

Finite Element Analysis in Offshore Geotechnics – A Thirty-Year Retrospective

J. S. Templeton, III

SAGE USA, Inc.

Keywords: Critical State Plasticity, Dynamics, Earthquakes, Experimental Verification, Foundations, Geo-mechanics, Jack-up Rigs, Offshore Structures, Piles, Soils, Soil Mechanics, Soil-Structure Interaction, Subsidence, Well Conductors

1. Abstract

This paper reviews, from the author's particular perspective, the history of the application of finite element analysis in geotechnical engineering for the offshore over the past 30 years, with emphasis on the unique and dominant role of Abaqus analysis in that history.

The paper covers foundation performance issues and applications for pile-founded, gravity base, and deep water structures. Applications to both exploration and production structures are treated. Many of the significant applications were complemented by extensive experimental test programs to confirm or validate the analyses.

Example analyses discussed include loading from waves and earthquakes as well as subsidence effects. The examples include applications of a wide variety of Abaqus element and material types as well as use of procedures for static, dynamic and soil consolidation analysis.

The paper discusses the application of Abaqus' geotechnical capabilities throughout their development and into the present, and it provides suggestions for further development of new capabilities to meet emerging needs.

2. Background and Introduction

The use of Abaqus for finite element analysis of offshore geotechnical problems began in 1981. By that year the finite element method was not new to this author, nor was it new to geotechnical analysis. Geotechnical analyses were among the earliest applications of the finite element method. According to Cluogh and Wilson (1999), their 1962 paper on the Norfolk Dam analysis was only the second paper ever published with the name, "finite element," in the title. The work reported in that paper included continuum finite element analysis of not only the dam but also the foundation. This author first engaged in finite element analysis in 1966, performing structural dynamic analysis for Saturn-Apollo space vehicles and first became acquainted with Critical State Soil Mechanics only after a subsequent return to full time graduate study at Tulane University, where

he joined a group engaged in research on mechanics of granular materials under Professor S. C. Cowin in collaboration with a group doing similar work at Cambridge University under Professor P. N. Bransby.

Throughout the 1960s and 1970s a large number of papers were published involving the use of the finite element method for solution of geotechnical problems. Smith (1982) in his text, *Programming the Finite Element Method with Application to Geomechanics*, cites dozens of important papers from that period on methods and applications of finite element for geotechnical work. Most of the early geotechnical analyses were done with special purpose programs, academic programs or “in house” programs, and while some involved nonlinear effects, most of these were accommodated within small strain theory or via modifications of programs originally coded for linear solutions. While some early analyses included plastic yield, these were typically elastic-perfectly plastic analyses with small strain or small strain-derived solutions.

A series of developments from the late 1960s through 1980, however, would combine to remove prior restrictions and set the stage for an entirely new era in geotechnical finite element analysis:

1. Roscoe and Burland (1968) developed a modified Cam Clay theory within the framework of Critical State Soil Mechanics. The original Cam Clay theory (see Schofield and Wroth, 1968) had included a logarithmic interaction between deviatoric and pressure stresses at yield. The modified theory replaced this with an elliptical form, thus indicating more realistic behavior for stress states near isotropic pressure and providing a yield surface with a form more amenable to incorporation in a complete plasticity formulation and suitable for use with the finite element method.
2. Hibbitt, Marcal and Rice (1970) established the basis for a rigorous approach to large strain formulation for finite element analysis. This work was further elucidated by McMeeking and Rice (1975) who demonstrated that the common approximate approach to accommodating large strains in finite elements (that of updating coordinate locations obtained from small strain theory) can result in significant error unless stress increment and stiffness corrections are also made.
3. Nagtegaal, Parks and Rice (1974), studied the problem of shear locking of finite elements under large deformations (a particularly serious problem for nearly incompressible material behavior such as is typical of soils under undrained conditions). They concluded that none of the element formulations existing at the time were free of this problem and they developed a new formulation for elements with selectively reduced integration and provided a demonstration that this new formulation could be properly extended to give proper results for finite strains. This formulation completely resolved the problem of shear locking with near zero volume change.
4. Critical State Soil Mechanics began to see important and productive use in numerical solutions of important foundation analysis problems in the offshore oil industry. (See for example, Miller, Murff and Kraft, 1978, and Carter, Randolph and Wroth, 1979.)
5. In the late 1970s, Hibbitt, Karlsson and Sorensen, Inc. produced Abaqus, the first fundamentally nonlinear, commercial general purpose finite element program. Notably, the program incorporated a rigorous large deformation capability based on the work of Hibbitt,

Marcal and Rice (1970) and of McMeeking and Rice (1975) and it included the selectively reduced integration elements of Nagtegaal, Parks and Rice (1974).

6. Nystrom (1980) was the first to propose incorporation of Critical State Soil Mechanics and effective stress solution capability into a commercial general purpose finite element program. Being familiar with the emerging use of Critical State Soil Mechanics for foundation analysis in the offshore industry and with the existing capabilities of the Abaqus program, Nystrom specifically proposed that the Modified Cam Clay theory of Critical State Soil Mechanics, along with a capability for solving fully coupled effective stress and pore water flow problems be incorporated into the Abaqus program. (See also Nystrom, 1984a.) As described by Potvin (1990), this incorporation was accomplished as part of a joint undertaking by Hibbitt, Karlsson and Sorensen, Inc. and Exxon Production Research Company.

In late 1980, this author joined the offshore geotechnical staff at Exxon Production Research Co., just in time for the early 1981 release of the first Abaqus version with the new soil mechanics capabilities, and as leader of the EPR offshore Soil Mechanics group he had the privilege of championing and experiencing several early improvements, including yield surface extensions to the modified Cam Clay, integration of the large deformation and Critical State capabilities and incorporation of the extended Drucker Prager yield condition.

Immediately from their initial availability in 1981, the Abaqus soil mechanics capabilities saw important application to problems in geologic processes, pile capacity, and performance of gravity base structures, although the products of these initiatives did not begin to see the light of publication until the mid-1980s.

3. Geologic Processes, Consolidation and Subsidence

The first publication to report on use of Abaqus for analysis of a geologic problem was that of Templeton et al. (1985). That paper describes a historically comprehensive inter-disciplinary study undertaken to clear obstacles to development in the complex geological environment of the deep water Gulf of Mexico beyond the continental shelf. The effort marked the offshore industry's first geotechnical project to employ integration of geotechnical, geologic and geophysical studies to provide proper interpretation of complex deepwater site conditions. A key element of the integration was the use of Abaqus finite element analysis to hindcast the entire geologic history of soil deposition and consolidation of the soils in the Gulf of Mexico's Mississippi Canyon. The finite element analysis required the use of large strain theory in combination with a coupled pore water flow and effective stress response solution via Critical State theory and Modified Cam Clay Plasticity. Site investigation data had shown that the soils in the canyon fill had only a small fraction of the strength of similarly constituted soils on the outer continental shelf. The results of the Abaqus analysis, which agreed well with geologic data and with alternate solution methods, showed conclusively that the low strengths were the result of an "under-consolidated" condition owing to rapid deposition of the canyon fill. This was part Exxon's work on the Zinc Project which received the outstanding achievement award from the Offshore Technology Conference, and the Templeton et al. (1985) paper was the recipient of the ASCE Hall of Fame Award for Offshore Technology in Civil Engineering.

Templeton, Little and Sraders (1991) reported on Abaqus-based finite element analysis of subsidence effects on offshore structure foundations. Using results from that analysis, Little, Moore and Templeton (1991) produced the first published application of continuum finite element analysis to determine design criteria for the effects of subsidence on the foundations of offshore structures. The work was a major contribution to Freeport MacMoRan's pioneering offshore sulfur production project in the Gulf of Mexico. That project was the recipient awards for outstanding project of the year from both the Offshore Technology Conference and the National Society of Professional Engineers.

4. Gravity Base Structures

The first application of the Critical state Soil Mechanics capabilities in Abaqus to a problem in the performance of a Gravity Base Structure (GBS) foundation was the work of P. K. Leung in the early 1980s. In that work, as described by Potvin (1990), an idealized model of a portion of a GBS base with skirts attached was analyzed in Abaqus for its ability to resist horizontal load, using coupled pore pressure / effective stress analysis and Modified Cam Clay theory. A small scale physical model of the same foundation unit was subjected to testing in a centrifuge at the University of Manchester Institute of Science and Technology (UMIST). Centrifuge testing is an important tool in experimental soil mechanics because it is the only practical way to achieve proper similitude for effects of gravity stresses in small scale tests of soil foundations. This work provided the first cross-validation of experimental and numerical foundation analyses to be accomplished using Abaqus analysis and centrifuge testing. It set the stage for many other important comparisons of results from Abaqus analysis and Centrifuge testing to come in following years.

Clukey and Templeton (1987) reported on the application of Abaqus analysis to the prediction of foundation consolidation for the Concrete Island Drilling Structure in the Beaufort Sea. This gravity structure was to be floated into position and placed on the seafloor during ice free summer conditions and then de-ballasted and allowed to achieve self weight consolidation strengthening of its foundation as the winter ice began to form. The Abaqus analysis (using coupled pore pressure / effective stress analysis and Modified Cam Clay theory) successfully showed that sufficient consolidation strengthening could occur in time to produce the foundation capacity that would be required for resistance of winter ice loads. The Abaqus analysis reported in this paper was the first published use of effective stress finite element analysis to determine the time necessary for self-weight consolidation strengthening of an offshore gravity base foundation.

5. Axial Performance of Driven Piles

The previously mentioned work of Miller, Murff and Kraft (1978), as well as that of Carter, Randolph and Wroth (1979), analyzed the problem of soil consolidation and setup around driven piles via one dimensional analysis. The one-dimensional problem solution by Carter, Randolph and Wroth (1979) became one of the original test problems for the Abaqus Critical State Soil Mechanics capabilities. Very good agreement was achieved with the Carter, Randolph and Wroth solutions, and this test problem survived as an important example which served well to introduce

many new Abaqus users to finite element effective stress analysis. Nystrom (1984b) extended this analysis to a 2-D axisymmetric analysis in order to include ground surface boundary effects and to permit axial loading of the pile after consolidation and setup. Nystrom's (1984b) analysis is the first known published complete 2-D effective stress finite element analysis of pile foundation consolidation, setup and subsequent axial loading.

As the oil industry moved into deeper waters in the late 1980s and early 1990s, the diameters of foundation piles grew and the ratio of pile diameters to wall thickness increased. This presented the expectations that soil disturbance during pile installation as well as subsequent re-consolidation and setup could each be greater or less than in previous practice, but whether the net result of these competing effects would be beneficial or detrimental was unknown, and the pile diameters and wall thickness ratios were outside of the range of the offshore industry's empirical data base on pile capacity. Bogard and Templeton (1992) addressed this question via review of historical pile setup data and with new analyses which made full use of Abaqus' large deformation capabilities to extend the range of the prior work. This was a research study supported by most major oil companies. The reported results from the analyses compared successfully to all of the historically published data on pile setup in clay and provided a new method for the prediction of setup of capacity for driven piles applicable to a wider range of pile diameters and wall thickness ratios than previously justified.

Abaqus analysis has also contributed significantly to the understanding of pile performance in sand. Hamilton and Templeton (1994) reported on a comprehensive study of axial capacity of piles in sand for the American Petroleum Institute (API). The results of the study provided resolution of serious questions that had been raised at the time regarding the accuracy of the method for calculation of pile capacity in sand given in the API's Recommended Practice, API-RP-2A. This study report included details of analysis and results from Abaqus analyses of the performance of piles under axial load in sand. Khan, Templeton and O'Neill (1994), reporting separately on the Abaqus results from that study, presented plots of skin friction along the length of the pile at various levels of pile head displacement. These results shed important new light on the progressive development of skin friction profiles with displacement and provided for the first time a theoretically based explanation for the existence of limits on pile skin friction in sand such as had been (contentiously) prescribed in API-RP-2A. This paper was one of the finalists for the ASCE's outstanding paper of the year award.

6. Suction Piles and Caissons

As offshore oil production structures have moved into still deeper waters in the period from the 1990s and into the present, there has been an ever increasing use of suction foundations. Suction piles (often called suction caissons, particularly when diameter is a large fraction of the embedment depth) are piles installed by suction forces rather than by driving. Suction installation becomes increasingly more attractive with increasing water depth because pile driving becomes increasingly more expensive and because greater water depth provides greater hydrostatic pressure and so also greater available suction. Abaqus has played an important role in connection with the use of suction piles via both total stress analysis and coupled pore water pressure / effective stress analysis. Effective stress analysis is essential to the determination of time-based pore pressure

diffusion effects on long term performance, and total stress analysis can provide better definition of response to short term loading, particularly when displacement and rotation results are important in addition to capacities.

Clukey and Morrison (1993) provided the first reported analysis of suction foundation performance via Abaqus-based effective stress analysis as well as results from centrifuge test programs. The degree of consistency of the Abaqus results with those from centrifuge testing increased confidence in both the experimental and analytical results.

Templeton (2002) provided a comprehensive discussion of the role of (total stress) finite element analysis in suction foundation design. The Abaqus-based work reported there included results of proprietary work from several sources as well as government and industry sponsored research and academic workshop related work. One interesting result included was an example of suction pile performance including consideration of separation of the soil from the pile wall (Figure 1). At the time there had been considerable industry concern about the possible effects of such separation on capacity. Abaqus analysis results showed that these effects would be quite small, negligibly so in deep water. Reported results also included Abaqus-based large strain results verified by alternate software and distinctly different than small strain results (Figure 2). Raines, Ugaz and Garnier (2005) reported on a very satisfactory comparison between Abaqus analysis results and results from a corresponding centrifuge test. The Abaqus analysis was performed using a derivative of the same finite element model used for the work of Templeton (2002). Displacement results from the analysis and test were in remarkable agreement. Significantly, the centrifuge test results were produced blind of the Abaqus analysis results.

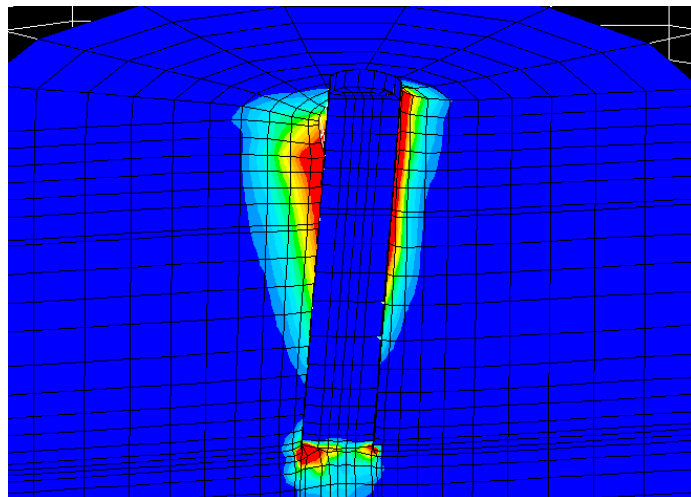


Figure 1. Contours of plastic strain for caisson with inclined load and separation.

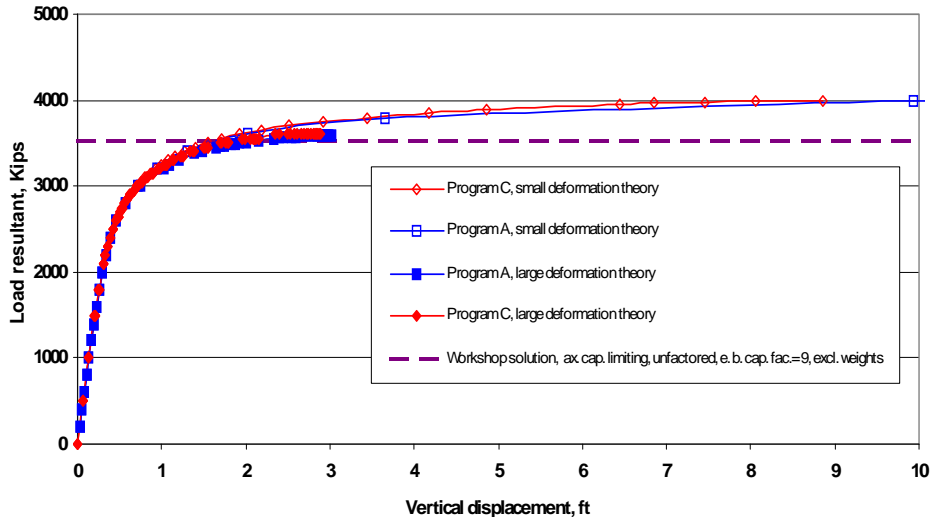


Figure 2. Load vs. displacement results for inclined load on suction caisson/pile, Abaqus (program A) results compared to results from comparison program (C).

Al-Khafaji et al. (2003) reported on Abaqus analysis of suction caisson foundation performance under vortex-induced vibration (VIV) loading. This was the first publication of any analysis of offshore foundation performance under VIV loading conditions, and the first publication of Abaqus analysis of foundation performance including coupled effective stress / pore water pressure effects under cyclic loading.

Clukey et al (2004) reported on application of numerical and experimental analysis to the problem of determining the long term ability of suction piles to maintain reverse end bearing capacity for resistance of long term uplift loads. This paper included results of Abaqus coupled pore pressure / effective stress analysis of suction caisson performance as well as results from several centrifuge test programs. Following are details of the modeling and analysis:

- A continuum finite element mesh was constructed to represent an 18-ft diameter suction caisson with an embedment (penetration) of 96 ft. The model was symmetric about the caisson's vertical axis.
- The caisson was modeled as an elastic material with Young's modulus and Poisson's ratio representative of steel using 8-node axi-symmetric continuum elements.
- The soil inside and outside the caisson was modeled as an elastic-plastic porous material using the Extended Modified Cam Clay constitutive model and 8-node axi-symmetric continuum elements with pore pressure. (The original yield surface in the Modified Cam Clay model is an ellipsoid in principal stress space. It is elliptical in the meridional plane and

circular in the Pi plane. The extended model used in this study allows a yield surface that is non-circular in the Pi plane.) With this model, elastic changes in volume of soil occur when the mean effective stress changes, and plastic changes in volume of soil occur when the size of the yield locus changes. The pore fluid flow in the model is governed by a diffusion law. The pore water flow behavior and the effective stress behavior are fully coupled through the effective stress principle.

- Material properties were assigned to the soil elements using measured soil properties based on data from a representative clay soil deep water site in the Gulf of Mexico.
- The response of the system to an upward directed load on the caisson was determined by performance of the soils consolidation procedure in Abaqus/Standard.
- Several different simulations were made. In a parametric study all caisson and soil parameters were held constant while the applied sustained load was varied. Each parametric value of sustained load was based on a percentage of the ultimate undrained static uplift capacity that had been determined by applying a monotonic load rapidly (approximately ¼ hour) in a separate simulation. In that analysis, the load achieved at an uplift displacement equal to 10% D was recorded as the ultimate undrained capacity (for short term loading).

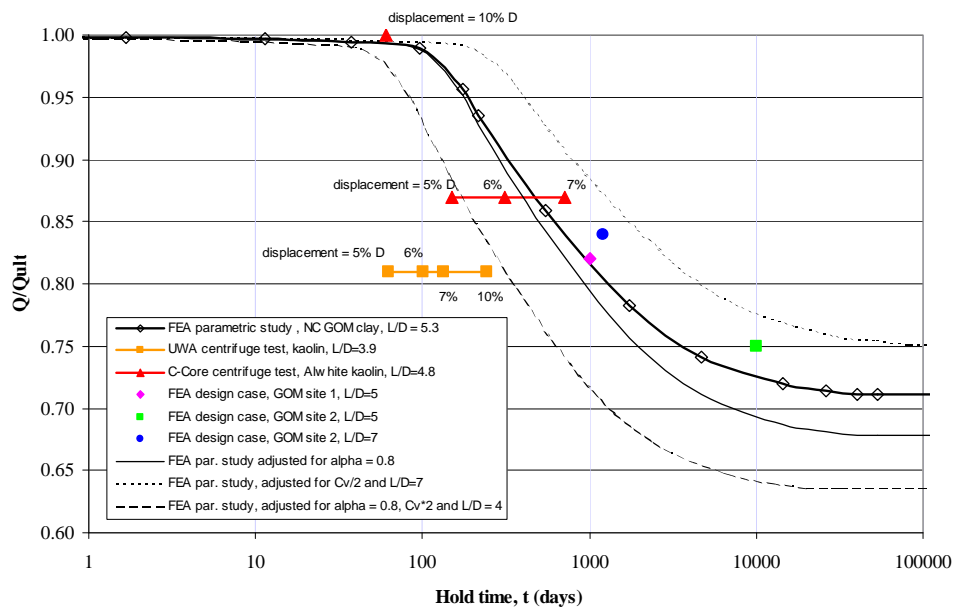


Figure 3. Capacity (normalized) vs. hold time for long term upward loading of suction caisson, compared to centrifuge test data and selected design cases.

- For each parametric value of sustained load, the time history of displacement for the sustained load simulations was observed, and the maximum hold time was recorded either when pullout failure occurred or a displacement of 10% of the diameter was achieved. Hold times were recorded ranging from 2 days to over 150 years. Capacity was not reduced from the short term value for hold times less than 10 days. Complete reduction of capacity to its long term value required hold times on the order of 100 years.

A summary of results from this study is presented in Figure 3. The Abaqus results were not only consistent with the centrifuge test data, they successfully explained the relationships between the data from disparate test programs and provided a basis for interpolation and extrapolation to conditions not tested. Prior to the publication of this paper there was divided opinion in the offshore engineering community as to whether or how to allow for reverse end bearing capacity of suction piles and caissons. Presently, the Clukey et al. (2004) paper is the definitive industry reference on the subject.

7. Lateral Soil Interaction with Piles and Well Conductors

Lateral soil interaction with piles and well conductors has traditionally been determined by a procedure known as p-y analysis. API RP-2A prescribes “p-y curve” formulae for this use based on the pioneering work in that area by Matlock (1970). Although some published experimental studies have shown indicated substantially greater stiffnesses and capacities than those in the API curves, offshore platform designers have long been content to use the traditional API curves for piles because the resulting conservatism had acceptably small consequence. In the case of well conductors, however, the result could be unacceptable levels of costly over-conservatism, particularly for cases in which the lateral soil-conductor interaction has large effects on the dynamic response of deep water risers in fatigue studies.

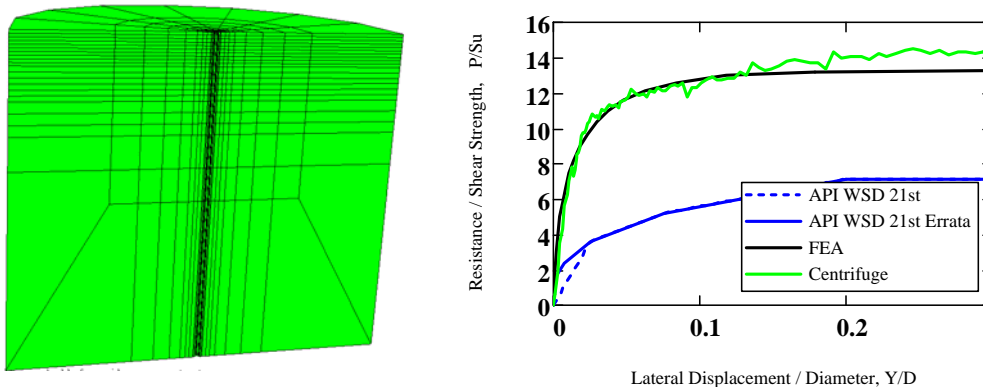


Figure 4. Abaqus mesh (left) for analysis of lateral well conductor/soil interaction and Abaqus results (right) for resistance vs. displacement compared to results from centrifuge testing and to indications of API (RP-2A).

Jeanjean (2009) and Templeton (2009a) reported on matched experimental and finite element studies of lateral soil interaction with well conductors in clay. The experimental work was accomplished via centrifuge testing, and the finite element analyses were performed using Abaqus analysis (Figure 4). The Centrifuge testing and finite element analyses were done completely double blind, and the degree of agreement between their results was remarkable. Both the stiffnesses and the ultimate capacities from the finite element analysis results were in quite good agreement with the centrifuge test data all along the length tested, and all of these stiffnesses and capacities were substantially greater than the values per API RP-2A.

The finite element approach of Templeton (2009a) to determination of lateral soil/well conductor interaction is presently under consideration as a referenced method in the new API recommended practice for offshore geotechnical work, and “enhanced” p-y curves based on Jeanjean (2009) are seeing increasing use for lateral soil interaction with well conductors and foundation piles.

8. Jack-up Rigs

Jack-up rigs are mobile offshore units that are floated onto location and then founded on the seafloor via legs that are jacked downward from the hull to embed in the seafloor and then to elevate the hull above the water. There are two broad types of jack-up rigs, independent leg rigs which have separate foundation units (called spud cans) and mat supported rigs which have all legs connected to a single foundation unit (or mat). Jack-ups are primarily used as temporary drilling units, but are sometimes converted to long term service as production platforms.

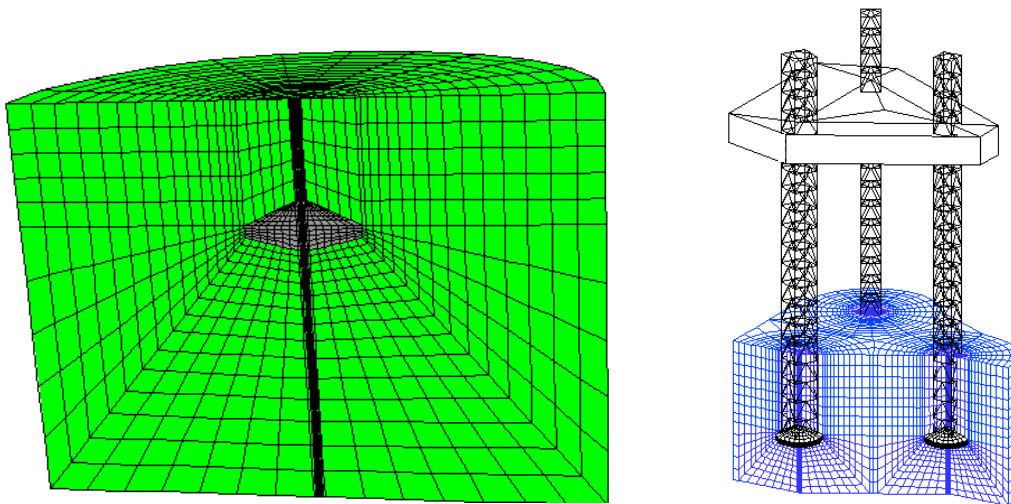


Figure 5. Models for jack-up rig analysis range from continuum models of soil around a single spud can (left) to models including complete jack-up rig with leg details and continuum representation of the foundation soils (right).

Whenever a jack-up is moved to a new location, a new assessment is required. Abaqus analysis has seen important use in performance of and in development of methods for these assessments, for both mat rigs and independent leg rigs, and for both temporary and long term applications. The Abaqus work has ranged from static analysis of foundation performance with single spud can models to 3-D, nonlinear, time domain dynamic analysis of full rig models (Figure 5).

Site assessment methods for independent leg rigs have traditionally been governed by the Society of Naval Architects and Marine Engineers (SNAME) Bulletin 5-5. Abaqus has, since the late 1990's, provided a spud can type for its Elastic-Plastic Joint Element, and the spud can foundation behavior prescribed for that element was based on the same work as used in the original issue of Bulletin 5-5 (see Van Langen, Wong and Dean, 1999). Wong et al. (2012) describe the new foundation assessment provisions of ISO 19905-1 for site assessment of mobile offshore units. These new provisions were developed to replace the earlier provisions of SNAME Bulletin 5-5 with new and significantly more realistic methods. The new methods were the product of a decade of research sponsored by the International Association of Drilling Contractors (IADC), and most of the important changes were supported by Abaqus based finite element analysis. Templeton, Lewis and Brekke (2003) describe combination of Abaqus analysis of spud can performance with data from SAGE Engineering's instrumentation of the Adriatic III rig during Tropical Storm Josephine to justify recommendations of a factor of 3 increase in the soil modulus recommendations, increases of up to 75% in moment capacity for deep penetrations and improvements in the form of the yield interaction formula. Templeton, Brekke and Lewis (2005) used additional Abaqus analysis results to further develop these recommendations and also to recommend specific allowance for foundation hysteretic damping. Templeton (2006) added specific recommendations for calculation of the recommended hysteretic damping. Templeton (2009b) used Abaqus analyses with the same spud can model as basis for significant increases in horizontal capacity recommendations for deep penetrations. In the end, critical validation for all of these improvements was provided by comparisons of hindcast Abaqus analysis results to the actual observed performance of the Adriatic III rig during Hurricane Rita (Figure 6).

Templeton, Lewis and Brekke (2009) reported on the comparison for total response via comparison to Abaqus nonlinear static pushover analysis, and Templeton and Lewis (2011) reported on comparison for cyclic response via Abaqus nonlinear time domain dynamic analysis of response to random wave loading. Notably the maximum horizontal displacement of 4.4 ft determined in the analysis compared very well to the maximum observed displacement 4.44 ft – and the analysis was done entirely blind of the displacement data. The dominant wave period in this case was 14 seconds. The dominant response period of 10 seconds from the analysis also agreed with observation. For periods as long as these, inertial dynamic effects in the soil are generally inconsequential and usually ignored. For earthquake loading, however, the effects of inertial wave propagation in the foundation soils can be dominant.

Research at SAGE in Houston during the mid 1990s had demonstrated that inertial wave propagation in soils could be accurately modeled via continuum finite element dynamic analysis and – importantly – that Abaqus infinite elements could be used to form proper non-reflecting boundaries in such analyses. This fueled the concept of the construction of complete dynamic analysis of offshore structures and continuum foundations to earthquake loading. Significant additional motivation for this undertaking came from the increase in computational speed, and the very great decrease in computational cost, that had occurred by this time. The fastest computers

available for the initial Abaqus work of the early 1980s were Cray supercomputers (see Potvin, 1990). Supercomputers of that type cost millions of dollars, and the Abaqus nonlinear dynamics test problem, T5-STD, had required about 2 ½ hours on a Cray in that era. By the mid 1990s, a Super SPARC class microprocessor workstation computer at SAGE costing a thousand times less was running that same test problem in one hour. (For completeness, it is noted that a PC class computer capable of running the T5-STD problem in less than one minute can now be purchased for under \$1,000.) By the late 1990s all of the technical development was in place at SAGE to enable the analysis of combined structure and continuum foundation response to earthquake loading in Abaqus, and such analyses had been performed in proprietary work, but the results of such analyses would not be published until a decade later.

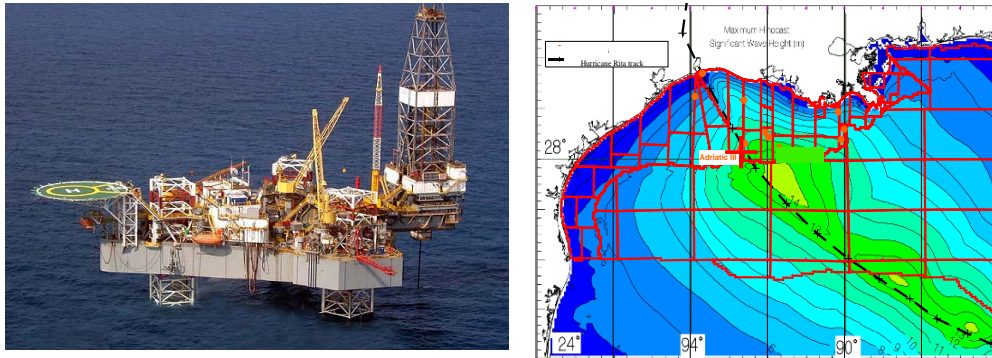


Figure 6. Photo of a LeTourneau 116-C rig (left, courtesy of Transocean) and location map (right, courtesy of Transocean and Ken Schaudt) showing the location of the Adriatic III LeTourneau 116-C rig (+) during Hurricane Rita in relation to the path of the storm center (- -) and storm surge contours.

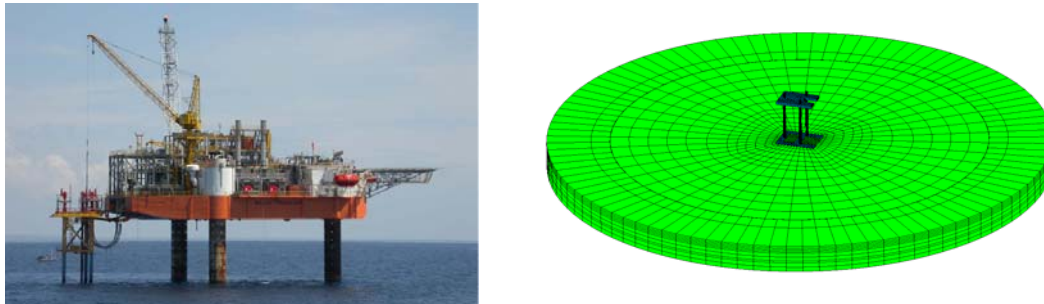


Figure 7. Photo of the Maleo Producer offshore of Madura Island, Indonesia (left, courtesy of Stewart Technology and Global Process Systems) and model plot (right) for Abaqus dynamic analysis of response to earthquake conditions.

The Maleo Producer (Figure 7) is a Bethlehem 250 mat supported jack-up rig converted to long term service as a gas production facilities platform offshore of Madura Island, Indonesia. Location of the rig in an active earthquake zone necessitated careful analysis of the performance of both structure and foundation under earthquake loading conditions. Templeton (2008) reported on the use of Abaqus time domain, nonlinear, dynamic analysis to accomplish this. This was the first reported use of finite element analysis to determine the nonlinear response of an offshore platform to earthquake analysis by incorporating the local ground response, soil-structure interaction and dynamic response of the structure and foundation all in a single continuum analysis. Details of the modeling and analysis included:

- A full-space (360-degree) 3-D model was developed with a simplified structural model and a 9-layer continuum model of the site soils. The 360-degree model allowed the incorporation of a structural model that was not symmetric, and it also allowed analyses with multi-directional shaking, so that all 3 components of earthquake ground motion could be applied simultaneously.
- The model represented a soil region 150 ft deep by approximately ½ mile wide. The depth was selected to be sufficient to reach firm soil conditions and to be sufficiently below expected variations in upward wave propagation to make the results relatively insensitive to the precise location of this boundary. The model width was selected to be sufficient to ensure one-dimensional response conditions at the far side boundaries, and it was also further established to be great enough for reflected waves traveling from the structure to be sufficiently diminished by the combination of travel time, travel distance and “R²” loss that their effects would be negligible. The adequacy of the dimension in these respects was verified by comparing the mudline motion at the far-field boundary to that from 1-D analysis.
- The modeled soil profile included soil stratigraphy and soil properties based on site-specific geotechnical data. The stiffness, mass and damping characteristics of the structure were specifically included in this model.
- Throughout the modeled soil region Abaqus first order hexahedral continuum elements with standard (selectively reduced) integration were used.
- Undrained total stress behavior was assumed throughout the model. Mass densities for the dynamic analysis were based on the interpreted total unit weights. Minimum material damping values were interpreted based on resonant column test data.
- Soil behavior for layer 9 of the model was taken as lightly damped elastic material using the interpreted, rate adjusted, shear modulus for that layer. Layers 1 through 8 were modeled as elastic-plastic kinematic work hardening materials. Parameters required to set the specific behavior of the model in each major soil unit were established by fitting the dynamic modulus reduction and damping buildup performance of the model to the dynamic test results.
- Nonlinear, time domain analyses were performed using the *dynamic procedure in Abaqus/Standard. Three-directional driving motions were imposed at the 150-ft depth. The input motions were based on scaled and depth corrected records selected to represent strength level (SLE) and ductility level (DLE) events.

- Twenty dynamic analyses were performed: 6 SLE and 6 DLE cases, 3 safety margin cases at 1.5 times SLE levels, 3 safety margin cases at 2.0 times DLE levels and 2 severe (DLE x 10) overload cases.

The analysis was successful in demonstrating to the certification authority that the Maleo Producer met the necessary performance requirements for the applicable design earthquake events, although prior analyses with simpler methods had proven incapable of doing this.

The type of combined model of structure and surrounding soil (Figure 7) used in this analysis has seen notable subsequent use in the offshore industry and has become known as a “soil island” model. The type of combined model of structure and surrounding soil used in this analysis has seen subsequent use in the offshore industry and has become known as a “soil island” model.

9. Conclusions and Recommendations for Further Development

In 1981, Abaqus became the first commercial general purpose finite element program capable of comprehensive nonlinear geotechnical analysis. As a result, it saw immediate use in providing valuable solutions to the most challenging problems in offshore soil mechanics and foundation engineering and soon became the high-end standard for this kind of work. Over three decades later, although there are now several other commercially available finite element programs capable of complex geotechnical analysis and in regular use in the offshore industry, Abaqus is still the leading program for high-end work of this kind. The following are identified as important reasons for this:

- 1. General quality.** The general quality of the Abaqus program far exceeds any other commercial general purpose finite element program. Abaqus was one of the very first finite element programs to achieve ISO 9001 certification. The initial ISO certification for Abaqus was granted in 1986, on the first application, without any substantial modifications to quality assurance procedures or additional verification work required beyond the already existing Abaqus verification work (Sorensen, 1997). Abaqus meets the applicable benchmarks of NAFEMS.
- 2. History of successful use.** Among commercial finite element programs, Abaqus has the longest history of successful use in foundation analysis of offshore structures. Significantly, this history has included important cases of successful verification against both field experience and centrifuge model tests.
- 3. Extensive material behavior library.** Abaqus’ extensive library of material behavior models particularly is especially well suited to offshore soils and foundation problems. Abaqus has at least a dozen material behavior models suitable for offshore geotechnical work. Few other programs have more than a fraction of that number. The large selection of material behaviors available facilitates the analyst’s choice of the best for any particular problem.
- 4. Selection of element types and features.** Abaqus provides the greatest selection of element types and features of any program available, including dozens of continuum 3-D elements and continuum plane elements. The Abaqus first order 2-D and 3-D elements are capable of highly accurate results for problems with plastic yield, with no volumetric locking in nearly incompressible materials.

5. Worldwide leadership in nonlinear FE analysis. Abaqus has long been recognized, worldwide, as the leading commercial program for nonlinear FE analysis. Historically, this resulted from the fact that Abaqus was written from the outset as a nonlinear program, while many competing programs began as linear codes and were later modified to include various sources of nonlinearity.

6. Rigorous large strain capabilities. The large deformation formulation in Abaqus is written from the standpoint of the nonlinear field theory of continuum mechanics, a fundamentally rigorous approach. The use of coordinate update from small strain solutions, a method used in some programs, can be less accurate, particularly if properly associated stress increment and stiffness corrections are not incorporated.

7. Unlimited 3-D feature applicability. Abaqus has unlimited applicability of all program features to 3-D analysis. This is essential to unrestricted formation of 3-D models and analyses. Abaqus is unrestricted in this respect because it was written initially from a 3-dimensional standpoint. Some other programs which began as 2-D programs presently have unfortunate restrictions for 3-D problems.

8. No program-imposed problem size limitations. Abaqus does not impose any limitations to problem size, such as maximum numbers of elements, nodes or degrees of freedom.

9. Level of program documentation, verification and technical support. Abaqus has levels of program documentation, verification and technical support that are simply unequalled in the finite element software industry. The manuals documenting the program for theory, examples, and user guidance are voluminous, comprehensive and well organized for convenient reference and ease of use. Technical support is available from a large full time support staff. The documented verification and benchmark problems number in the hundreds.

In addition, it should be noted that since the first introduction of soil mechanics capabilities in Abaqus, the program has seen substantial continuous improvement. Many of the new features have had specific utility to geotechnical analysis, notably infinite elements, explicit dynamics, contact surfaces, ALE adaptive meshing, Mohr Coulomb yield, Drucker Prager Cap, combined hardening, multiple exponential kinematic hardening, extended finite element method (XFEM), and – most interesting of late – Combined Eulerian-Lagrangian (CEL) analysis.

Still, there is ample need for continued Abaqus developments in areas of interest to geotechnical analysis. Among the needed developments are:

- availability of first order selectively reduced integration elements in Explicit (and CEL) analysis,
- availability of pore pressure-enabled elements in Explicit (and CEL) analysis,
- availability of creep in combination with the parabolic yield surface, and in combination with K values other than 1.0, in Drucker Prager plasticity,
- availability of creep in combination with the extended finite element method (XFEM) capabilities,

- availability of sub-yield surface kinematic hardening, in Drucker Prager and clay plasticity models,
- update of the Pi plane shaping function in Drucker Prager and Clay Plasticity to a more accurate, more robust and more widely accepted form (e.g. Matsuoka and Nakai, 1974),
- update of the formulations in the Spud Can option of Elastic-Plastic Joint elements from the present obsolete forms now currently unacceptable in current jack-up rig standards and recommended practices to those of the new ISO 19905-1.

With proper attention to continued program developments such as these, Abaqus seems likely to maintain its position of leadership for geotechnical finite element analysis well into the foreseeable future.

10. References

1. Al-Khafaji, Z. J., M. E. Audibert, M. K. Hossain, J. S. Templeton, III, and E. C. Clukey "Suction Caisson Foundation Design for Vortex-Induced Vibration Loading," Paper No. OTC 16843, Proceedings, Thirty-Fifth Offshore Technology Conference, Houston, Texas, 2003.
2. Bogard, J. D. and J. S. Templeton, III, "Setup of Large-Diameter, Thin-Walled Piles in Clay," FMMGI No. 0201-1496, 1992.
3. Carter, J. P., M. F. Randolph and C. P. Wroth, "Stress and Pore Pressure Changes in Clay during and after the Expansion of a Cylindrical Cavity," International Journal on Numerical and Analytical Methods in Geomechanics, Vol. 3, No. 4, pp. 305-322, 1979.
4. Clough, R. W. and E. L. Wilson, "Stress Analysis of a Gravity Dam by the Finite Element Method," Proc. Symp. On Use of Computers in Civil Engineering, Lisbon, Portugal, 1962.
5. Clough, R. W. and E. L. Wilson, "Early Finite Element Research at Berkeley," Fifth U.S. National Conference on Computational Mechanics, 1999.
6. Clukey, E. C. and M. J. Morrison, "A Centrifuge and Analytical Study to Evaluate Suction Caissons for TLP Applications in the Gulf of Mexico," Proc. ASCE Conf. on Foundations, Geotechnical Special Publications 38: 141-156. 1993.
7. Clukey, E. C., J. S. Templeton, III, M. F. Randolph, and R. Phillips, "Suction Caisson Response Under Sustained Loop Current Loads," Paper No. OTC 16843, Proceedings, Thirty-Sixth Offshore Technology Conference, Houston, Texas, 2004.
8. Hamilton, T. K., and J. S. Templeton, III, "Axial Capacity of Piles in Sand," Report to the American Petroleum Institute Geotechnical Resource Group, FMMGI Report No. 0201-1485, 1994.
9. Hibbitt, H. D., P. V. Marcal and J. R. Rice, "A Finite Element Formulation for Problems of Large Strain and Large Displacement," International Journal of Solids and Structures, 6, pp. 1069-1086, 1970.
10. International Standards Organization, "Site Specific Assessment of Mobile Offshore Units," Final Draft International Standard 19905-1, 2012.

11. Jeanjean, P., "Development of P-y Springs for Fatigue Analyses," Paper No. OTC 20158, Proceedings, Forty-First Offshore Technology Conference, Houston, TX, 2009.
12. Khan, N. I., J. S. Templeton, III and M. W. O'Neill, "Skin Friction Distributions on Piles in Sand," Proceedings of the International Conference, Civil Engineering in the Oceans V, College Station, Texas, 1992.
13. Little, R. L., G. H. Moore and J. S. Templeton, III, "Subsidence-Induced Sediment Motion: Foundation Design Criteria for Platforms in Main Pass 299," Paper No. OTC 6665, Proceedings, Twenty-Third Offshore Technology Conference, Houston, Texas, 1991.
14. Matlock, H., "Correlations for Design of Laterally Loaded Piles in Soft Clay," Paper NO. 1204, Proceedings, Second Offshore Technology Conference, Houston, TX, 1970.
15. Matsuoka, H., and Nakai, T., "Stress-deformation and Strength Characteristics of Soil Under Three Different Principal Stress," Proceedings Japanese Society of Civil Engineers, Vol. 232, pp. 59-70, 1974.
16. McMeeking, R. M. and J. R. Rice, "Finite-Element Formulations for Problems of Large Elastic-Plastic Deformation," International Journal of Solids and Structures, 11, pp. 601-616, 1975.
17. Miller, T. W., J. D. Murff and L. M. Kraft, "Critical State Soil Mechanics Model of Soil Consolidation Stresses around a Driven Pile," Paper No. OTC 3307, Proceedings, Tenth Offshore Technology Conference, Houston, TX, 1978.
18. Nagtegaal, J. C., D. M. Parks and J. R. Rice, "On Numerically Accurate Finite Element Solutions in the Fully Plastic Range," Computer Methods in Applied Mechanics and Engineering, 4, pp. 153-177, 1974.
19. Nystrom, G. A., Personal communication, 1980.
20. Nystrom, G. A., "The Use of Soil Mechanics Capabilities in a General Purpose Finite Element Program," Proceedings of ASCE Symposium on Prediction and Validation of Consolidation, San Francisco, 1984a.
21. Nystrom, G. A., "Finite-Strain Analysis of Piles in Clay," Analysis and Design of Pile Foundations: Proceedings of a Symposium Sponsored by the Geotechnical Engineering Division and a Session Sponsored by the Technical Council on Codes and Standards, ASCE, ed. by J. R. Meyer, 1984b.
22. Potvin, A. B., "Overview of Abaqus Applications At Exxon Production Research," Proceedings, Abaqus Users Conference, Providence RI, 1990.
23. Raines, R. D., O. Ugaz and J. Garnier, "Centrifuge Modeling of Suction Piles in Clay," Proceedings, Int. Symp. on Frontiers in Offshore Geotech., ISFOG, Perth, Western Australia, 2005.
24. Roscoe K. H. and J. B. Burland, "On the Generalised Stress-Strain Behaviour of 'Wet' Clay," Eng. Plasticity, Cambridge Univ. Press, 535-609, 1968.
25. Schofield, A. W. and C. P. Wroth, "Critical State Soil Mechanics, McGraw Hill," London, England, 1968.

26. Smith, I. M., "Programming the Finite Element Method - With Application to Geomechanics," John Wiley & Sons, New York, 1982.
27. SNAME, "Technical & Research Bulletin 5-5, Guideline for Site Specific Assessment of Mobile Jack-Up Units," 1991 and following editions.
28. Sorensen, E. P., Personal communication, 1997.
29. Templeton, J. S. III, "The Role of Finite Element Analysis in Suction Foundation Design," Paper No. OTC 14235, Proceedings, Thirty-Fourth Offshore Technology Conference, Houston, Texas, 2002.
30. Templeton, J. S. III, "Jack-up Foundation Performance in Clay," Paper No. OTC 18367, Proceedings, Thirty-Eighth Offshore Technology Conference, Houston, Texas, 2006.
31. Templeton, J. S. III, "Time Domain FE Seismic Analysis of Mat-Supported Jack-Up Structure on Soft Clay," Paper No. OTC 19645, Proceedings, Fortieth Offshore Technology Conference, Houston, Texas, 2008.
32. Templeton, J. S. III, "Finite Element Analysis of Conductor/Seafloor Interaction," Paper No. OTC 20197, Proceedings, Forty-First Offshore Technology Conference, Houston, Texas, 2009a
33. Templeton, J. S. III, "Spud Can Fixity in Clay, Further Results from a Study for IADC," Proceedings, 12th Int. Conf., The Jack-up Platform, City University, London, England, 2009b.
34. Templeton, J. S. III, J. N. Brekke and D. R. Lewis, "Spud Can Fixity in Clay, Final Findings of an IADC Study," Proceedings of the Tenth International Conference, The Jack-Up Platform, Design Construction and Operation, London, England, 2005.
35. Templeton, J. S. III and E. C. Clukey, "Foundation Load/Deflection Analysis for Concrete Island Drilling Structure Under Ice Loading," Proceedings of the Sixth international Offshore Mechanics and Arctic Engineering Symposium, Vol. 4, pp. 3-40, Houston, Texas, 1987.
36. Templeton, J. S. III, and D. R. Lewis, "Hysteretic Damping in Jack-up Dynamics," Proceedings, Thirteenth International Conference, The Jack-up Platform, City University, London, England, 2011.
37. Templeton, J. S. III, D. R. Lewis and J. N. Brekke, "Spud Can Fixity in Clay, First Findings of a 2003 IADC Study," Proceedings of the Ninth International Conference, The Jack-Up Platform, Design Construction and Operation, London, England, 2003.
38. Templeton, J. S. III, D. R. Lewis and J. N. Brekke, "Assessment of Jack-Up Survival in Severe Storms," Paper No. OTC 20255, Proceedings, Forty-First Offshore Technology Conference, Houston, Texas, 2009.
39. Templeton, J. S. III, R. L. Little, and G. A. Sraders, "Subsidence-Induced Sediment Motion: Analysis by Finite Element Methods for Sites in Main Pass 299," Paper No. OTC 6517, Proceedings, Twenty-Third Offshore Technology Conference, Houston, Texas, 1991.
40. Templeton, J. S. III, J. D. Murff, R. H. Goodwin and L. W. Klejbuk, "Evaluating Soils and Hazards in The Mississippi Canyon," Paper No. OTC 4864, Proceedings of the Seventeen Offshore Technology Conference, Houston, Texas, 1985.

41. Van Langen, H., P. C. Wong and E. T. R. Dean, "Formation and Validation of a Theoretical Model for Jack-Up Foundation Load-Displacement Analysis." *Marine Structures* 12(4): 215-230, 1999.
42. Wong, P. C., J. S. Templeton, III, O.A. Purwana, H. Hofstede, M.J. Cassidy, M.S. Hossain, and C.M. Martin "Foundation Modeling and Assessment in the New ISO Standard 19905-1," Paper No. OTC 23521, Proceedings, Forty-Fourth Offshore Technology Conference, Houston, Texas, 2012.

11. Acknowledgements

The work discussed in this paper received support from numerous sources over several decades. Much of the content presented here was drawn from earlier cited publications. The reader is directed to the acknowledgement statements in the cited references for comprehensive acknowledgements on all of this work. Gratitude is especially expressed for the important support, funding and permission to publish the underlying published work, particularly to The American Petroleum Institute, The International Association of Drilling Contractors, ExxonMobil, BP Americas, Freeport McMoRan, The French Institute for Marine Research, Global Process Systems, Lewis Engineering, Matthews-Daniel, Stewart Technology, Transocean and SAGE USA.

The author also wishes to express thanks for the contributions of his former colleagues at Exxon Production Research and McClelland Engineers as well as to his present and former SAGE colleagues, M. K. Hossain, PhD., P.E., F. B. Biegler, Z. M. Oden Duffy, PhD., E. L. Templeton-Barrett, PhD., M. J. Barrett, and A. A. Rahim, PhD., P.E., all of whom contributed to the Abaqus analyses discussed here.