Helical Buckling of Coiled Tubing in Directional Oil Wellbores

Summary

Coiled tubing is used in a variety of oil well operations including drilling, completions, and remedial activities. For each of these applications coiled tubing offers the benefits of reduced costs, speed, and reduced environmental impact. Coiled tubing possesses a limitation however, in that it may buckle in service. In this situation the tubing may be damaged, and operations may be delayed or disrupted. In this Technology Brief, we provide a methodology for evaluating the buckling behavior of coiled wellbore tube.
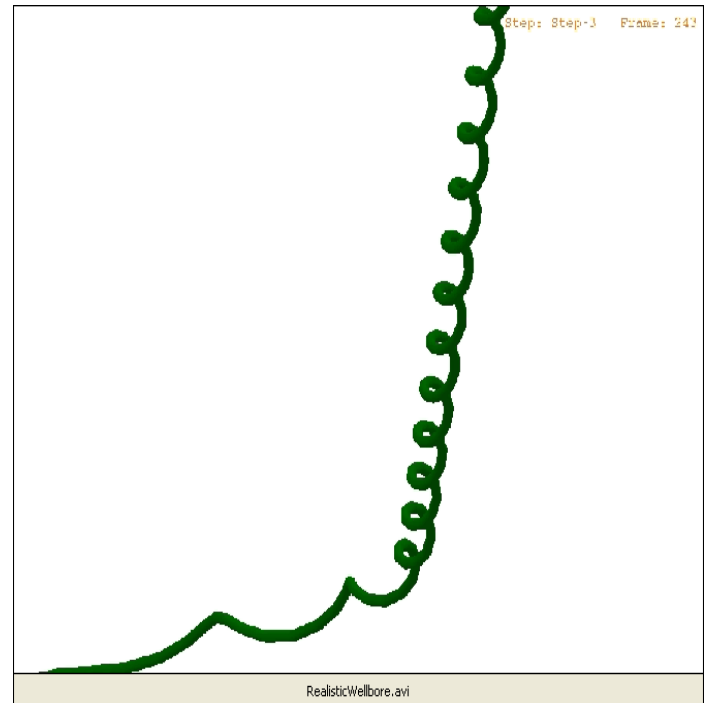
Background

Buckling of a coiled wellbore tube occurs when the axial forces required to produce movement of the tubing within the wellbore exceed a critical level. The coiled tubing first buckles into a sinusoidal shape; as the compressive forces increase, the tubing will subsequently deform into a helical shape. The force required to push coiled tubing into a well increases rapidly once helical buckling occurs. The frictional drag exponentially increases until it finally overcomes the insertion forces, resulting in a condition known as "lock-up" (i.e., the tubing will no longer move further into the well despite additional force applied). In this situation, the coiled tubing may plastically deform or fail from the compounding of stresses related to bending, axial thrust, and pressurization.

The buckling and possible failure of the coiled tubing may prevent the completion of the planned activity and often necessitates an effort to extract the tubing from the wellbore. The financial impact of such an extraction can be significant.

In this Technology Brief, we outline the procedure for evaluating the behavior of coiled wellbore tubing using Abaqus. With this approach the buckling, post-buckling, and lock-up behavior of the drill pipe can be studied. Further, post lock-up methods such as vibration loading and downhole lubrication can also be considered.

Modeling and Methods Development

A modeling methodology is first developed and verified with the results of a laboratory helical buckling test. A coiled tubing insertion into a realistic well is then used to demonstrate these modeling techniques.

RealisticWellbore.avi

Key Abaqus Features and Benefits

- Ability to model actual wellbore geometry and coiled tubing/drill pipe

- Determination of lock-up metrics by modeling post-buckling behavior

- Calculation of anticipated plastic deformation in the coiled tubing

- Inclusion of residual strains in coiled tubing due to placement on reels

- Assessment of post-lockup methods for obtaining extended reach

Verification Model

For the verification study, we follow the work of Salies, et al. in SPE 28713 "Experimental and Mathematical Modeling of Helical Buckling of Tubulars in Directional Wellbores" [1]. In particular, we consider the vertical well application as described in Figure 14 of the referenced document.

Geometry and Model

A steel pipe (commercial stainless steel ASTM 316 seamless), 644-inches long with an outer/inner diameter of 0.25 in./0.21 in. was utilized as the test article. A 644-inch Plexiglas tube with a 3-inch inner diameter was chosen to simulate the confining hole. The steel pipe and Plexiglas were oriented perpendicular to the ground (i.e., in the vertical orientation). The steel pipe was modeled using 3000 three-dimensional linear beam (B31) elements with pipe sectional properties.

The two ends of the pipe are constrained to lie along the centerline (axis of revolution) of the tube. The fixed end is assumed to be fully constrained (translations and rotations) while on the driven-end only translations are constrained. The compression load is applied via a translational displacement. For the experimental test, the loading rate was controlled to avoid any dynamic effects; hence, the analysis was performed as a static procedure.

The interaction between the pipe and Plexiglas was modeled using three-dimensional tube-in-tube interface (ITT31) elements. As reported in [1], a Coulomb friction coefficient of 0.38 was used between the Plexiglas and the pipe.

Analysis Procedure

Introducing a geometric imperfection in the steel pipe is an important part of the solution strategy; without imperfections, only uniaxial compression occurs in a static procedure. In this analysis the imperfections are linear combinations of the eigenvectors of the linear buckling problem. If details of imperfections caused in a manufacturing process are known, it is normally more useful to use this information as the imperfection. However, in many instances only the maximum magnitude (e.g. manufacturing tolerance) of an imperfection is known. In such cases assuming the imperfections are linear combinations of the eigenmodes is a reasonable way to estimate the imperfect geometry [3].

The simulation can be divided into two overall stages: a linear buckling analysis, followed by a post-buckling analysis.

The first ten eigenmodes are extracted during the buckling analysis. The imperfection is introduced as a combination of the second, fourth and sixth modes. The results are sensitive to the size and number of modes utilized for imperfection seeding. It was found in this study that in order to capture helical buckling, it is imperative to seed the mesh not only in the sinusoidal direction but also have imperfections perpendicular to that plane.

Once the imperfections are defined, a three-step static analysis is performed. A gravity-loading step is completed, followed by displacement-controlled end loading to peak displacement. A final step unloads the structure. In each step, only default solution controls are required to obtain converged results.

Results and Discussion

Figure 1 contains an animation showing the isometric and end views of the pipe during the loading and unloading phases of the deformation.

Figure 2 shows the computed and experimentally measured load-displacement behavior. Five stages of the buckling process, A through E, are designated in Figure 2. The following observations are made:

POINT A: The pipe has buckled into a sinusoidal shape and has reached the side of the wellbore.

POINT B: The deformed shape of the pipe starts to transition into a helical pattern.

POINT C: The pipe has approximately 60 degrees of one revolution in contact with the wellbore. The load-deflection response begins to stiffen.

POINT D: A complete 360 degree helix is formed.

POINT E: Deformation state at maximum applied loading.

As seen by the difference in the loading and unloading curves, friction plays a significant factor in wellbore buckling response. During loading, contact forces between the pipe and Plexiglas generate frictional loads. Once the end load is decreased, a hysteresis is shown to develop in the load-deflection curve. In fact, the unloading path was found to represent a similar curve to that obtained in a
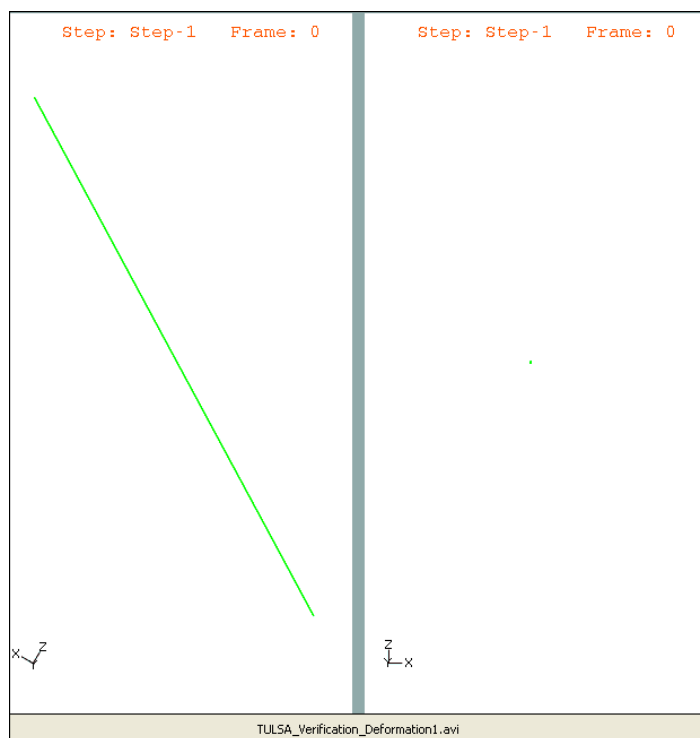

Figure 1: Isometric and end views of pipe helical buckling deformation for the verification model (click to animate)
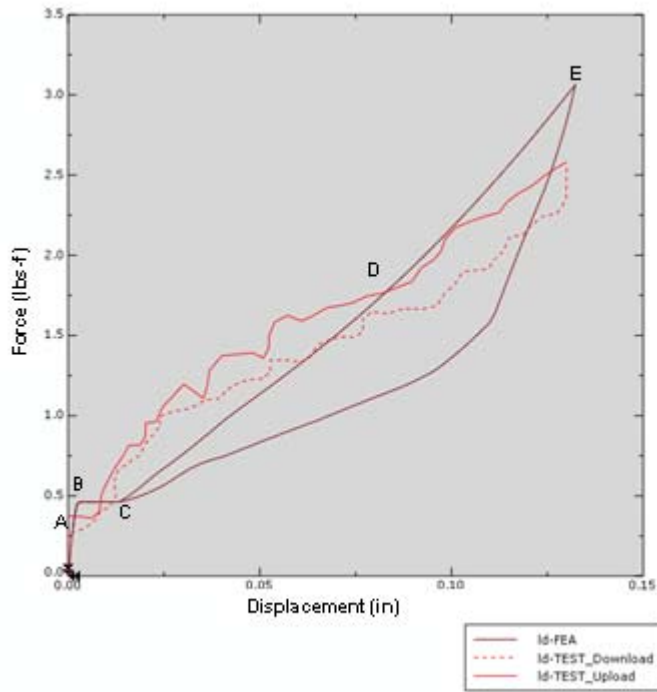
Figure 2: Load-deflection for the verification test. Experimental results [1] are included for reference.

separate zero friction analysis. This is consistent with experimental results [1] where high frequency excitation applied to the test pipe was found to greatly reduce the effective friction coefficient of the load-deflection curve during a loading sequence. Overall, good agreement is found between the experimental and finite element results for this laboratory test.

An analytical expression has been derived in [4] that relates the axial load in a helically buckled pipe with a constant pitch and in the presence of gravity. In the absence of friction between the pipe and confining hole, the value of the section force in a vertical pipe that has assumed a stable helical configuration is given as $2\pi^2 EI/t^2$ where t is the half pitch of the helix, E is the Young's modulus, and I is the second moment of area of the pipe cross-section.

In order to compare with the analytical results, the finite element model is rerun while ignoring frictional effects. The half helical pitch computed between two representative nodes near the vertical center of the pipe axis is found to be 181.34 inches. Using this half pitch value and substituting the values of E and I into the analytical expression, the predicted axial force in the pipe is 1.734 lbf.

From the finite element analysis, the average section force between these two nodes is 1.745 lbf, which is within 1% of the analytical prediction. Thus, the finite element and analytical results are in excellent agreement for the case of frictionless contact.

Realistic Wellbore

In comparison to the verification problem, a realistic pipe insertion analysis has several differences: (1) the primary loading is due to the weight, internal pressure and buoyancy of the pipe; and (2) the drill pipe insertion process must be modeled.

Geometry and Model

Following He & Kyllingstad [2], a representative North Sea horizontal well (2800-m vertical well x 1400-m horizontal with a 600-m turn radius) with a 1.75-in. x 0.134-in. (44.45-mm x 3.4036-mm) [outer diameter x wall thickness] coiled tube in a 6-in. (152.4-mm) ID diameter wellbore casing was studied.

The 4000-m long coiled tubing was modeled using 2250 three-dimensional beam (PIPE31) elements. PIPE31 elements were used instead of B31 in the event that internal pressurization of the coiled tubing is required at a later date (pressurization is not included in the current simulation). For this application, no imperfection is required in the coiled tubing since the curved wellbore will "perturb" the geometry. However, a residual deformation induced in reeling and unreeling the coiled tubing from the spool could be used to seed the coiled tubing mesh.

The steel coiled tubing material is modeled using an elastic constitutive relationship with a Young's modulus of 2.07E11 Pa and a Poisson's ratio of 0.3 with a density of 7810 kg/m^3. For this analysis, the coiled tubing remains well below the elastic limit.

The insertion process is completed as follows:

- The wellbore is extended vertically in its initial state (1800 m above ground level). This extension supports the coiled tubing from excessive sinusoidal buckling before entering the actual wellbore.

- The entire 4000 m long coiled tubing is placed into the model in a vertical orientation and inserted to an initial depth of 2200 m - the point at which the vertical portion of the well begins to transition to horizontal.

- The insertion of the coiled tubing is performed by gravity and by enforcement of a translational degree-of-freedom at one node that displaces the tubing into the wellbore.

As a modeling assumption, gravity is only applied to elements that are below ground at the start of simulation. The interaction between the coiled tubing and wellbore casing was modeled using three-dimensional tube-in-tube interface elements (ITT31). A Coulomb friction coefficient of 0.6 was assumed (consistent with steel-on-steel values available in literature).

Analysis Procedure

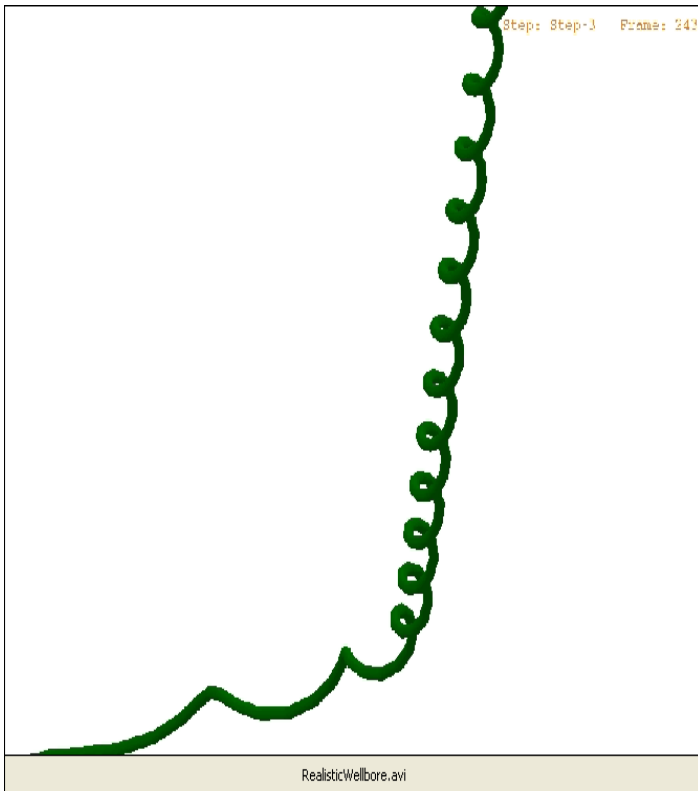The analysis can be broken into a three-stage process:

Figure 3: Deformation in the pipe from Abaqus/Standard analysis of insertion into a realistic directional wellbore (click to animate)

- A static stage to apply gravity on the drill pipe

- A static stage to insert the pipe until the appearance of the first buckling instability

- A dynamic stage to continue insertion until lock-up is achieved

Results and Discussion

Figure 3 contains an animation showing the downward motion of the coiled tubing and its deformation at the vertical to curved section interface up to the point of lock-up. Note the development of the planar sinusoidal buckling before the helical buckling begins to occur.

The load-deflection curve shown in Figure 4 represents the input tension force at the wellhead versus the inserted displacement. Note that based on the gravity load modeling assumption (only elements initially below ground level
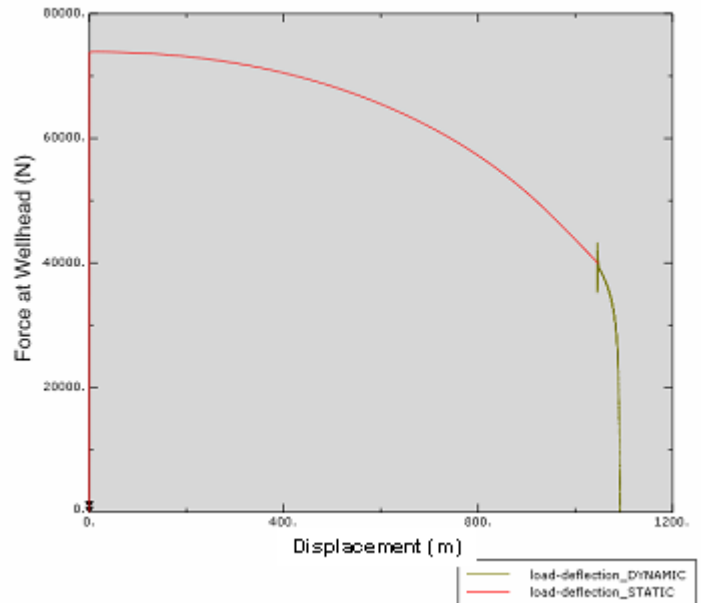

Figure 4: Load-deflection curve from Abaqus/Standard analysis of pipe insertion into a realistic directional wellbore

(2200-m of coiled tubing) are subjected to gravity), the initial tension force starts at ~70000 N. As insertion continues, the tension force is reduced via end-loading and frictional forces. Lock-up is defined when the tension force equals zero. This is consistent with the analysis where significant helical buckling of the coiled tubing and the end of further horizontal reach of the coiled tubing tip into the wellbore is observed. This insertion force-displacement curve matches critical buckling forces in a realistic well [2] subject to these modeling simplifications.

From this point, post-lockup procedures can be investigated (e.g., vibration) to extend the reach of the coiled tubing. Modeling of these techniques was not explored in this current study.

Summary

In this Technology Brief we have demonstrated a methodology for evaluating the helical buckling behavior of a coiled wellbore tube from a finite element modeling approach. While work remains in understanding and preventing helical buckling, the current analyses show that quality results can be obtained in a timely fashion using Abaqus/Standard.

References

1. Salies, J.B., Azar, J.J. and Sorem, J.R., "Experimental and Mathematical Modeling of Helical Buckling of Tubulars in Directional Wellbores," SPE Paper 28713, 1994.

2. He, X. and Kyllingstad, A., "Helical Buckling and Lock-up Conditions for Coiled Tubing in Curved Wells", SPE Paper 25370, 1995.

3. Arbocz, J., "Post-Buckling Behaviour of Structures: Numerical Techniques for More Complicated Structures," in *Lecture Notes in Physics*, Ed. H. Araki et al., Springer-Verlag, Berlin, 1987, pp. 84-142.

4. Tan, X.C. and Digby, P.J., Buckling of Drill String Under the Action of Gravity and Axial Thrust, in International Journal of Solids and Structures, Volume 30, No. 19, 1993, pp 2675-2691.

Abaqus References

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

- Analysis User's Manual

    - 'Static stress analysis,' Section 6.2.2

    - 'Eigenvalue buckling prediction,' Section 6.2.3

    - 'Implicit dynamic analysis using direct integration,' Section 6.3.2

    - 'Introducing a geometric imperfection into a model,' Section 11.3.1

3DS SIMULIA