IPG 2023
23 y 24 de noviembre
Bogotá Plaza Hotel. Bogotá D.C. - Colombia

Con el auspicio de: ASME, arpel
Organiza: ACIEM Asociación Colombiana de Ingenieros

6ª CONFERENCIA INTERNACIONAL GEOTECNIA DE DUCTOS - IPG

IPG2023-0035

FINITE ELEMENT MODELING FOR PIPELINE DESIGN UNDER GEOTECHNICAL INSTABILITY PROCESSES AND IMPACT FORCES

Johan Camilo Garzón Cubides
IGL
Bogota Colombia

Jose Vicente Amortegui Gil
IGL
Bogota Colombia

Antonio Lopez
IGL
Bogota Colombia

SUMMARY

This article shows the results of some practical cases in which the use of the Finite Element Method (MEF) has made it possible to analyze the stress-deformation effect generated by displacements caused by processes of geotechnical instability and impact loads on hydrocarbon transport infrastructure. The work is divided into two parts; the first presents the methodology used for the development of numerical models, focusing on geometry, boundary conditions, the choice of constitutive models and the different elements that represent the pipeline infrastructure, including steel pipes and the types of analyses carried out, elastics, elastoplastics and limit analysis using MEF. In a second section, some practical cases will be shown, including the damage suffered by valve stations due to explosions and the possible materials that can be used to dissipate the energy of impact forces, examples of buried welded steel pipes subject to displacements generated by processes of instability and the planning of geotechnical works and actions to safeguard the integrity of the pipes

1. INTRODUCTION

In general, the results of geotechnical analyses present great uncertainties, which in many cases are reduced by the experience of engineers. However, these uncertainties are associated with the fact that the materials are heterogeneous and their geomechanical behavior is complex. Estimating uncertainty has been a task that has interested many geotechnical engineers for some time. For example, Casagrande [1] has pointed out the importance of evaluating not only the risks and calculations inherent to geotechnical works, but human faults must also be evaluated. In recent decades, progress has been made to include probabilistic and statistical techniques in geotechnical analyses to manage uncertainty, many of these methodologies are already requirements of some technical standards.

Among the advances in knowledge to reduce uncertainties, it is to understand and simulate the stress-deformation behavior of materials using constitutive mathematical models, which have been integrated into finite element analysis (MEF). This method was introduced by Clough and Woodward in 1967, in which a mass of soil is divided into discrete units called finite elements, which create a mesh containing predefined nodes and edges. This method uses the formulation of displacements to represent the efforts at each node.

In oil pipeline geotechnics, the use of the finite element method has begun in recent years as an answer to understanding the interaction of an element of high stiffness (pipe), as it is embedded in a mass of soil that is constantly changing.

2. FINITE ELEMENT PROGRAM

To illustrate some examples of pipeline geotechnics, the OptumG2 program was used (Krabbenhøft et al., 2019), which models the stability of two-dimensional models using the finite element method combined with limit analysis (plasticity theory).

OptumG2 is a two-dimensional finite element computing program applied to Geotechnical Engineering based on numerical techniques. The OptumG2 finite element analysis algorithm performs direct upper (for a kinematically possible field) and lower limit (for a statically admissible field) calculations combined with the Strength Reduction technique of stability problems, including stability of structures, retaining walls, slopes and artificial slopes, foundations, among others. In addition, it allows elasto-plastic analyzes to be carried out with a variety of constitutive models (stress - deformation relationships) for the design of pipelines in the face of geotechnical instability processes and impact forces.

3. ANALYSIS MODELS

To illustrate the advantages of the finite element method in real cases, the results of the analysis of the stress-deformation effect generated by displacements caused by geotechnical instability processes and impact loads on the hydrocarbon transportation infrastructure are presented below.

3.1 Impact forces on hydrocarbon transportation infrastructure

Valve stations are protective structures for hydrocarbon transport infrastructure, mitigating the vulnerability of the system. During operation, these structures are susceptible to impacts from detonations or rockfalls, so they are designed to withstand these impacts and maintain protection over the transport structure after impact.

In the case of explosive elements, the impact generated by the shock wave on the surface of the valve station can be simulated following the Kingery-Bulmash equations for calculating blasting overpressures [3].

Plasticity analyzes were carried out on different configurations of protective walls, considering variations in the geometry and materials that compose them, looking for the best protection alternatives. FIGURE 1 shows one of these models where the behavior of a gabion wall 2 m thick by 3 m high is simulated.

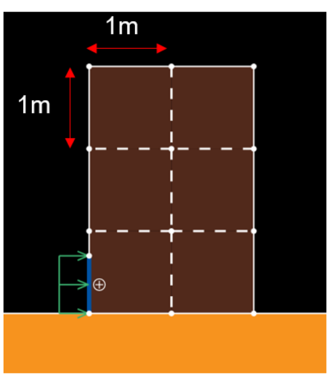

FIGURE 1 INITIAL ANALYSIS MODEL

The results of the modeling are presented in FIGURE 2, where the displacements produced by the impact that result in the loss of part of the material that made up the wall without generating instability in the wall are highlighted (1). The loss of material obtained from the simulation matches what was observed in the field (2).

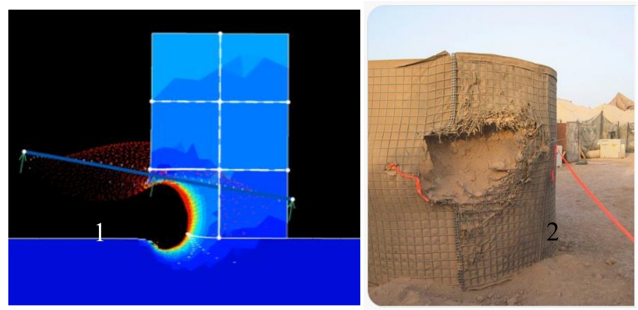

FIGURE 2 COMPARISON OF SIMULATION RESULTS AND FIELD RESULTS.

The FIGURE 3 presents the result of the simulation using a metal pipe lattice with bags of cement floor (1), a gabion wall of 2.0 m of base by 3.0 m of height (2), a reinforced earth wall of 4.0 m of base and 5.0 m of height (3) and reinforced concrete walls (4). The results are consistent with the stiffness and weight of each material, as well as the geometry of the wall section.

From the results, it is concluded that the greater the mass of the wall, the lower the displacements caused by the impact. This is observed when comparing the displacements produced in the concrete wall and the wall on reinforced earth. In addition, the geometry of the section changes the displacements, as can be seen when comparing the performance between the 2 m and 3 m wide gabion wall. In the latter a smaller displacement is observed.

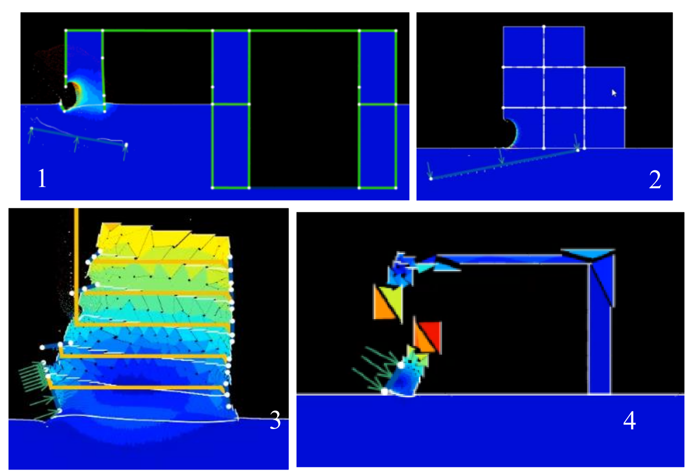

FIGURE 3 SIMULATION RESULTS FOR DIFFERENT WALL CONFIGURATIONS

3.2 Earth retaining structures.

To form a Right of way of a flow line, a gabion-like gravity structure was installed to confine the fill and the pipe was installed at the top of the wall above the ground surface.

The gabion wall has a length of 50 m, made up of 5 levels and has a variable height with a maximum of 5 m. Due to a possible increase in pore pressure, in the central area of the retaining wall, there is evidence of bulging on the face of the structure, which have caused settlements in the upper part of the slope, putting the flow line at risk, as shown in FIGURE 4.

FIGURE 4 ABOVE: THE GABION WALL IS OBSERVED WITH A DEFORMATION IN ITS HEAD. BELOW: THE EXISTING FLOW LINE THAT WAS INSTALLED AT THE TOP PART OF THE WALL.

To prevent further increases in the displacements of the containment structure, it is recommended to install additional gabions that support compressive loads, arranged in the form of struts, as illustrated in FIGURE 5.

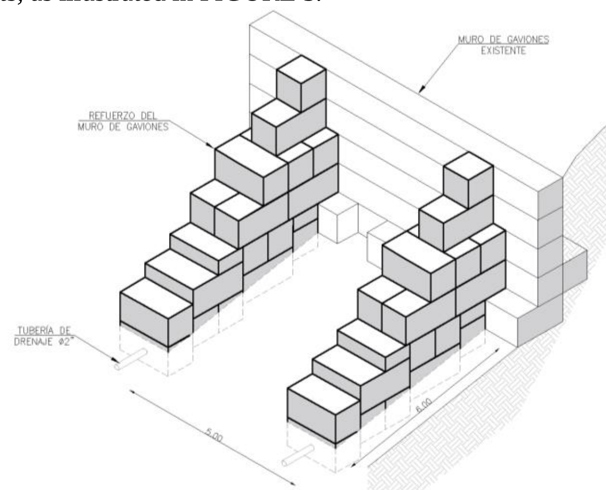

FIGURE 5 GABION WALL REINFORCEMENT (STRUTS).

For the design of the containment structure for the right of way, a three-dimensional finite element model was proposed, in order to determine the Safety Factors of the current condition and with the implementation of the recommended works, as shown in FIGURE 6. The properties of the materials are presented in TABLE 1.

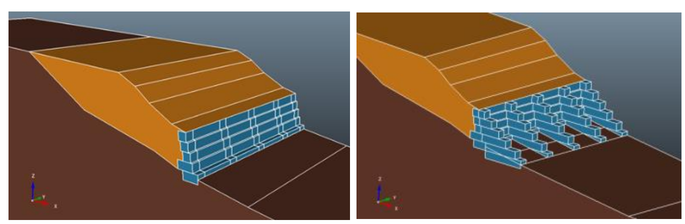

FIGURE 6 THREE-DIMENSIONAL ANALYSIS MODEL. ON THE LEFT IT IS SHOWN ON THE WALL IN THE CURRENT CONDITION AND ON THE RIGHT WITH THE REINFORCEMENT.

Material	E (MPa)	Unit weight (kN/m3)	C´ (kPa)	ϕ (°)
Soil Natural	25	18.0	fifteen	32
Fill Material	30	19.0	5	30
Gabions	fifty	17.0	70	60

TABLE 1: MATERIAL PROPERTIES

In the analysis model, the horizontal displacements of the retaining wall and the settlements that occurred in the upper part of the structure were evaluated. According to the analysis, the settlements are 7 cm, as illustrated in FIGURE 7.

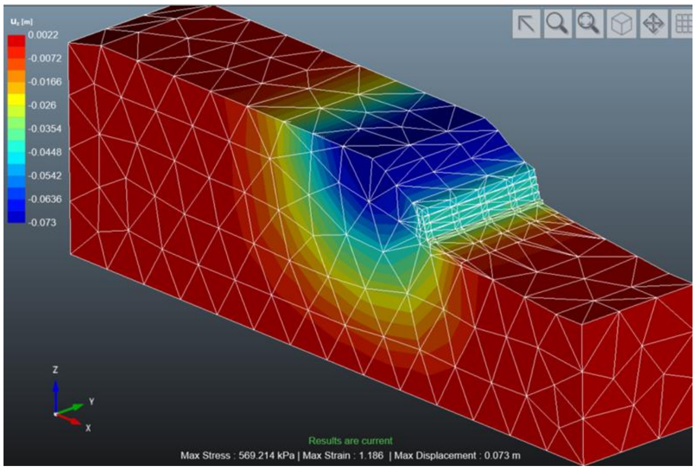

FIGURE 7 VERTICAL DISPLACEMENTS OF THE MODEL.

FIGURE 8 shows that the Safety Factors for the current condition are 1.2 and for the condition with struts it is greater than 1.5. It is detailed that for the latter, the possible failure surface is located on the hillside above the gabion wall, but does not involve the area where the flow line is located.

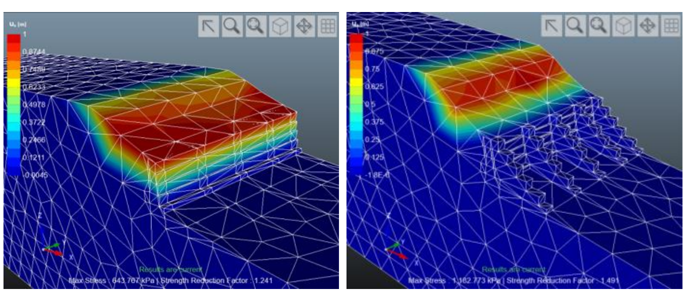

FIGURE 8 SAFETY FACTORS FOR THE CURRENT CONDITION AND WITH THE RECOMMENDED WORKS.

3.3 Pipe subjected to landslide.

In a process of instability, the earth pressure causes an initial deflection of the pipe, which offers resistance and opposes the movement of the ground. However, the ground surrounding the pipe can reach the state of failure due to the punching exerted by the pipe lengthwise, so the earth pressure that acts on the pipe corresponds to a passive earth pressure applied to the longitudinal area of the pipe.

Therefore, in the event of a landslide, a steel pipe is able to resist the applied external forces and in response it deforms in the direction of the movement of the ground, as illustrated in the FIGURE 9. This figure shows that a pipe moved approximately 8 meters in response to the earth pressure caused by a slide that impacted the pipe and its integrity was not affected.

FIGURE 9 DISPLACEMENT OF THE PIPE, DUE TO AN INSTABILITY PROCESS.

To analyze a similar situation as the one that occurred in the pipeline in Figure 1, three-dimensional modeling was carried out to estimate the deformations of a 24” pipe that is buried, which is subjected to permanent ground deformation. The analysis model is shown in FIGURE 10 and TABLE 2 summarizes the properties of the soil and the pipe used in the analysis.

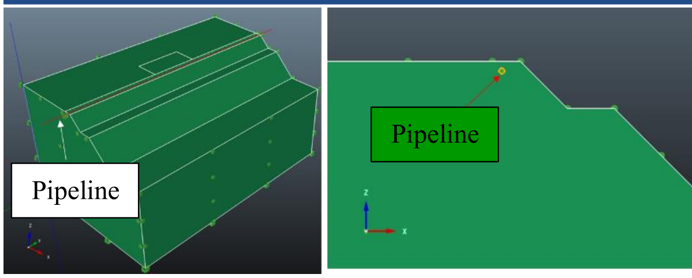

FIGURE 10 THREE-DIMENSIONAL MODEL IN THE OPTUM G3 PROGRAM TO DETERMINE THE BEHAVIOR OF A PIPE IN AN INSTABILITY PROCESS

Material	E (MPa)	Unit weight (kN/m3)	C´ (kPa)	ϕ (°)
Floor	30	18.0	10	35
Pipeline	E (MPa)	Moment of inertia (m4/m)	Creep strength (MPa)	Area section (m2/m)
API 5LX65	2E+05	7E-08	448	9.525E-3

TABLE 2: PROPERTIES OF THE SOIL AND THE PIPE USED IN THE ANALYSIS.

As a result of the analysis, FIGURE 11 shows a sequence of images that represents the deformation of the terrain, due to an instability process. The maximum horizontal displacement is 0.98m in a width of 20m.

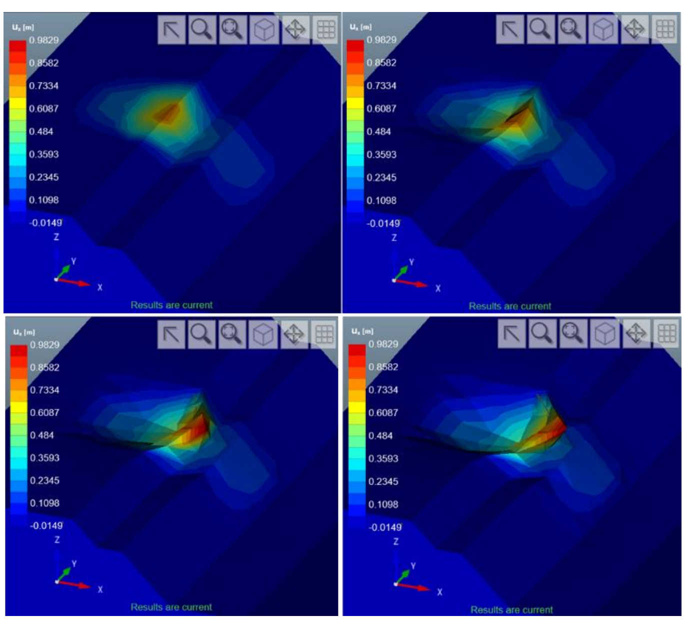

FIGURE 11 HORIZONTAL DISPLACEMENT IN THE GROUND DUE TO A PROCESS OF INSTABILITY.

In response to this movement, the 24" pipe displaces 0.90m in a section of 36m. Due to the pipe have greater rigidity than soil, it distributes the ground pressure over a greater width than where the ground movement occurs, as shown in FIGURE 12.

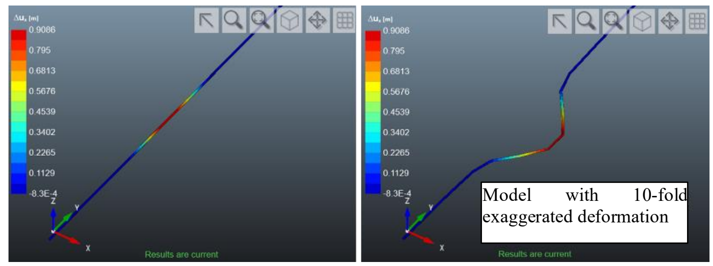

FIGURE 11 HORIZONTAL DISPLACEMENTS OF THE 24" PIPE, AS A RESPONSE TO THE THRUST OF THE GROUND.

By including the resistance and deformation properties of the pipe within the analysis model, it is possible to obtain the bending moments and axial loads that the pipe suffers in the event of an instability process. In this case, the maximum bending moment obtained from the finite element analysis is of 9340 kNm/m and an axial force of 8938 kN/m. In addition, the energy dissipation on the pipe can be obtained, that is, the sections in which a stress concentration occurs can be identified, as seen in FIGURE 13.

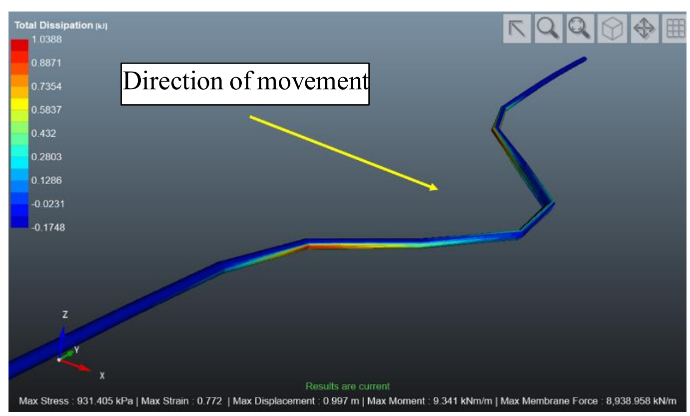

FIGURE 12 RESULTS OF THE DISSIPATION OF ENERGY ON THE PIPE.

4. CONCLUSIONS

With the finite element method, it is possible to model heterogeneity, anisotropy, complicated boundary conditions, complex loading conditions and arbitrary geometries.

Unlike limit equilibrium methods, the finite element method does not require making assumptions about the shape of the failure surface prior to carrying out the analysis. The "shape" of the failure surface is a result of finite element analysis and can be observed in the displacement or by deformation.

According to the results obtained from some practical cases in which the Finite Element Method (FEM) was used, it was possible to analyze the stress-deformation effect generated by displacements caused by geotechnical instability processes and impact loads on the infrastructure of hydrocarbon transportation, which help reduce uncertainty in works designs.

By being able to include the resistance and deformation properties of the pipes in the Finite Element analyses, it is possible to carry out the soil-pipe interaction, to obtain not only ground displacements, but also the bending moments and axial loads that the pipe is subjected to ground instability processes.

REFERENCES

[1] Casagrande B. Role of the Calculated Risk in Earthwork and Foundation Engineering,Journal of Soil Mechanics and Foundations. Proceedings of the American Society of Civil Engineers, ASCE, volume 19, 1965

[2] Krabbenhøft K, Galindo S, Zhang X, Krabbenhøft J. AUS: Anisotropic undrained shear strength model for clays. International Journal for Numerical and Analytical Methods in Geomechanics, volume 43, issue 17, 2019.

[3] Kingery C, Bulmash G. Air blast parameters from TNT spherical air burst and hemispherical surface burst. Technical report ARBL-TR-02555.