Demirci, H. E., Bhattacharya, S., Karamitros, D., Alexander, N., & Singh, R. M. (2018). Finite element model of buried pipelines crossing strike-slip faults by ABAQUS/EXPLICIT. In *Recent Advances in Earthquake Engineering in Europe: 16th European Conference on Earthquake Engineering-Thessaloniki 2018* (Geotechnical, Geological and Earthquake Engineering; Vol. 46). Springer, Cham.

16TH EUROPEAN CONFERENCE ON
EARTHQUAKE ENGINEERING THESSALONIKI 18 - 21 JUNE 2018

FINITE ELEMENT MODEL OF BURIED PIPELINES CROSSING STRIKE-SLIP FAULTS BY ABAQUS/EXPLICIT

Hasan Emre DEMIRCI[1], Subhamoy BHATTACHARYA[2], Dimitris KARAMITROS[3], Nicholas ALEXANDER[4], Rao Martand SINGH[5]

ABSTRACT

In seismically active regions, buried pipelines can be subjected to severe ground deformations due to faulting, landslides, lateral spreading and seismic settlement. The permanent ground deformations (PGD) may induce large strains in the pipeline. Prediction of those pipe strains is required to evaluate pipeline performance and to design earthquake resistant pipelines. Therefore, development of analytical or finite element (FE) model of the problem is necessary. The behavior of buried pipelines crossing faults have been analyzed by several researchers in the past using static analysis due to static nature of fault loading. Consequently, ABAQUS/Standard module is widely used to solve soil-pipeline interaction problems under faulting since it is capable of performing static/quasi-static analysis. In this study, an ABAQUS/Explicit model, which is mostly used for dynamic analysis, was developed to simulate pipelines crossing strike-slip faults considering an energy balance concept. The model results were compared to ABAQUS/Standard and analytical results for validation purpose. The comparison of the results shows that ABAQUS/Explicit module is a reliable tool to effectively analyze quasi-static loading problems.

Keywords: Faulting; buried pipelines; permanent ground deformations, soil-pipe interaction; quasi-static analysis

1. INTRODUCTION

Past earthquake-related pipeline damage highlighted the vulnerability of buried pipelines to permanent ground deformation (PGD). Different types of pipeline failure modes such as joint failure, tension failure, beam buckling and local buckling failure have been observed during past earthquakes. The pipeline failure severely affects economies, industries and societies. Therefore, earthquake resilient design of buried pipelines is one of the important aspects in geotechnical and structural engineering.

Finite Element (FE) codes (ABAQUS, ANSYS, etc.) are widely used to investigate the response of buried pipelines to PGD since FE method is one of the most powerful numerical tools, considering the nonlinear soil/pipe behaviour, contact between soil and pipe, and geometric nonlinearity. One dimensional (1D), three dimensional (3D) and hybrid FE models have been used in earlier studies (Lim et al. 2001, Takada et al. 2001, Karamitros et al. 2007, Vazouras et al. 2010, 2012, 2015) in order to validate analytical methodologies, perform parametric studies and design pipeline considering strain based performance criteria for tensile, beam buckling, local buckling failure and ovalization.

Buried pipelines crossing active faults are considered as a complex soil-structure interaction problem. These soil-pipe interaction problems have been analyzed by many researchers using static analysis due to the nature of fault loading on pipe is static rather than dynamic. In the analysis, the pipeline is assumed to be moving slowly and this assumption negates inertial force development. Consequently, behaviour of pipelines crossing faults can be investigated by using static (quasi-static) analysis (Robert et al. 2014).

---

[1] PhD Candidate, Civil Engineering, University of Surrey, United Kingdom, h.demirci@surrey.ac.uk
[2] Professor, Civil Engineering, University of Surrey, United Kingdom, s.bhattacharya@surrey.ac.uk
[3] Lecturer, Civil Engineering, University of Bristol, United Kingdom, d.karamitros@bristol.ac.uk
[4] Senior Lecturer, Civil Engineering, University of Bristol, United Kingdom, Nick.Alexander@bristol.ac.uk
[5] Lecturer, Civil Engineering, University of Surrey, United Kingdom, r.singh@surrey.ac.uk

ABAQUS/Standard module is capable of carrying out static and quasi-static analysis and the module has been mostly used for solving soil-pipeline interaction problems in the literature (Vazouras et al. 2010, 2012, 2015). There may occur convergence problems causing difficulties in reaching ultimate solution in ABAQUS/Standard model due to complexity in the soil-pipe interaction model including non-linear material behaviour of soil and pipe, geometric nonlinearity of the pipe and contact between soil and pipe. ABAQUS/Explicit module is very powerful to deal with complicated contact problems.

In this study, lateral movement of a pipe in clay soil has been simulated by using both ABAQUS/Standard and ABAQUS/Explicit modules and the results (soil resistance–lateral displacement) are compared to the values suggested in ASCE Guidelines (1984) for validation purpose. The efficiency of ABAQUS/Explicit module is investigated for simulating pipelines crossing PGD zones, particularly buried pipelines crossing strike-slip faults. The maximum tensile and compressive pipe strains obtained by ABAQUS/Explicit analysis are compared to those values obtained by analytical methodologies (Sarvanis and Karamanos (2017), Karamitros et al. 2007) in order to validate 3D FE model.

2. ABAQUS/EXPLICIT FOR SOIL-PIPELINE INTERACTION PROBLEMS

ABAQUS/Explicit module offers a powerful technique for the solution of dynamic problems (high speed events when inertial forces are not negligible). Inertia plays a significant role in the solution. There are several strategies to analyse quasi-static problems by using ABAQUS/Explicit. Energy balance concept is essential to be considered in order to have quasi-static analysis in ABAQUS/Explicit.

Energy content provides very useful indication to evaluate whether numerical simulation results obtained by ABAQUS/Explicit reflect a quasi-static solution. The system kinetic energy relative to system internal energy (ALLKE/ALLIE) should not exceed 1-5% throughout the majority of the quasi-static analysis. Hence, inertia force in the system is negligible. The system kinetic energy (ALLKE) and internal energy (ALLIE) can be checked in time history output.

The reduced integration first order elements have numerical difficulty called hourglassing. Hourglass modes are zero energy modes (ZEM) and they do not generate any strain or stress. However, they can affect numerical solution. Mesh elements are excessively flexible due to this numerical difficulty. There are several algorithms available in ABAQUS for controlling internal hourglass forces, which are applied to resist the hourglass mode deformation. The artificial energy generated by internal hourglass forces (ALLAE) can be checked in time history output. Artificial energy relative to internal energy (ALLAE/ALLIE) should be less than 5% in order to have reliable results.

3. VALIDATION OF ABAQUS/EXPLICIT MODEL

3.1 Lateral movement of pipe in the soil medium

A 3D FE simulation of the pipe moving laterally in the soil medium was created by using ABAQUS v 6.14. This simulation was modeled by using both ABAQUS/Standard and ABAQUS/Explicit module. Two different stages were created to simulate the real field conditions: 1) gravity loading and 2) lateral movement of the pipe. Gravity loading is applied to whole model to simulate initial stress in the soil and on the pipe due to self-weight of the soil and pipe. In the second stage, the pipe is displaced by 0.15 m laterally. Reduced integration continuum solid elements (C3D8R) were used to model soil medium and Mohr-Coulomb model (elastic perfectly plastic) was selected to represent the stress-strain relationship in the soil. Reduced integration shell elements (S4R) were chosen to model the pipe and Isotropic Von Mises yield model was used for steel pipe element. Tangential and normal contact were used to model the interaction between the soil and pipe. The friction between the soil and pipe was simulated by tangential contact algorithm with a proper friction coefficient (μ). The normal contact with selecting hard contact algorithm allows the separation of the pipe and soil surfaces. 3D FE models used for the analysis are shown in Figure 1. The parameters used in the 3D FE model are given in Table 1.

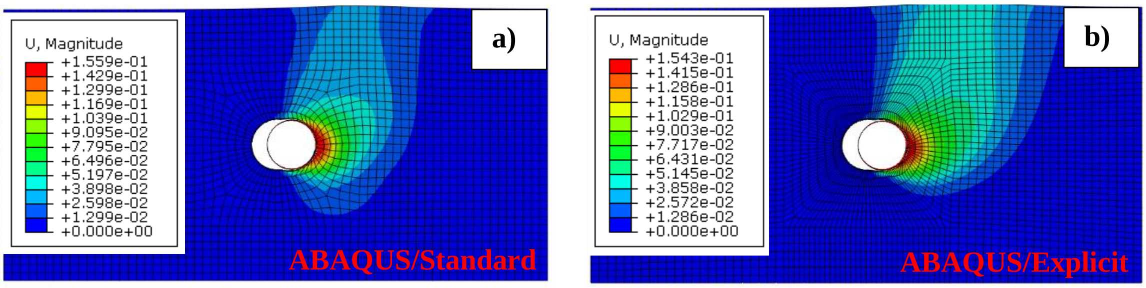

Figure 1. 3D FE model of lateral movement of the pipe in soil medium for a) ABAQUS/Standard module b) ABAQUS/Explicit module

The bottom wall of the soil medium was restricted as an encastre and side walls of the soil were restricted in horizontal direction. The uniform displacement of 0.15 m was applied to the pipe in lateral direction as shown in Figure 1. A fine mesh was employed for the soil surrounding the pipe since maximum stresses and strains in the soil were expected to develop in this region.

A comparison between ABAQUS/Standard and ABAQUS/Explicit results is shown in Figure 2. The comparison demonstrates that the predictions of load-lateral displacement relations are quite satisfactory. The prediction of soil-pipe interaction according to ASCE (1984) guideline shows close trend with respect to FE results.

Table 1. Engineering properties of the soil and pipe and contact parameters used for FE modelling

Soil: Clay			
Elastic		Plastic	
E (MPa)	25	φ (°)	0
ϑ	0.3	Ψ (°)	0
		c (kPa)	50
Pipe: Steel Pipe			
Elastic		Plastic	
E (GPa)	210	Yield Stress, MPa (σ_y)	450
ϑ	0.3		
Contact			
Tangential		Normal	
μ	0.3	Hard Contact	

3.2 Pipelines crossing strike slip faults

A quasi-static nonlinear analysis of the 0.9144 m diameter pipeline crossing a strike-slip fault was performed by applying fault displacements incrementally. The pipeline located 2.5 m below the ground and the dimensions of the soil medium was chosen as 5 m depth and 10 m width as presented in Figure 3a. The length of the model was selected as 60 pipe diameter (60D) as in Vazouras et al. 2010, Sarvanis and Karamanos (2017). Two different loading steps were utilized to simulate the soil-pipeline interaction problem: 1) gravity loading, 2) fault displacement. In first step, gravity loading was applied to the whole model and in the second step, fault displacements of 1m with 20° fault crossing angle (0.940 m in -x direction and 0.342 m in -z direction) were applied to left hand side of the soil block (moving block) and other soil block (fixed block) was restrained in horizontal and axial directions. Equivalent boundary springs proposed by Liu et al. 2004 were used to take into account boundary conditions in the field (Figure 3b). Longitudinal pipe strain distribution at red dashed zone is demonstrated in Figure 3d. A fine mesh was employed for the regions close to the fault trace and the soil surrounding pipe.

The engineering properties of the soil and pipeline, the characteristics of the fault and parameters used in the 3D FE model are given in Table 2.

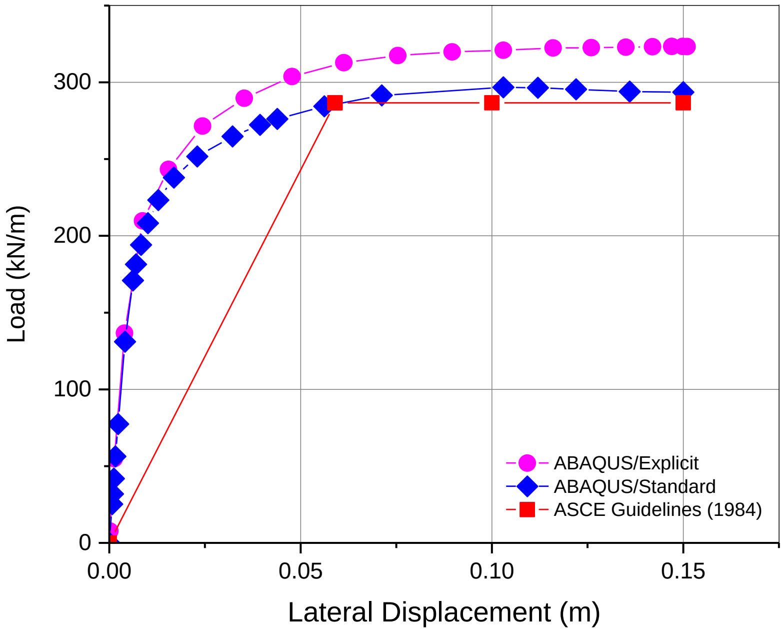

Figure 2. Lateral soil-pipe interaction for ABAQUS /Explicit, ABAQUS/Standard and ASCE Guidelines (1984)

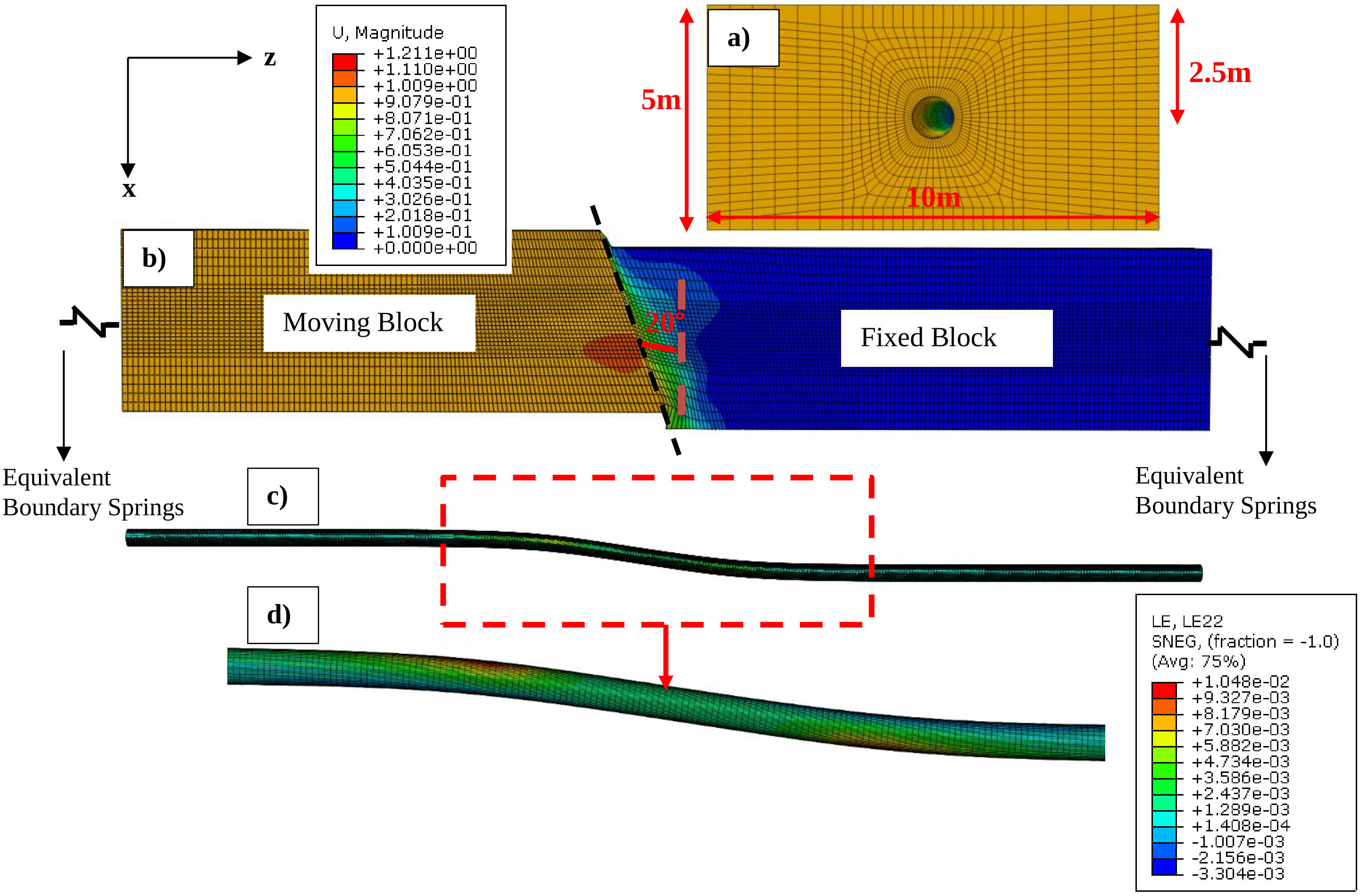

Figure 3. a) Cross-section of 3D Soil Continuum model, b) plan view of the 3D FE model showing displacements of moving block and fixed block, c) displacement profile of the pipeline, d) longitudinal pipe strains in the dashed red zone

Maximum tensile and compressive pipe strains obtained from 3D FE model are compared to analytical methods proposed by Karamitros et al. 2007 and Sarvanis and Karamanos (2017) for validation of the model. Figure 4 demonstrates comparison of maximum tensile and compressive pipe strains obtained from 3D FE model and analytical methods for various fault displacements. The comparison shows that the prediction of maximum tensile and compressive pipe strains are quite consistent with analytical results. The FE results show the same trend as analytical results and the values of compressive and tensile pipe strains are very close, proving that explicit solution technique can be used for this soil-pipe interaction problem.

Table 2. Engineering properties of the soil and pipe and contact parameters used in the 3D FE model

Soil: Medium Sand			
Elastic		Plastic	
E (MPa)	20	φ (°)	32
ϑ	0.35	Ψ (°)	1
		c (kPa)	1
Pipe: Steel Pipe			
Elastic		Plastic	
E (GPa)	210	Yield Stress, MPa (σ_y)	490
ϑ	0.3		
Contact			
Tangential		Normal	
μ	0.3	Hard Contact	

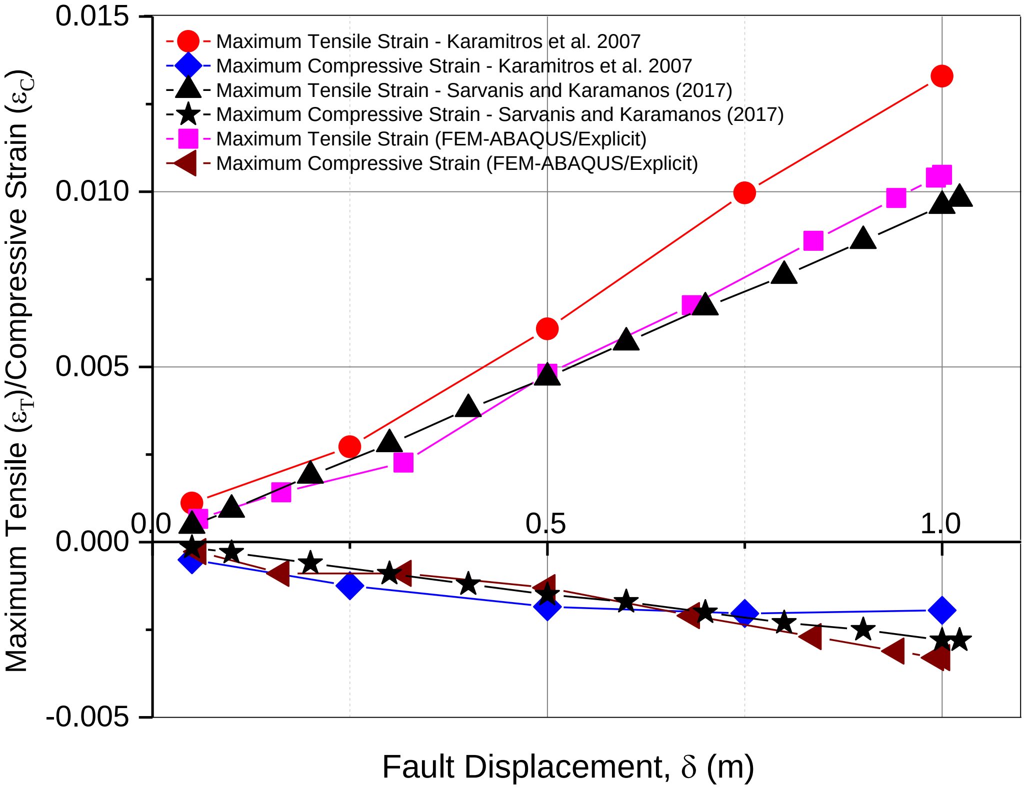

Figure 4. Maximum tensile and compressive pipe strains obtained from FE analysis and analytical methodologies

The energy content for ABAQUS/Explicit model of the pipeline crossing strike-slip fault is shown in Figure 5. Variation of artificial energy (ALLAE), kinetic energy (ALLKE), internal energy (ALLIE) and total energy (ETOTAL) with time can be seen. ALLKE is very small relative to ALLIE, confirming that the soil-pipeline interaction problem is under quasi-static loading. ALLAE is an artificial energy generated by internal hourglass forces and it needs to be less than 5% of ALLIE. In the model, ALLAE is around 8% of ALLIE, which is reasonable.

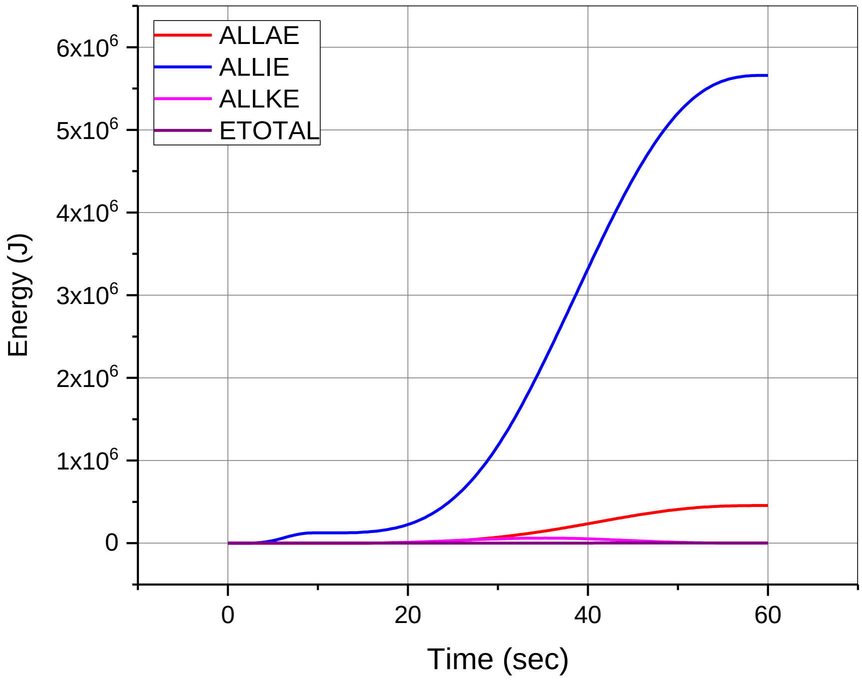

Figure 5. Energy content for ABAQUS/Explicit model, showing variation of ALLAE, ALLKE, ALLIE and ETOTAL with time

4. CONCLUSIONS

This paper focuses on the development of a finite element model using ABAQUS/Explicit module to predict the behaviour of pipelines crossing strike-slip faults. The FE models were validated with ABAQUS/Standard module and analytical results in terms of load-displacement relationships and longitudinal pipe strains for various fault displacements. Good agreement was achieved and the explicit solution technique can be considered as a reliable tool to effectively simulate quasi-static loading problems such as pipelines crossing faults.

5. ACKNOWLEDGMENTS

The authors wish to thank Turkish Government (Republic of Turkey Ministry of National Education) for funding this PhD research study.

6. REFERENCES

ASCE (1984). Guidelines for the Seismic Design of Oil and Gas Pipeline Systems, Committee on Gas and Liquid Fuel Lifelines, American Society of Civil Engineers, Reston, VA.

Karamitros DK, Bouckovalas GD, and Kouretzis GP (2007). Stress Analysis of Buried Steel Pipelines at Strike-Slip Fault Crossings. *Soil Dynamics and Earthquake Engineering*, ASCE, 27(3):200-211.

Lim Y, Kim M, Kim T and Jang J (2001). The Behavior Analysis of Buried Pipeline: Considering Longitudinal Permanent Ground Deformation. *Pipelines 2001*:1-11.

Liu AW, Hu YX, Zhao FX, Li XJ, Takada S, and Zhao L (2004). An equivalent boundary method for the shell analysis of buried pipelines under fault movement. *ACTA Seismologica Sinica*, 17 (1):150-156.

Robert DJ, Soga K, and Britto AM (2014). Analysis of Soil-Pipeline Interaction using ABAQUS/Explicit. *8th Australasian Congress on Applied Mechanics*.

Sarvanis GC, Karamanos SA (2017). Analytical model for the strain analysis of continuous buried pipelines in geohazard areas. *Engineering Structures*, 152:57-69.

Takada S, Hassani N and Fukudu K (2001). A New Proposal for Simplified Design of Buried Steel Pipes Crossing Active Faults. *Earthquake Engineering & Struct. Dynamics*, 30(8):1243-1257.

Vazouras P, Karamanos SA, and Dakoulas P (2010). Finite element analysis of buried steel pipelines under strike-slip fault displacements. *Soil Dynamics and Earthquake Engineering*, 30(11):1361-1376.

Vazouras P, Karamanos SA, and Dakoulas P (2012). Mechanical behavior of buried steel pipes crossing active strike-slip faults. *Soil Dynamics and Earthquake Engineering*, 41:164–180.

Vazouras P, Dakoulas P, and Karamanos SA (2015). Pipe-soil interaction and pipeline performance under strike-slip fault movements. *Soil Dynamics and Earthquake Engineering,* 72:48–65.