Abaqus Technology Brief

TB-10-PSB-1
Revised: March 2010

Modeling the Interaction of Subsea Pipelines with the Seabed

Summary

The interaction of a subsea pipeline with the seabed is a complex phenomenon. Operational loads can cause a subsea pipeline to buckle or “walk” over the seabed, leading to very high pipeline stresses. In some cases however, the buckling phenomena can be beneficially used to relieve excessive stresses by allowing the pipeline to deform at pre-determined locations. The understanding and prediction of these phenomena is therefore crucial for subsea pipeline design.

Background

The nature of the interaction between a subsea pipeline and the seabed depends on the material properties of the pipe and soil, the initial embedment of the pipe, the breakout resistance between the soil and the pipe, and the formation of new soil berms as the pipe moves.

A full three-dimensional Lagrangian simulation of these aspects would require a soil plasticity model in a large strain setting. The size of such problems is prohibitive, and the large plastic deformations of the soil would necessitate repetitive re-meshing as berms form in the soil and get overridden by the pipe.

Alternatively, subsea pipeline analyses can be modeled by representing the pipeline as a one-dimensional beam in space, with the seabed as an analytical or discrete rigid surface that supports the pipeline through contact. The pipeline-seabed interaction can be modeled with a special friction definition that includes the effects of axial and transverse soil breakout resistances, and the effect of the formation and override of new soil berms.

This friction model has been implemented in a user subroutine that is now available for Abaqus/Standard. It includes pipeline-soil interaction behavior based on the observations reported in the SAFEBUCK study [1], and can also model the effect of trench formation when the pipe moves transversely in a cyclic fashion.

In this Technology Brief, we show how the coupled Eulerian-Lagrangian (CEL) method in Abaqus/Explicit can be used to calibrate the parameters that define the pipeline-soil frictional behavior. These parameters are then used in the pipeline-soil friction user subroutine in Abaqus/Standard as part of predicting the in-service buckling deformations of the pipeline.

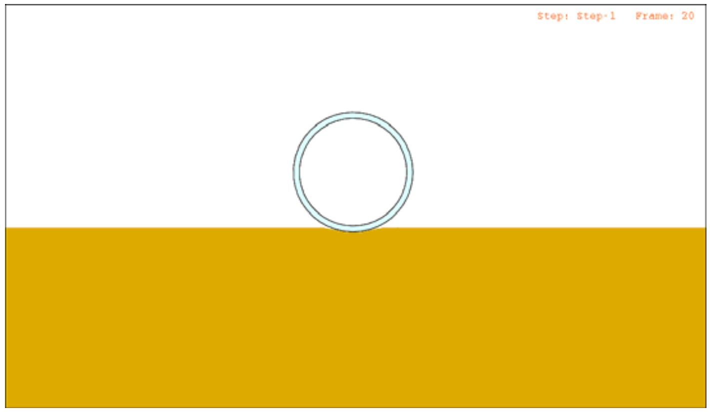

Figure 1: Click to animate

Key Abaqus Features and Benefits

- Coupled Eulerian-Lagrangian method in Abaqus/Explicit for modeling very large deformations
- Quasi-static dynamic analysis capability in Abaqus/Standard
- User subroutine FRIC applied to modeling the special pipeline-seabed interaction

Problem Description and Analysis Approach

Soil berms often get created on the sides of pipelines placed onto the seabed and can form due to sedimentation or small cyclic motion of the pipeline during placement or service. The berms obstruct the large lateral motion of the pipeline; their presence creates what is known as breakout resistance as the pipeline tries to move over the berm. A breakout resistance in the axial direction is also observed, but this resistance is due to increased skin friction that appears when the pipeline has rested on the soil for an appreciable time.

In this analysis we first simulate the formation of soil berms due to small lateral cyclic motion of the pipe; the mechanisms causing lateral breakout resistance of the pipeline are thus accounted for. Values for the axial breakout resistance have been assumed, and we will not simulate the formation of soil berms due to sedimentation.

The analysis considers a steel pipe lying on the seabed that is moved laterally back-and-forth to create soil berms. The pipe partially embeds itself due to this lateral motion. Figure 1 shows an animation of this phenomenon.

The partially embedded pipe is then monotonically moved laterally, and the resistance offered by the soil is monitored. The parameters required for the pipeline-soil friction user subroutine are then evaluated based on how this resistance changes with the lateral motion of the pipe.

The pipeline-soil friction user subroutine is then used in the buckling simulation of the pipeline when it is subjected to thermal and mechanical loads.

Geometry and Model

A 500 m long pipeline is used for the buckling simulation. For the CEL analysis, we consider only a small length of the pipeline as this allows us to obtain the transverse resistance offered by the soil.

The initial configuration of the model for the CEL analysis is shown in Figure 2. The pipe has an outer diameter of 40 cm and a thickness of 2 cm. In order to perform a three-dimensional analysis that is close to a plane-strain condition, we consider only a 5 cm length of the pipe. The Eulerian domain is shown in Figure 3, with the pipe on the right end; the domain is 1.2 m deep and 20 m long. The upper 0.6 m depth of the Eulerian domain is modeled as a void region with no soil material content.

For the CEL analysis, the pipe is assumed to be filled with water at the same pressure as the water on the pipe exterior. The pipe is assumed to be rigid with a reference node defined at the pipe center. The soil is defined with the Drucker-Prager model using a friction angle of 45 degrees and a cohesion value of 100 Pa. Initial geostatic stress conditions are defined in the soil.

For the subsequent Abaqus/Standard pipe buckling analysis, we model the pipeline using PIPE31H elements. The submerged density of the pipe is maintained, and an elasto-plastic material is used for the constitutive behavior. The seabed is modeled using planar discrete rigid elements.

The buckling analysis also includes the effect of a transverse sleeper. It is centrally placed on the seabed and divides the pipeline into two parts; it is used to localize the formation of the pipeline buckle. The sleeper is indirectly taken into account by prescribing a vertical component of displacement at the pipeline center that is retained throughout the analysis.

Figure 3: Eulerian domain with the void region and the placement of the pipe. Void region is yellow, soil region is brown, pipe is at far right.

Analysis Procedure

In Step 1 of the CEL analysis, we apply gravity loading and allow the pipe to come to rest on the seabed. In the next step the pipe is moved laterally back-and-forth using a prescribed velocity with a periodic amplitude. This motion allows the pipe to settle and partially embed into the soil. Berms get created on the sides of the pipe in this analysis step. In the third step, the pipe is moved laterally until it reaches the end of the Eulerian domain.

Parameters governing the pipeline-soil frictional behavior are then obtained by plotting the lateral resistance offered by the soil against lateral pipe motion. These parameters are then used in the pipe buckling analysis in Abaqus/Standard.

Results and Discussion

CEL Analysis

Figure 4 shows the slight deformation of the soil after the pipe has come to rest under gravity loading. Figure 5 shows the soil berms and degree of embedment after the pipe is cyclically displaced. Figure 6 shows the soil configuration as the pipe emerges the trench.

The lateral resistance offered by the soil to the motion of the pipe is a function of the pipe displacement. In Figure 7, the horizontal reaction force at the pipe reference point is plotted against the pipe lateral displacement. After an initial peak, the resistance decreases with increasing displacement. With a further increase in the lateral displacement, the lateral resistance increases again as the pipe collects more soil material as it moves. This increase and decrease is found to occur periodically. As a comparison, Figure 8 shows how the lateral resistance changes for the minimum embedment observed from just the weight of the pipe.

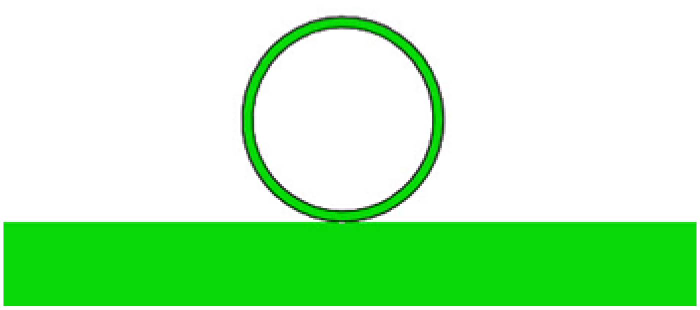

Figure 2: Initial configuration of the pipeline and seabed model for the CEL analysis

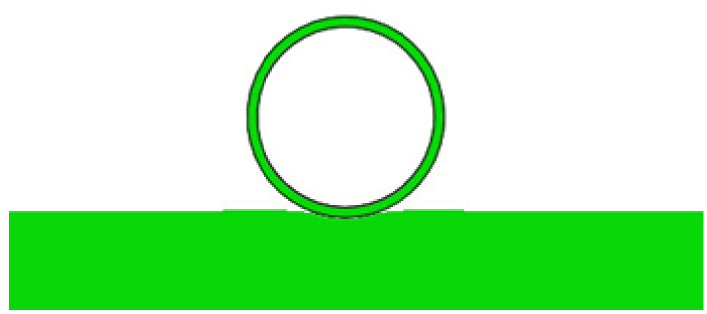

Figure 4: Position of the pipe after it has come to rest on the seabed due to the action of gravity

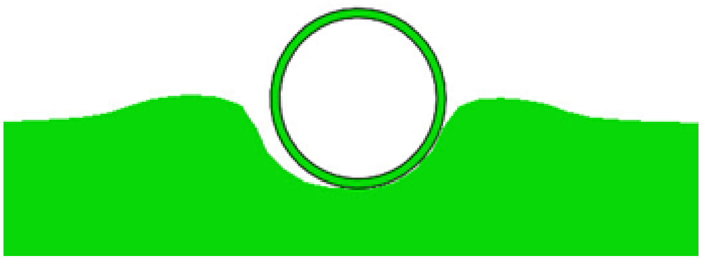

Figure 5: Soil berms and the partial embedment of the pipe after being cyclically displaced

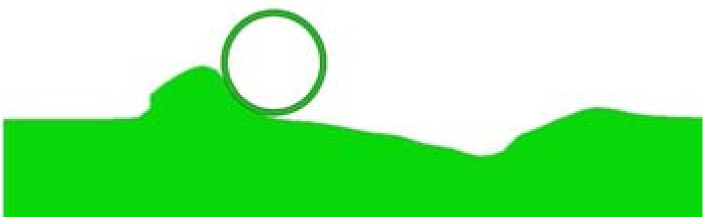

Figure 6: The pipe pushes the berm further laterally and has almost come out of the trench

From the resistance history shown in Figure 7, parameters were selected for the pipeline-soil friction user subroutine [2]. The subroutine requires the specification of the coefficient of friction as a function of slip. Points A, B, C, D, and E in Figure 7 were chosen, and the corresponding (μ, γ) points in the pipeline-soil frictional behavior are shown in Figure 9. The behavior is assumed to be symmetric about point O.

The base friction coefficient, μ_t, was selected to be 0.76, obtained by dividing the minimum lateral resistance experienced by the pipe by its submerged weight.

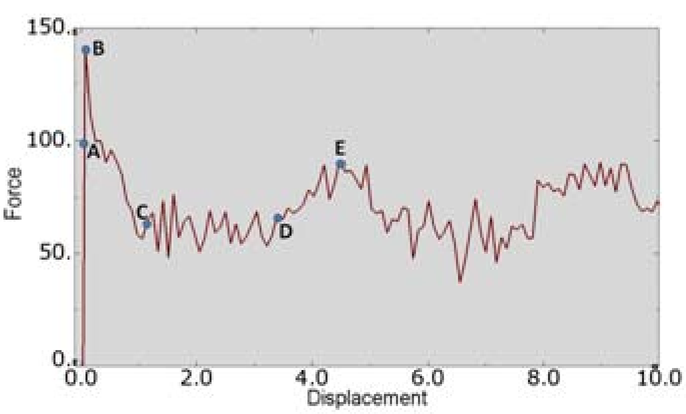

Figure 7: Horizontal reaction at the pipe reference node as a function of the lateral displacement for the large embedment case. Values at points A, B, C, D, and E are used to define the parameters for frictional behavior.

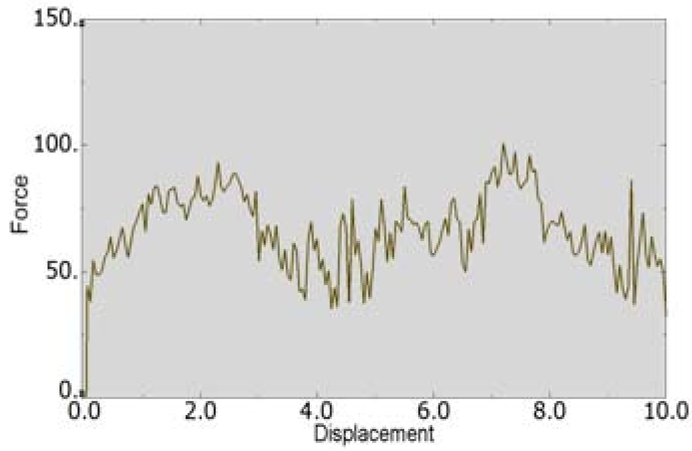

Figure 8: Horizontal reaction at the pipe reference node as a function of the lateral displacement for embedment just due to gravity

Buckling Analysis

The buckling analysis in Abaqus/Standard consists of 13 steps. In Step 1, the pipeline is placed on the seabed and is subjected to gravity loads. In Step 2, the central region of the pipeline is lifted by 2 m to simulate the uplift of the sleeper. In Step 3, the pipeline node on the sleeper is displaced transversely by 1 m to act as a seed for the buckle to form. The ends of the pipeline are fixed axially, and the temperature of the pipeline is increased by 90° C in Step 4. This increase in temperature causes the pipeline to form a buckle in its central region. In Step 5, the temperature is reduced to 0°, and the pipe is subjected to an internal pressure of 10 MPa while the ends are allowed to move axially. These loads and boundary conditions result in a reduction in the buckle magnitude. The temperature is again increased to 90° C in Step 6, while the ends are fixed and the internal pressure is removed. These temperature and pressure cycles are then repeated three additional times. Steps 1 and 2 are performed as static

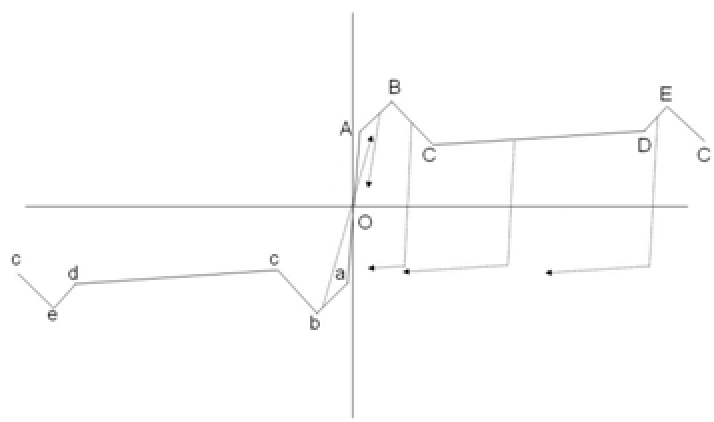

Figure 9: Pipeline-soil frictional interaction behavior in the transverse direction [2]

Figure 10: Contact shear force vectors in the central region of the pipeline

analyses and the subsequent steps are performed as quasi-static dynamic analyses. Breakout resistances for the axial and transverse friction are activated in Step 3.

Figure 10 shows the distribution of the contact shear force vectors in the central region of the pipeline at the end of Step 12. The higher magnitude of contact shear forces near the buckle can be attributed to the higher breakout resistances and also to the higher contact pressure in the region where the pipeline descends down the sleeper and contacts the soil.

Conclusion

In this brief we have described a methodology to obtain the friction coefficient data that is needed to perform a pipeline buckling analysis in Abaqus/Standard. In this approach, we use the coupled Eulerian-Lagrangian technique in Abaqus/Explicit to account for the large plastic deformation of the soil that causes resistance to the large-scale motion of the pipe. The resistance data then defines the frictional behavior that is used in the pipeline-soil friction user subroutine. The buckling analysis is then performed in Abaqus/Standard, capturing the effect of the seabed without physically modeling the soil.

References

1. “The Safe Design of Hot On-Bottom Pipelines with Lateral Buckling using the Design Guideline Developed by the SAFEBUCK Joint Industry Project,” Bruton, D., Carr, M., Crawford, M., and Poiate, E., Deep Offshore Technology Conference - Brazil 2005.

2. “Modeling the Interaction of a Pipeline with the Seabed,” Simulia Online Support System Answer 4094.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 documentation references:

- Analysis User’s Manual
  - ‘Eulerian analysis,’ Section 14.1.1
- User Subroutines Reference Manual
  - ‘FRIC: User subroutine to define frictional behavior for contact surfaces,’ Section 1.1.8

About SIMULIA

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