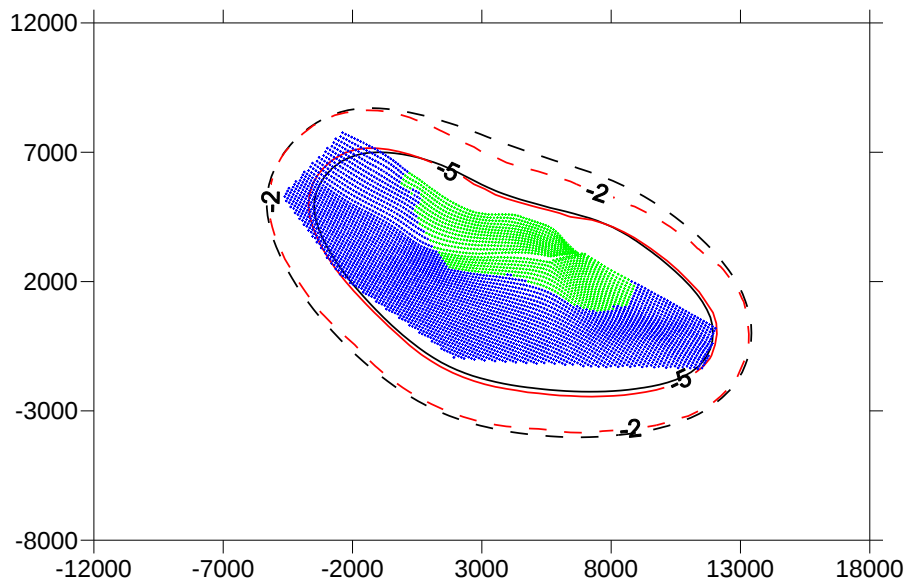


Figure 11-Iso-subsidence lines (in cm) for the elastic run at 2020: semi-analytical (black) and FE (red). The hydrocarbon saturated area of the first layer (green) and the surrounding aquifer (blue) are also shown.

3.13 Reopen odb

After the elastic simulation is performed, it is possible to maintain the grid and mesh geometry and run the elasto-plastic simulation using the option Reopen odb.

For this elasto-plastic analysis, one needs to use an odb file that is created using absolute pressure conversion instead of relative pressure conversion.

The values of the elastic moduli for burden regions are then changed as described in §3.8. The property type Elasto-plastic is selected for active regions and different values for each layer are assigned as explained in §3.10.

3.14 Initial Conditions

The initial conditions for void ratio, pore pressure and initial geostatic stress are applied by selecting Initial Conditions from the plug-in and filling in the dialog box as shown in Figure 12.

Void ratio and pore pressure values are directly read from the odb file created by the translator, while an initial linear stress is selected and applied for the geostatic simulation as explained in §3.15.

Initial Conditions

Eclipse Model ODB: C:/temp/Brugge_Eclipse.odb

Initial Pore Pressures

☐ Choose separate output file for pore pressures:

Odb File Name:

☒ Directly from .odb file ☐ Use mapping tools

Initial Void Ratios

☒ Directly from .odb file ☐ Use mapping tools

Initial Stresses

Vertical coordinate 1:

Vertical coordinate 2:

Stress 1 (active elements):

Stress 2 (active elements):

Stress 1 (inactive elements):

Stress 2 (inactive elements):

Lateral coefficient:

Figure 12. GUI for specifying initial conditions.

3.15 Geostatic Analysis

The geostatic procedure is used as the first step of a geomechanical analysis; gravity and other types of loads are applied during this step. Ideally, the loads and initial stresses should exactly equilibrate and produce zero deformations. In complex problems it may be difficult to specify initial stresses and loads that equilibrate accurately; consequently, the displacements corresponding to the equilibrium solution might be large.

An enhanced version of the geostatic procedure now allows for obtaining a stress distribution that results in small displacements. For this procedure we start by assuming an initial linear stress distribution specified previously. The UTOL parameter, which is an optional parameter in the *GEOSTATIC keyword invokes this new enhanced version. The value of UTOL is equal to the tolerance for maximum change of displacements. Abaqus/Standard ensures that the maximum absolute value of the displacement at a node is smaller than the tolerance times the characteristic element length in the model. If the UTOL parameter is used without any value specified, the default value of 10^{-5} is used. Moreover, the vertical stress (mainly governed by the density distribution and the gravitational acceleration) is maintained during the geostatic analysis; the horizontal stress components are then changed iteratively until the horizontal stress ratios are within the desired limits.

Figure 13 shows the final result in terms of vertical displacement for the geostatic analysis: a ratio of 0.54 has been specified and 5 iterations were needed to obtain the equilibrium and the final stress distribution.

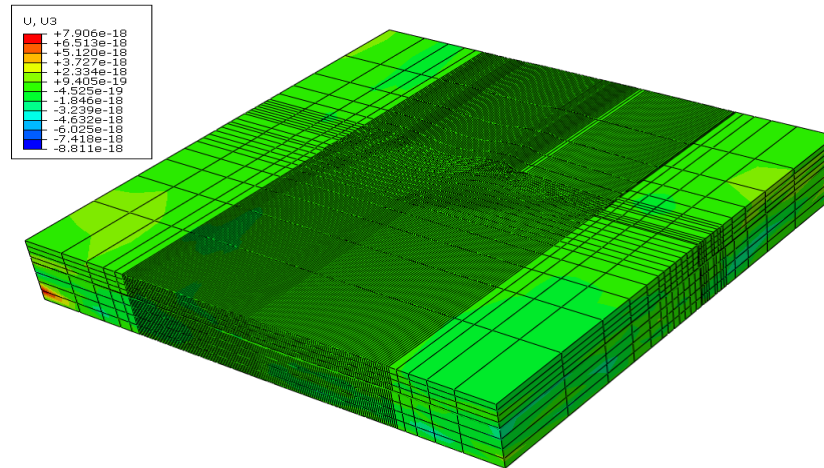


Figure 13 - Vertical displacement in the final geostatic analysis.

3.16 Geostatic Results

Upon completion of geostatic analysis, the resulting odb is read and the computed effective vertical stresses are then used to compute the plastic properties and to update the elastic ones.

3.17 Plastic Property Assignments

Selecting the Update Properties item, it is possible to define on a region-by-region basis the quantities that are required for the *CLAY PLASTICITY definition. Additional coefficients α and β are required to compute the compressibility as a function of the vertical effective stress σ . The relationship is as follows:

$$cm = \alpha \cdot \sigma^{-\beta}$$

The compressibility is then used to calculate the logarithmic plastic bulk modulus λ as follows:

$$\lambda = cm \cdot (1 + \varepsilon) \cdot \sigma$$

where ε refers to the element void ratio.

The computations of compressibility and logarithmic plastic bulk modulus are performed on an element-by-element basis, and then averaged over the entire equilibration region.

For the inactive reservoir regions and burden regions, the elastic modulus E is computed as follows:

$$E = \frac{(1+\nu) \cdot (1+2 \cdot \nu)}{cm \cdot (1-\nu)}$$

The logarithmic elastic bulk modulus K is then computed by the relationship: $K = \lambda / 3$. The values used for the plastic simulation are reported in Table 2:

	Elastic modulus [Pa ⁻¹]	Logarithmic Elastic Bulk Modulus	Logarithmic Plastic Bulk Modulus	Compressibility Coefficient α	Compressibility Coefficient β
Layer 1	1.95E+08	0.015	0.045	0.262	1.122
Layer 2	2.01E+08	0.015	0.044	0.277	1.127
Layer 3	2.06E+08	0.014	0.042	0.293	1.131
Layer 4	2.12E+08	0.014	0.041	0.310	1.136
Layer 5	2.18E+08	0.013	0.040	0.328	1.141
Layer 6	2.25E+08	0.014	0.043	0.348	1.146
Layer 7	2.32E+08	0.014	0.042	0.368	1.151
Layer 8	2.40E+08	0.014	0.041	0.390	1.156
Layer 9	2.48E+08	0.012	0.036	0.420	1.163
Over-burden 1	1.73E+08			0.020	0.898
Over-burden 2	1.55E+08			0.030	0.932
Over-burden 3	1.40E+08			0.068	1.005
Over-burden 4	1.28E+08			1.741	1.286
Under-burden 1	2.97E+08			0.032	0.938
Under-burden 2	3.71E+08			0.029	0.931
Under-burden 3	4.95E+08			0.034	0.945
Under-burden 4	7.43E+08			0.049	0.977

Table 2- Values used for the plastic simulation.

3.18 Property Overrides

The logarithmic elastic and plastic bulk moduli are then recomputed using the vertical stress obtained after the geostatic step. By clicking on the Update Properties in the tree and then selecting the Override Moduli option, the user can compare the values assigned before and after the geostatic step in order to investigate the effects due to the stress field obtained to reach the equilibrium. Moreover, the user has the ability to override the properties manually; the user can retain the initially assigned values, or can reassign other different values.

The results obtained for the Brugge field are not too different with respect to the moduli assigned before the geostatic step; for a comparison, some of them are shown in Figure 14.

Logarithmic Bulk Moduli Overrides				
Elastic & Plastic Bulk Moduli				
Global Layer Number	Equilibrated Region Name	Original Log Elastic Bulk Modulus	Computed Log Elastic Bulk Modulus	Computed Plastic Bulk Modulus
1	RSV_Res_Layer_1_Water_Region_1	0.015	0.0144438060429	0.0437691092208
1	RSV_Res_Layer_1_Oil_Region_1	0.015	0.0147270747419	0.046274992179
2	RSV_Res_Layer_2_Water_Region_1	0.0145	0.0141108246028	0.0427600745538
2	RSV_Res_Layer_2_Oil_Region_1	0.0145	0.0144640483475	0.0438304495378
3	RSV_Res_Layer_3_Water_Region_1	0.0138	0.013629507367	0.0413015374757
3	RSV_Res_Layer_3_Oil_Region_1	0.0138	0.0140721616631	0.0426429141307
4	RSV_Res_Layer_4_Water_Region_1	0.0137	0.0132686494751	0.0402080287123
4	RSV Res Layer 4 Oil Region 1	0.0137	0.0137940369152	0.0418001118643

Figure 14 - Comparison between the original and the computed logarithmic elastic bulk modulus.

The computed moduli have therefore been maintained. The plastic simulation is then run at different time steps; Figure 15 shows some results obtained in terms of vertical displacement.

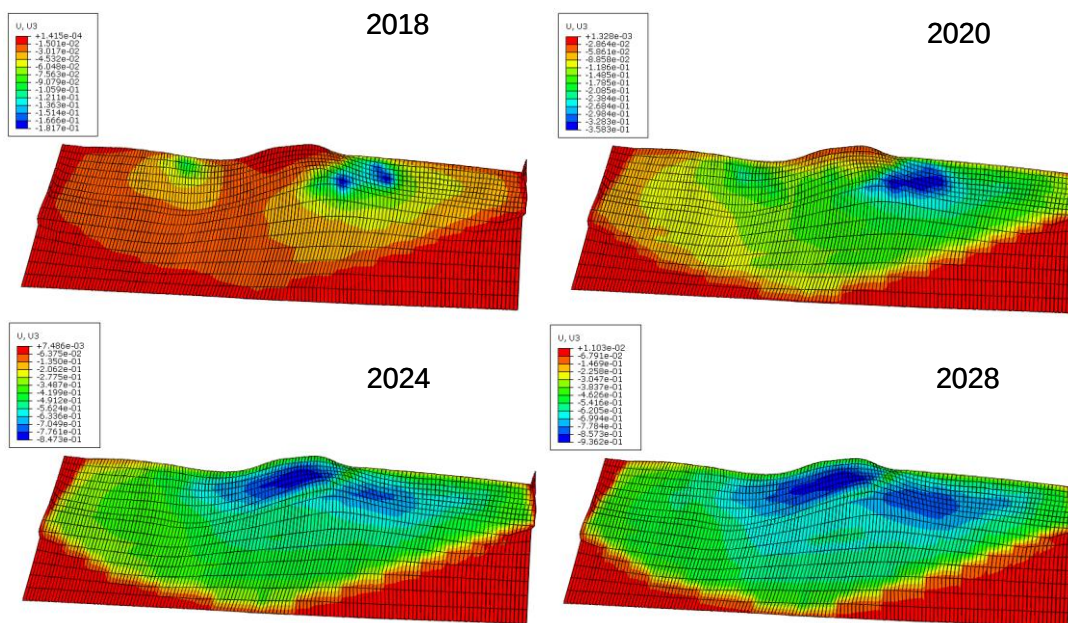


Figure 15 - Vertical displacement at the top of the first reservoir layer at different years.

4. Conclusions

This paper describes how the new capabilities of Abaqus 6.11 in combination with the RM plug-in have allowed for an automation of the workflow used to perform a geomechanical analysis on a field scale model directly in Abaqus/CAE.

In particular, the following new functionalities in Abaqus 6.11 have been used:

- The translator which establishes a link between Eclipse and Abaqus/CAE. All the information from the reservoir model in terms of grid, properties and pressure are transmitted to the FE model, which is then populated by these data in a completely consistent way.
- The meshing tools for adjusting elements and nodes where needed and for performing upscaling. Moreover the burden region creation was properly done through these capabilities.
- The automated geostatic procedure.
- The assignation of material properties and load history automatically within Abaqus/CAE to run both the elastic and elasto-plastic simulations

In addition to these new capabilities, the GUI allows for following a logical scheme in order to easily and automatically executing all the steps required in the creation of a geomechanical model. The possibility to clearly have a sequence of steps to be performed significantly improves the efficiency in terms of user time of the preprocessing phase needed to build a full field geomechanical model.

The new capabilities implemented in Abaqus/CAE and the RM plug-in have then shown to be particularly appropriate for reservoir geomechanical simulations. The analysis can now be more proper and precise, and the automation allows for shortening the time required to setup and run reservoir geomechanics simulation models.

5. References

1. Abaqus User's manual 2011.
2. Boot, R., "Level Control Surveys in the Groningen Gas Field", *Verhandelingen Kon. Ned. Geol. Mijnbouw. Gen.*, Vol. 28, pp. 105-109, 1973.
3. Bruno, M. S., "Subsidence-Induced Well Failure", *SPE Drilling Engineering*, 1992.
4. Capasso G., Mantica S., "Numerical Simulation of Compaction and Subsidence Using Abaqus", *AUC*, 2006.
5. Da Silva, F. W., Debande G. F., Pereira C. A., and Plischke B., "Casing Collapse Analysis Associated With Reservoir Compaction and Overburden Subsidence", *SPE* 20953, 1990.
6. Floris, F.J., Bush, M.D., Cuypers, M., Roggero, F., and Syversveen, A.R., "Comparison of Production Forecast Uncertainty Quantification Methods - An Integrated Study", paper presented at 1st Conference on Petroleum Geostatistics, Toulouse 1999.

7. Geertsma, J., "A Basic Theory of Subsidence due to Reservoir Compaction: the Homogeneous Case", *Verhandelingen Kon. Ned. Geol. Mijnbouwk. Gen.*, Vol. 28, pp. 43-62, 1973.
8. Geertsma J. and Van Opstal G., "A Numerical Technique for Predicting Subsidence Above Compacting Reservoirs Based on the Nucleus of Strain Concept", *Verhandelingen Kon. Ned. Geol. Mijnbouwk. Gen.*, Vol. 28, pp. 63-78, 1973a.
9. Mindlin R.D. and Cheng D.H., "Thermoelastic Stress in the Semi-Infinite Solid", *J. of Applied Physics*, Vol. 21, p. 931-933, 1950.
10. Ostermeier, R. M., "Deepwater Gulf of Mexico Turbidites – Compaction effects on Porosity and Permeability", *SPE* 26468, 1995.
11. Schlumberger, "Eclipse Reference Manual 2009", 2009.
12. The BRUGGE Field, <http://www.isapp.nl/brugge-field-case>
13. Terzaghi, K., "The Shearing Resistance of Saturated Soils and the Angle Between the Plane of Shear", *Proc. of 1st Int. SMFE Conference, Harvard Mass.*, Vol.1, pp. 54-56, 1936.
14. Zaman, M. M., Abdulrahheem, A. and Roegiers, J. C., "Reservoir Compaction and Surface Subsidence in the North Sea Ekofisk field", *Subsidence due to fluid withdrawal*, Elsevier Science, pp. 373-419, 1995.

6. Acknowledgments

The two years fruitful and productive cooperation with Simulia US and Simulia Italia allowed for developing the work presented in this paper. The authors are very grateful to all the developers from Simulia who took part on this project implementing new capabilities in the code to enable this new workflow. In particular, Jeff Haan and his team for developing the translator, Matthew Rees and Konstantin Kovalev for creating new mesh editing tools and Mike Shubert for creating the Reservoir Modeler Extension. The contribution of each of them has been necessary and fundamental for the good outcome of the project.