Ocean Systems Engineering, *Vol. 5, No. 1 (2015) 31-40*
DOI: http://dx.doi.org/10.12989/ose.2015.5.1.031

Analysis of offshore pipeline laid on 3D seabed configuration by Abaqus

Ali Shaghaghi Moghaddam[*1,2], Saeid Mohammadnia[2] and Mohammad Sagharichiha[2]

[1]Young Researchers and Elite Club, Takestan Branch, Islamic Azad University, Takestan, Iran
[2]Pipeline Engineer, Iranian offshore and construction company (IOEC), Vila street, Tehran, Iran

(Received September 15, 2014, Revised March 4, 2015, Accepted March 8, 2015)

Abstract. Three dimensional (3D) non-linear finite element analysis of offshore pipeline is investigated in this work with the help of general purpose software Abaqus. The general algorithm for the finite element approach is introduced. The 3D seabed mesh, limited to a corridor along the pipeline, is extracted from survey data via Fledermous software. Moreover soil bearing capacity and coefficient of frictions, obtained from the field survey report, and are introduced into the finite element model through the interaction module. For a case of study, a 32inch pipeline with API 5L X65 material grade subjected to high pressure and high temperature loading is investigated in more details.

Keywords: offshore pipeline; Abaqus; stress and strain analysis; three dimensional

1. Introduction

With the growing demand for energy, nowadays it is a big challenge to use gas and oil reservoirs and export it for domestic use. It is known that the biggest gas reservoir is located in the Persian Gulf, which is shared with Arabian countries. As a consequence it is of particular importance to extract more gas in the mutual fields. Based on this fact, different phases are originated in the shared areas. When the gas is retained in the wellheads, it is being processed and then is exported to onshore platforms, e.g., Assaluy and Tonbak. The gas fields are somehow 100 kilomerets far away the onshore plants. For exportation, submarine pipelines are employed for almost 1000MMSFD for each line. So many issues should be considered to export gas from offshore platform to the onshore. Pipeline size, route configuration, stability analysis, free span issue, stress check, local buckling analysis and etc. are of main concerns which must be studied. Any faults in the system, result in danger, environmental issue and too much cost. Offshore industry is inherently a costly process which consume too much time and cost. To prevent any danger and failure from offshore pipeline, a details analysis is required. Usually, offshore pipelines are designed for 25 year life, with a safety factor which is tabulated in (DNV OS F101). By the introduction of high pressure and high temperature gas, the pipeline tends to expand. Hence, it is of importance to have an understanding of pipeline buckling before putting the pipeline into

*Corresponding author, Assistant Professor, E-mail: Shaghaghi@ioec.com

http://www.techno-press.org/?journal=ose&subpage=7 ISSN: 2093-6702 (Print), 2093-677X (Online)

operation. Hydrogen induced cracking (HIC), and Sulphide stress cracking(SSC) are likely to occur in the existence of any excess of axial strain in the presence of sour service. Consequently, the axial strain in the pipeline has to be remained below an allowable value. On the other hand, the maximum stress on the pipeline should be lower than the allowable strength.

DNV OS F101 outlines the rules of submarine pipeline design and limit state design is incorporated in this code. The necessary equations for allowable stress design are provided in (ASME B31.8 and API RP-1111). The use of finite element analysis tools for the design and simulation of subsea oil and gas pipelines are studied in (Jukes *et al*. 2008). Pipeline integrity assessment considering soil influence is studied in (Wang *et al*. 2008). The integrated Design Approach of South Pars Phases 2, 3 and 6, 7, 8 Gas Field is considered in (Langford *et al*. 2005). The influence of Internal Pressure, axial Force and bending moment on pipeline integrity is studied in (Vitali *et al*. 1999). Some experience on pipeline engineering is investigated in (Jukes 2007). Three dimensional finite element analysis of 4 inch smart flange on offshore pipeline has been presented in (Shaghaghi and Mohammadnia 2014). In that work it was aimed to analyze the failure of a smart flange on offshore pipeline located on a curve. With a comprehensive numerical analysis it was shown that lateral buckling is likely to occur in curved segment of the pipeline in the presence of temperature and pressure. And consequently large bending moment is introduced in the location of smart flange.

Based on the several past experiences on 3D finite element modeling of south pars phase's projects by the authors, the present paper is aimed to investigate a full three dimensional finite element analysis of submarine pipeline. To this purpose, field data are employed for seabed modeling and soil factor characteristic. Fledermous software is used for seabed modeling and some in-house MathCad sheets are developed to extract soil data. Seabed mesh and soil data are incorporated in Abaqus to study a full 3D finite element analysis. General algorithm of finite element analysis is presented here and discussed in details. Finally, a case of study is integrated to analyze the results.

2. Finite element approach

Nowadays for the growing industry of offshore technology, it is highly demanded for accurate estimate of the integrity of the system. To assess the integrity of the pipeline laid on seabed, older standards were based on ASD criteria, allowable stress design. Which was a cross check of allowable strength, some percent of yield strength, compared with maximum stress observed in the structure. However in the competitive market to save cost, the limit state approach was introduced. DNV OS F101 outlines the limit state approach for pipeline, e.g., maximum bursting pressure, local buckling check and etc. These are mostly strain based analysis with a nonlinear analysis to consider elastic plastic material properties. For instance, the axial force and bending moment in the cross section of the pipeline is required for local buckling check. As a consequence considering different aspects such as soil properties, material elastic plastic properties, seabed configuration, temperature and pressure profile along the route and etc. the trend of axial force and total section bending moment along the pipeline has to be evaluated. Based on the body of introduction stated above, a nonlinear analysis of pipeline is mandatory for limit state design. Nowadays numerical approaches with finite element models are employed for a detail analysis. With the help of general finite element software, such as Abaqus, one can include all parameters for accurate analysis of the pipeline. In the present work it is aimed to investigate a nonlinear analysis of offshore pipeline laid on seabed to assess the integrity. It is of main importance to consider most governing factors for accurate estimation.

The general algorithm for numerical analysis is shown in Fig. 1. At the first stage the three dimensional configuration of seabed has to be imported to the finite element software. To this aim the XYZ data from the field survey is mapped to the Fledermous. It should be noted that XYZ field survey report involves the easting, northing and seabed elevation, described by X, Y and Z, respectively. As a consequence, the elevation profile of seabed along the pipeline is imported in the analysis according to XYZ data. Considering the route of the pipeline, a cloud of points within a certain corridor representing the seabed is extracted from Fledermous. These points are subsequently imported to the Abaqus to form a mesh representing the seabed. The size of element in the seabed mesh is a user defined value which is controlled in Fledermous output setting. For more illustration, a section of the seabed is depicted in Fig. 2. For the next stage, based on the CAD data from general field layout, the pipeline route is drawn in Abaqus part module. The material property of the pipeline is imported to Abaqus and then is assigned to the Pipeline. Then the route and seabed is assembled in the assembly module, Fig. 2. The interaction between pipeline and seabed is defined in the Interaction module. The seabed bearing capacity and coefficient of friction are of main importance to be correctly considered. Special care has to be taken into account for this stage. More information is presented in the next section. For more detail analysis, governing loads and boundary conditions have to be imposed to the model carefully. From the hydraulic analysis report, temperature and pressure profiles along the route are extracted. The hydraulic analysis report is a very important document studying the hydraulic and surge analysis based on the nominal pipe size and flow rate of the exporting gas. The profiles are then imported to the Abaqus for assigning temperature and pressure to each node along the route. Finally the job is run and the results can be visualized. In the next section more details of numerical modeling is presented for more clarification of the approach.

3. Modeling issues

3.1 Seabed modeling

As discussed in the previous section, the 3D configuration of the seabed is imported to finite analysis through Fledermous. A corridor with a certain width from the XYZ data is exported to Abaqus by a cloud of points. A group of mesh is created from the cloud of points to represent the seabed. The seabed type is considered as three dimensional discrete rigid. However it should be noted that the soil properties have to be assigned to seabed mesh. The soli bearing capacity and coefficient of frictions are the most important factors that have to be included into the analysis. Soil vertical stiffness and coefficient of friction along the pipeline and transverse to the pipeline have to be considered. it should be noted that according to the field survey, the soil type and characteristic is extracted. Hence with respect to soil type, at any point along the pipeline, the soil properties are assigned. The soil stiffness in vertical direction is characterized as shown in Fig. 3, Which k1 (N/mm2), k2 (N/mm2) and p1 (N/mm) are soil parameters which are extracted from an in-house Mathcad sheet based on the soil survey report. It has to be mentioned that for operation case and hydrotest case, the soil data are somehow different and should be calculated separately.

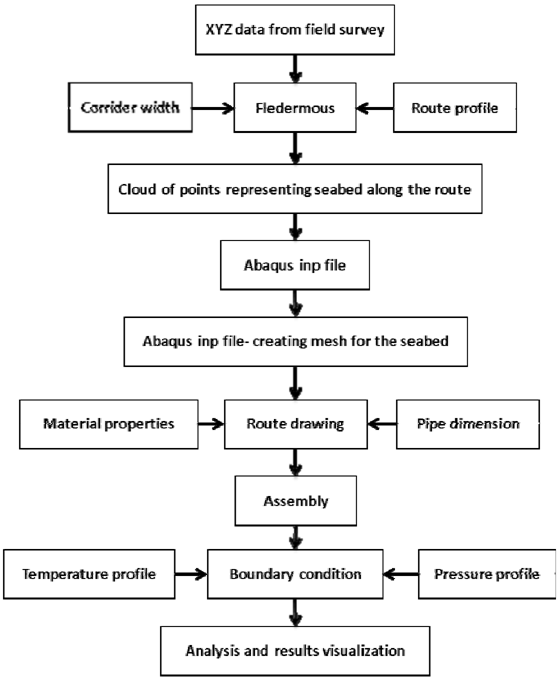

Fig. 1 General algorithm for the finite element analysis

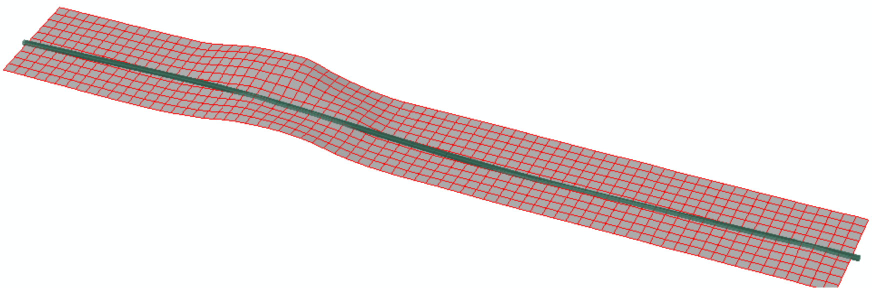

Fig. 2 Three dimensional seabed demonstration

After estimating the factors for vertical soil stiffness, they are assigned to the rigid discrete seabed through the contact property manager, normal behavior, in the Interaction module. In the next step, the coefficient of friction of soil has to be integrated in the finite element analysis. It has to be mentioned that as the soil type varies along the pipeline, corresponding soil data has to be assigned to respective pipeline section. This is an important task due to high influence of soil on pipeline behavior. Similar to vertical soil stiffness, coefficient of friction is extracted from the MathCad sheet based on the soil survey report. Soil coefficient of friction is distinguished for axial and lateral direction. Where the axial direction is aligned along the normal to the pipe cross section and perpendicular to that is the lateral direction. As a consequence, it is of main concern to employ anisotropic soil friction along the axial and lateral direction.

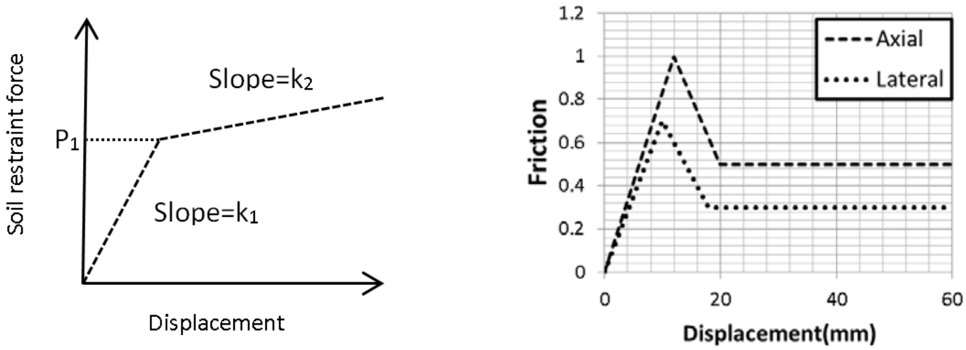

Fig. 3 Soil bearing capacity and coefficient of friction

The main important task in analyzing offshore pipeline is to create proper interaction between pipe and seabed. The interaction between pipe and seabed should consider the axial and lateral coefficient of frictions. To reach this aim, at first the Surface-to-surface contact is considered for the interaction type in Abaqus interaction module. Then the seabed and pipe are selected as master and slave surfaces, respectively. It is of importance to select the slave type as Surface, not node region. And subsequently the pipe is selected and then Circumferential surface type is considered. The discretization method is considered as surface to surface. As shown in Fig. 3 the coefficient of friction increases with a constant slope to a peak value with respect to the pipeline movement. However, afterward it reaches a residual value for the remaining pipeline movement. As a consequence, the soil resistance to pipeline movement increases to a certain amount and subsequently decreases to a residual value. Abaqus CAE is not capable of considering peal value for soil friction coefficient. Only initial slope and residual value can be integrated. However for more details analysis, User subroutine FRIC is employed to define frictional behavior for contact surfaces.

3.2 Pipeline design data

API materials grades are used for submarine pipeline, X52- X65 and etc., (DNV OS F101). Elastic-plastic material properties can be found in DNV OS F101. It should be noted that due to raised temperature near the wellhead, material properties vary. This is characterized by material de-rating and has to be included in the finite element analysis. In the Fig. 4 elastic-plastic a material property for 3 temperature values is drawn. Finally elastic-plastic material properties for different temperature are incorporated in Abaqus. For any other temperature, Abaqus interpolates to find the respective material properties.

Pipeline nominal diameter is extracted from the hydraulic analysis report. An anti-corrosion coating is applied to pipeline outer diameter to protect it from seawater. Mostly to assure bottom stability, a concrete coating is applied to the pipeline. The inner of the pipeline is remained uncoated, unless it is cladded to protect against internal corrosion. The concrete coating effects to stress analysis is ignored in this analysis and its contribution to submerged weight of line has been taken into account. Detail of the pipeline cross section near shore is shown in Fig. 5.

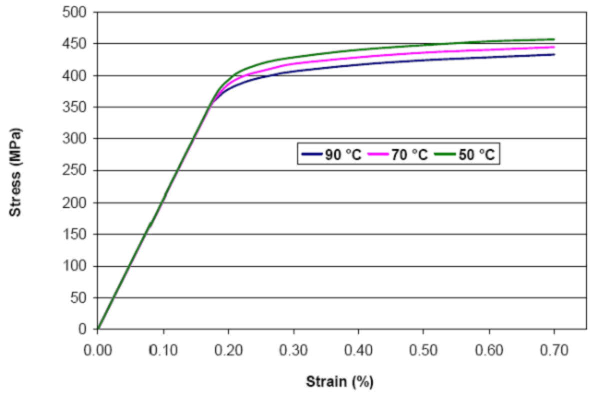

Fig. 4 Stress-strain curve for different temperature

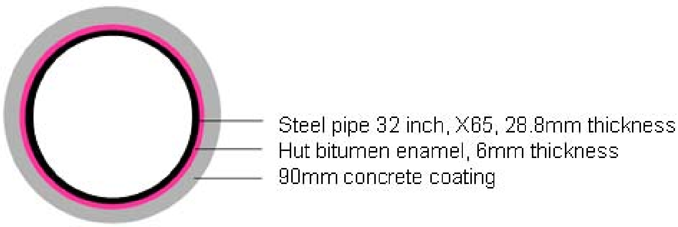

Fig. 5 Detail of the pipe cross section in the near shore segment, 0-500 m

For the finite element analysis 2-node pipe element in space, PIPE31, is considered, which the element size is one OD. As the governing soil data and concrete thickness vary along the route, the pipeline is discretized to some segments; where in each segment the situation is similar. Then for the finite element modeling different section is defined to tailor the properties of each segment.

4. Numerical example

Lateral buckling of 4inch pipeline was studied in (Shaghaghi and Mohammadnia 2014) with three dimensional finite element analysis. In that work the failure of a smart flange was investigated with 3D finite element analysis of smart flange and lateral buckling. In the present work the numerical analysis of offshore pipeline is presented as the case of study. Hot section of a submarine pipeline, almost 10 kilometers from offshore platform, is studied. Pipeline data are extracted from respective documents such as design basis, on bottom stability analysis, thermal buckling analysis and etc. Pipeline data are tabulated in Table 1. As discussed previously, the submarine pipeline is discretized to some segments. Table 2 depicts pipe data, soli data and coating data for each segment.

Temperature and pressure profile extracted from hydraulic analysis is also shown in Fig. 6. These profiles are imported to Abaqus and corresponding value is assigned to each node on the pipeline.

Table 1 Pipeline data

Pipe nominal diameter	32 inch
Steel Grade	API 5L X-65
Steel density(kg/m^3)	7800
Young's modulus(MPa)	210000
SMYS(MPa)	450
Concrete coating density(kg/m^3)	3040
Anti Corrosion density(kg/m^3)	1400

Table 2 Pipeline segments data

KP range (km)	**Pipe thickness(mm)**	**Concrete thickness(mm)**	**Anti-corrosion coating Hut bitumen enamel(mm)**	**soil type**
0-0.5	28.8	90	6	Clay
0.5-3	20.6	50	6	Clay
3-7	20.6	68	6	Soft Clay
7-10	20.6	50	6	Soft Clay

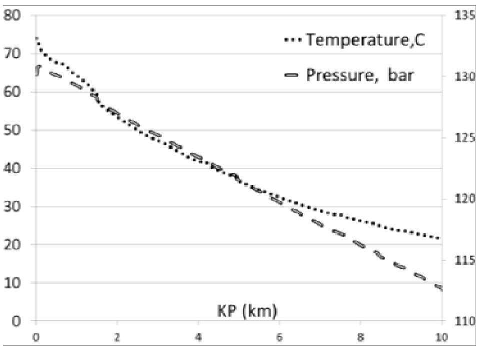

Fig. 6 Temperature and pressure profile

One of the important issues in offshore pipeline is the rectification of free spans. To study free span, bottom roughness analysis is done. The maximum length of the pipeline which is not in contact with seabed and also the maximum normal distance of the pipeline from seabed are extracted from bottom roughness analysis. After installation, any free spans out of the allowable range have to be corrected, with grout bag or supports. From the full three dimensional finite element analysis of the pipeline with Abaqus, a very important result can be extracted. COPEN output is the normal distance of any node from the seabed which can be used for bottom roughness analysis. For a small section of the pipeline, a sample COPEN is shown in Fig. 8. For a detail analysis COPEN is drawn along the path on the pipeline.

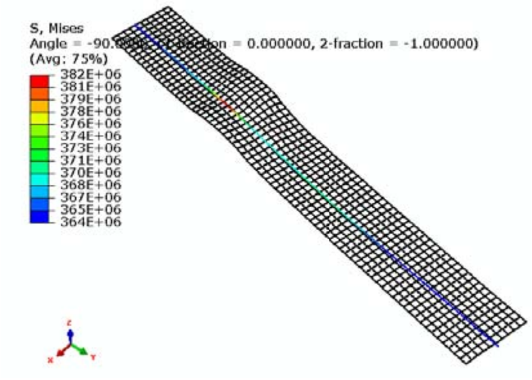

Fig. 7 Von mises stress for a segment of the pipeline

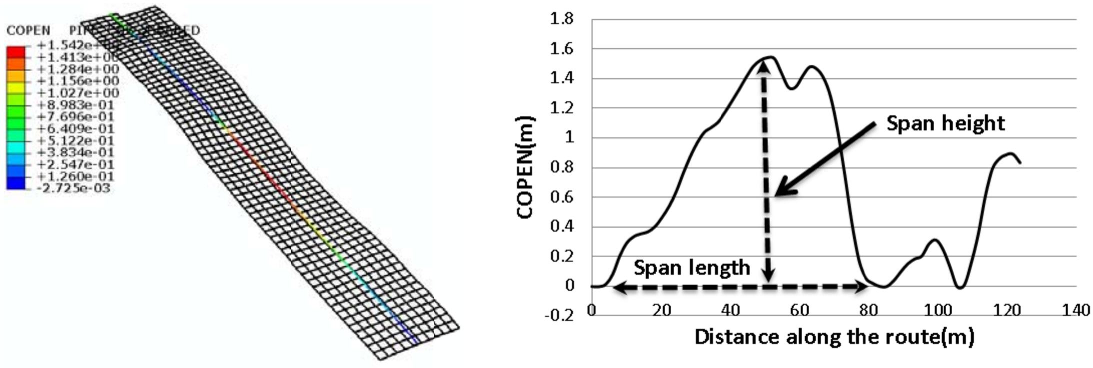

Fig. 8 Normal distance of any node from the seabed, COPEN

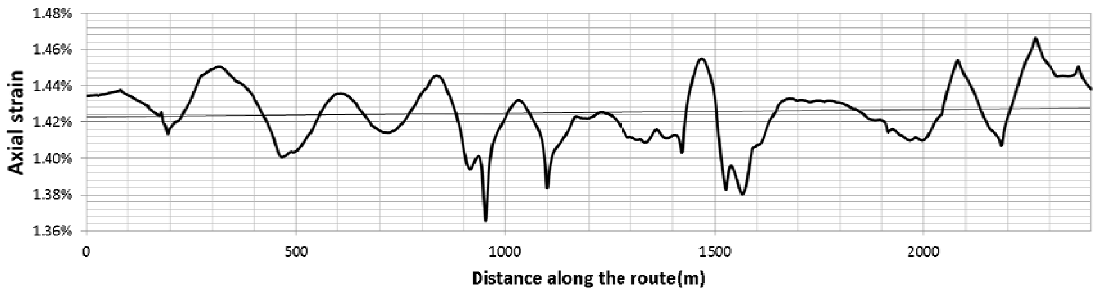

Fig. 9 Axial strain along the pipeline

In the presence of sour service the maximum axial strain has to be in acceptance range, (DNV OS F101). The excessive tensile axial strain motives for hydrogen induced cracking and sulphide stress cracking. For this circumstance, the axial strain has to be checked along the pipeline. In Fig. 9 the axial strain is shown for a section of the pipeline for the presented example. A more detail investigation is required for whole pipeline length. The results have to be interpreted and necessary action has to be taken in the case of excessive strain.

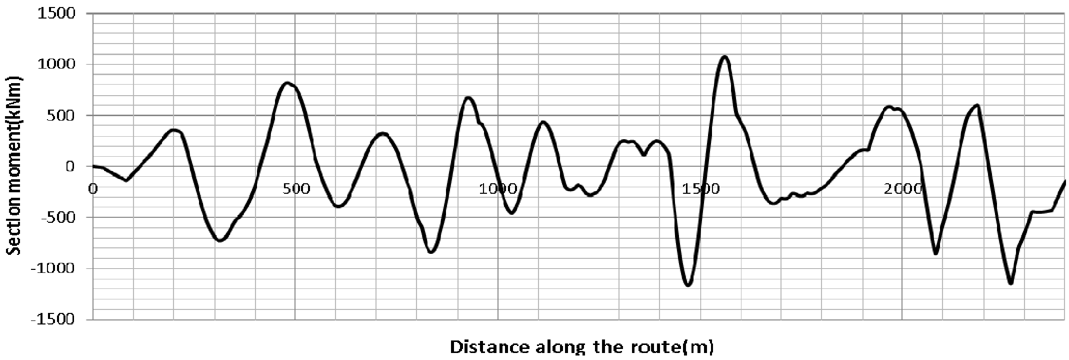

Fig. 10 Total section moment along the pipeline

From Fig. 9 it is inferred that for the selected pipeline segment staring at the free end, the axial strain is positive. This pipeline segment is in the active length region. The pipeline segment between the free end and Virtual Anchored Point is called active length. At virtual anchored point, the net longitudinal strain is equal to zero. The net longitudinal strain between the free end and the virtual anchored point is governed by strains due to the effect of end cap, Poisson ratio, thermal expansion and coefficient of friction. Moving from the free end, the effect of negative strain due to influence of coefficient of friction gets raised. Accordingly the net longitudinal strain in active length is positive and tends to zero at virtual anchor point. One of the importance check for limit state design, is the local buckling check. This is a strain base analysis to resist plastic buckling of the pipeline subjected to axial force and total section bending moment. More details can be found in DNV OS F101. Total section bending moment and axial force along the route have to be extracted from finite element analysis. For the sake of clarification, in Fig. 10 the total section moment is shown for a pipeline segment. From the final analysis different outputs can be requested. For instance, CSLIP1 and CSLIP2 are axial and lateral displacement of the pipeline, respectively. CSLIP2 are employed for lateral buckling check. End expansion of the pipeline which is used for spool design can be extracted as CSLIP1. Effective axial force along the pipeline, hoop stress, plastic strain and etc. can be requested from Abaqus analysis.

The total section bending moment and axial force are very informative output which further are used for post processing stage, for instance local buckling check or Engineering Critical Assessment (ECA).

5. Conclusions

Offshore engineering demands more reliable design to prevent any unwanted damages and failures. Due to complexity of submarine pipelines and existence of different sources of nonlinearities, numerical approaches are employed for modeling rather than analytical ones. The nonlinearities of the problem comprise of soil resistance, elastic-plastic behavior of pipe material, 3D seabed configuration and etc. Design data for modeling of submarine pipelines are extracted from different documents such as basis of design, thermal buckling report, on bottom stability analysis and hydraulic analysis report. On the other hand, topography survey report for seabed configuration and soil survey data are imported to the finite element analysis. In the present work seabed configuration is extracted from XYZ data from fields survey through software Fledermous. Moreover, soil bearing capacity and soil coefficient of friction are calculated from an in-house MathCad sheets. A nonlinear implicit analysis with general static step is performed. Different output results can be requested from Abaqus Visualization module, e.g. stress, strain, effective axial force and total bending moment and etc. A path is defined along the pipeline and all results are reported on the path. Based on the standards and codes and also general specification, the results have to be interpreted for further modification.

References

ASME B31.8, Gas Transmission and Distribution Piping Systems.

API RP-1111, Design, Construction, Operation and Maintenance of Offshore Hydrocarbon Pipelines (Limit State Design).

DNV OS F101, Submarine Pipeline Systems.

Jukes, P. (2007), "Innovative Pipelines and Subsea Engineering Experience", *Key Note Paper at The Proceedings of the International Conference on Subsea Technologies*, St. Petersburg, Russia, 25-28 June.

Jukes, P., Eltaher, A. and Wang, J. (2008), "The use of Advanced Finite Element Analysis Tools for the Design and Simulation of Subsea Oil and Gas Pipelines and Components", *Proceedings of the Petroleum Exhibition and Conference of Mexico*.

Langford, S., Maschner, E. and Kenny, J.P. (2005), "South Pars Phases 2, 3 and 6, 7, 8 Gas Field Development – An Integrated Design Approach", *Proceedings of the 28th Annual 'OPT' Conference*, Amsterdam 1-2.

Shaghaghi, A.M. and Mohammadnia, S. (2014), "Three dimensional finite element analysis of 4 inch smart flange on offshore pipeline", *Ocean Syst. Eng.*, **4**(4), 279-291.

Vitali, L., Bruschi, R., Mork, K., Levold, E. and Verley, R., (1999), "HOTPIPE Project: Capacity of Pipes Subject to Internal Pressure, Axial Force and bending Moment", *Proceedings of the ISOPE Conference, Brest*, France.

Wang, J., Wang, S., Eltaher, A.N. and Juke, P. (2008), "Pipeline integrity assessment with pipeline, soil and berm interaction and external load", *Proceedings of the Petroleum Exhibition and Conference of Mexico*.