MecSol2013
International Symposium on Solid Mechanics
April 18-19, 2013
Porto Alegre, Rio Grande do Sul, Brazil

Fitness for Service Assessment in a Piping System of an Hydrotreating Unit Resulting from Increased Flow

Hervandil Morosini Sant'Anna
Refining Department
Petrobras
hmsantanna@petrobras.com.br

Murilo Fonseca Leal
Refining Department
Petrobras
muriloleal@petrobras.com.br

ABSTRACT

Aiming to increase operational efficiency (and, consequently, to increase the profits related to the fuel production), the optimization staff of a refinery (management responsible for production scheduling, analysis of the production process and product quality.) decided to increase the flow rate of a flow line of a gas fuel cooling system in a Hydrotreating Unit.

During preliminary tests, in was observed the occurrence of vibrations, whose displacement amplitudes were measured at certain points in the field.

Thus, to allow the testing without compromising the structural integrity of the line, or even present a risk to personal safety and the environment, an evaluation was performed using the Finite Element Method through Modal and Harmonic analysis, determining the dynamic behavior by checking the state of stresses and displacements at the pipe.

The results of numerical analysis were compared with observations made in the field, and a methodology to correlate the flow of the stream with the offset in the line was suggested.

Keywords: hydrotreating, tubing, finite elements method, harmonic analysis.

1 INTRODUCTION

In general, any industrial activity aims, ultimately, the profit. In the petroleum industry such a rule is not different: it is not rare equipment be used beyond their design limits in order to increase their efficiency and therefore production.

When done in a systematic manner, generally problems are not expected, because the mechanical equipments, without any exception, are designed using safety factors. To this end, before modifying a process variable (pressure, temperature and flow, to name the most common ones), tests are performed in order to make sure that there will not be any unforeseen problems during normal operation.

To this end, a Petrobras refinery has recently conducted a flow increasing test in a pipe system in a Hydrotreating unit. However, during the test was noticed a significant increase in the amplitude of displacements in pipe sections. For security reasons, the tests were paralyzed, and it was decided to develop a more detailed study to determine the causes (and consequences) of such vibrations, seeking any correlation between the flow and the observed shifts in structure.

This study aims to determine a way to perform the test flow in a safe manner, avoiding excessive vibration or other structural damage to the pipes. An indirect correlation was determined, ie, considering the hypothesis that the increased flow in the piping system is directly related to the increase in vibration, we just have limited this vibration. So, if for a given flow rate the displacement offset at some control point reach a certain threshold value, this would indicate that the structure would be close to its resonance.

To this purpose, a model was built using the Finite Element Method for performing dynamic analysis (modal and harmonic). As will be seen, the displacements obtained in the model had a good correlation with the values obtained in the field (both in direction and in magnitude), for a given flow value.

In industry, this type of assessment is referred "fitness for service". In the following section shall be defined some basic concepts about the subjects developed in this study.

2 BASIC CONCEPTS

2.1 Process Piping

Basically, process piping are equipments designed to carry fluids from one location to another within the industrial plant. They differ from pipelines, since the latter are used to transport fluids outside industrial area to which it belongs, and even having different failure mechanisms [1].

In relation to the assessments that can be made, whether the project itself as studies of fitness for service, the following parameters should be checked:

i. The proper design code;
ii. Design pressure and temperature;
iii. Material type and their thermo-mechanical properties, including those belonging to clads or linings;
iv. The piping thickness and length;
v. The piping geometry (curves, loops, straights, etc);
vi. Supports;
vii. Allowable stresses.

The following pictures show some details of the process piping assessed in this study:

Figure 1: Overview of pipe system close to the heat exchanger.

Figure 2: Portions of the piping exiting the pressure vessel.

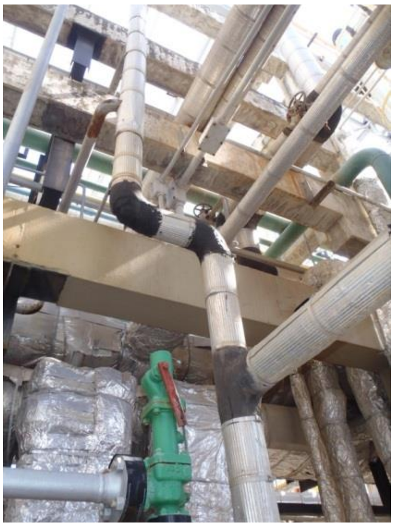

Figure 3: Pipe detail near the heat exchanger.

Figure 4: Detail of one of the pipe brackets, next to the heat exchanger.

2.2 Hydrotreating Unit (HDT)

The hydrotreating process aims to stabilize and remove undesirable contaminants from petroleum products (usually naphtha, kerosene, diesel oil or lubricating oils), by the hydrogenation of olefins, aromatics, sulfur compounds, nitrogen compounds, oxygen compounds, organochlorine compounds and organometallic compounds under appropriate conditions of pressure and temperature [2].

It is a very important process unit since:

i. Enables compliance with the increasing demands of environmental and occupational health, by reducing emissions and toxicity of products;
ii. Improves product quality by the saturation of olefins (causes instability), removal of sulphur (causes corrosion and pollution) and nitrogen (causes instability and poisoning of catalysts), among others.

Figure 5 shows a basic flowchart of a typical HDT unit:

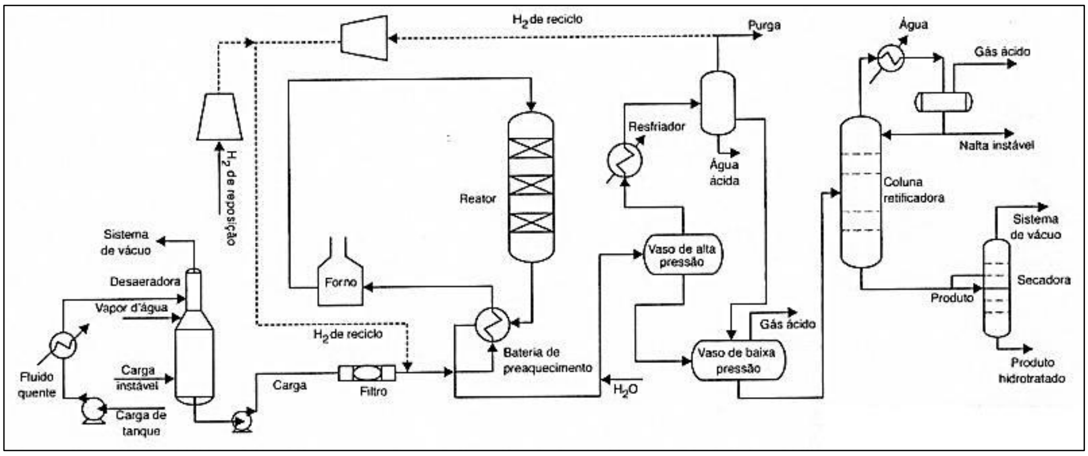

Figure 5: Typical HDT unit flowchart.

The unit operates primarily in two stages, the *Reaction Section*, in which the impurities present in the hydrocarbon molecules are broken in the presence of hydrogen at high temperatures and pressures and separated for elimination, and the *Section of Gas Separation and Fractionation,* in which treated hydrocarbons are separated and recovered.

2.3 Fitness for Service

Fitness for service can be defined as a set of quantitative methods used to determine the structural integrity of equipment with some sort of defect or those operating outside their design conditions, ensuring that it can continue with its safe operation, or if some sort of intervention is needed [3].

Fitness for service is a very important step in the overall product process assessment. The "health" and "longevity" of a mechanical equipment is equivalent to what occurs in humans: it depends on your *genome*, ie, their design, materials and forms of construction; of your *lifestyle*, in this case, the way in which the equipment is operated, within the conditions for which it was designed and, not least, the quality of their *medical examinations*, ie, how the equipment is maintained and periodically inspected.

In this study, a particular case of damage mechanism is assessed: *vibration*. To do so, the main question to be answered is: “was the piping system designed to withstand such amplitude of vibrations?" In his book, Antaki sets the excessive vibration as a special case of overloading, which is understood as any kind of load (force, moment, displacement or rotation) that exceeds the design limits.

2.4 Mechanical Vibrations Analysis

Briefly, mechanical vibration can be defined as the oscillation of a point on a structure around a reference. Vibration analysis is of fundamental importance for many different areas of engineering, helping in predictive maintenance of machinery, construction of large civil engineering structures and studies of resistance of materials among other examples.

The next expression is the basic movement or dynamic equilibrium equation:

$$M\ddot{x}(t) + C\dot{x}(t) + Kx(t) = F(t) \tag{1}$$

Where: M, C and K are, respectively, the mass, damping and stiffness matrices. F is the external forces vector. If a *modal analysis* is performed, F(t) = 0. By the other hand, if an *harmonic analysis* is done, then F(t) has a sine shape.

As a last concept for this subject, *resonance* is the tendency of a system to oscillate at maximum amplitude at certain frequencies or wavelengths, known as *resonant frequencies*. At these frequencies, even small periodic forces can produce large amplitude vibrations, because the system stores vibrational energy. When the damping is small, the resonance frequency is approximately equal to the natural frequency of the system which is the frequency of free vibration, determined by modal analysis.

3 METODOLOGY

Assess vibrations in equipments, resulting from changes in the flow of fluids is a complex task, because there are two coupled physics in the phenomenon [4]: Fluid and Solid Mechanics. To this end, studies on fluid-structure interaction are required. However, the use of this technology still has its limitations when used in non-academic problems, because the math involved is extremely complex and the simulations require a time often excessive front the immediate needs of a real problem occurring in the industry.

To overcome this issue, we have the Finite Element Method: when properly used, it has proven to be a remarkably agile and flexible tool, since it allows evaluating a wide range of situations and problems at a relatively small computational cost.

As reviewed, this study performs an indirect assessment between the flow and vibration by controlling the displacement induced by resonance in some parts of the line (a control point or probe). For this purpose, harmonic analysis is performed to calculate stresses and displacements, these later limited to avoid the resonance frequency.

Such indirect correlation between the flow and the vibration in the line was conceived as follows: before the test, displacements are calculated at the resonance of the structure. While increasing the flow, the inspection measured the displacement induced by vibration. In other words, if a certain flow caused the displacement predicted to the resonance range, this flow value should be avoided (or at least should be quickly passed by this value), in order to not activate the resonance in the structure.

3.1 Problem Description

The line assessment was performed through the Finite Elements Method [5], using the copyrighted software Ansys Mechanical APDL R14.

A pipe model was built using the element PIPE289, a modified Timoshenko beam element, specific for using with piping loads, such as internal or external pressure, and variable temperature across the cross section.

The following analyses were performed:

- Modal Analysis: free vibration assessment, in order to identify the natural frequencies and the mode shapes;
- Harmonic Analysis: forced vibration, with loads changing in time according to a sinus function, at the frequency range obtained from the modal analysis. Displacements and stresses are obtained using this approach.

Figure 6 shows part of the piping system. This draw was obtained from the refinery inspection group, in order to represent the measured displacement:

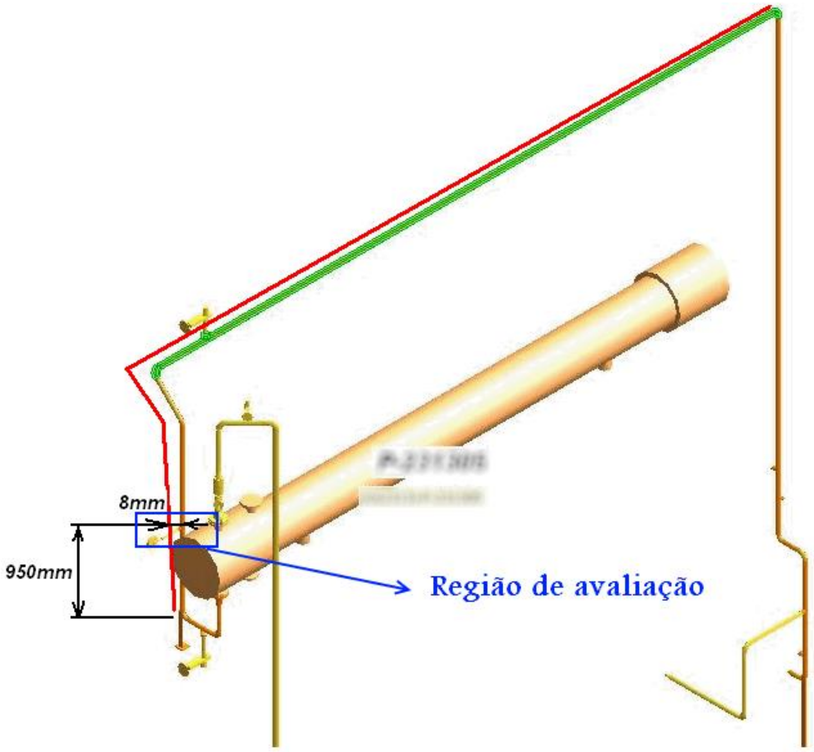

Figure 6: `Point where the displacement was measured.

3.2 Materials, Boundary Conditions and Finite Elements Model

The material is the carbon steel SA-106 gr. B. Mechanical properties were obtained from the design code ASME B31.3 for the operational temperature of 44°C:

- Young modulus: 200000 MPa
- Poisson Ratio: 0,3
- Specific weight: 7850 kg/m^3
- Ultimate tensile stress: 415 MPa
- Yield stress: 240 MPa
- Allowable design stress: 137 MPa

The pipe system is thermally isolated with a 25 mm thick calcium silicate coating, whose specific weight is 245 kg/m^3.

Figure 7 represents the piping cross section with its calculated geometrical properties:

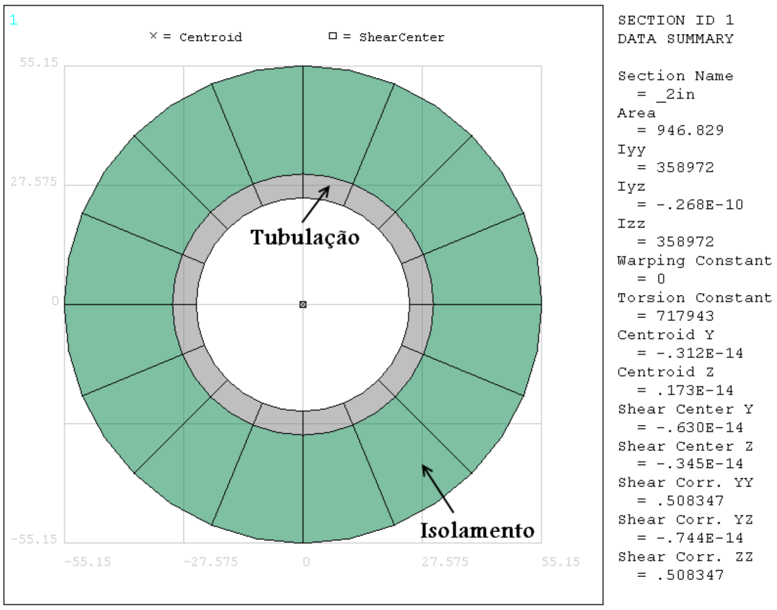

Figure 7: Cross section and its geometrical properties (basic unit: mm).

Loads considered are: *inner pressure* (4,7 kgf/cm^2), *operational temperature* (44°C) and *self-weight* (calculated by Ansys, based on the volume, gravity and specific weight of both steel and refractory). Further, the *fluid weight* is considered, through its self-weight (1000 kg/m^3).

Figure 8 shows the finite element model, with the boundary conditions. Mesh details are also shown in Figure 9:

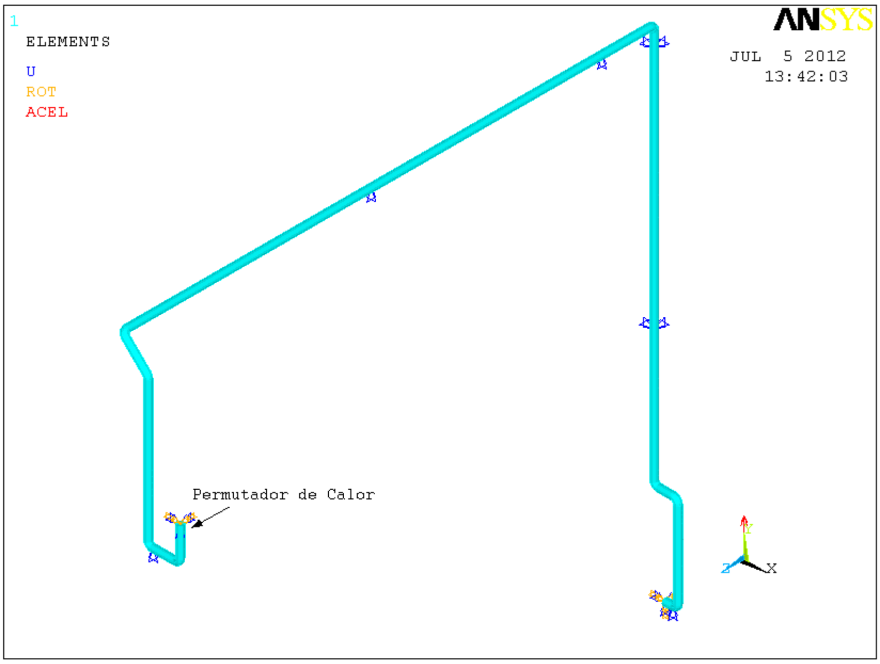

Figure 8: Finite element model, showing some boundary conditions.

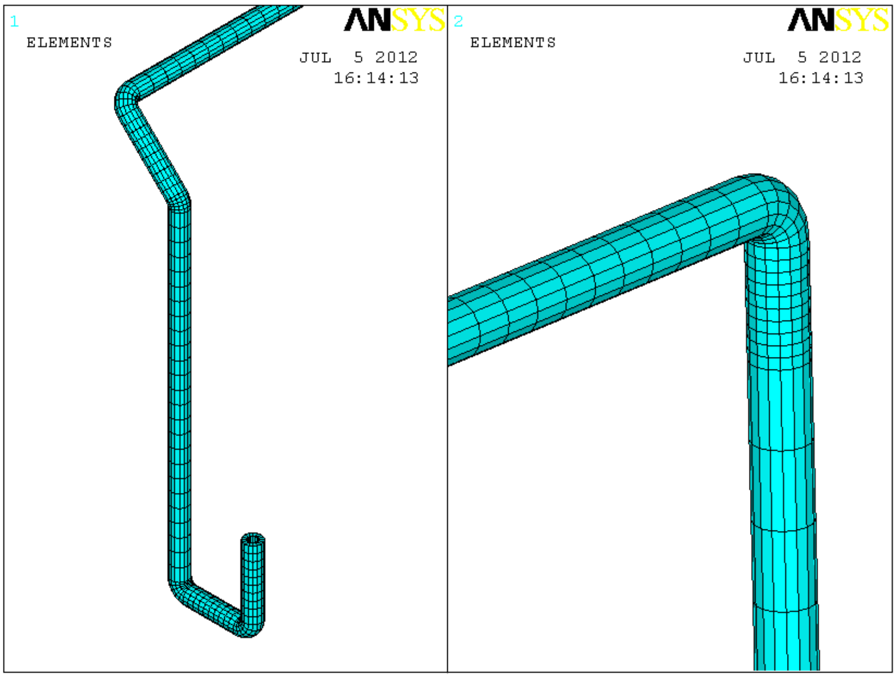

Figure 9: Finite element mesh details.

4 RESULTS

4.1 Modal Analysis

The next pictures show the first three vibration modes and natural frequencies obtained from the modal analysis. Frequencies, highlighted in the figures, are calculated in Hz. The mode shapes are shown together with the non-deformed structure (black line):

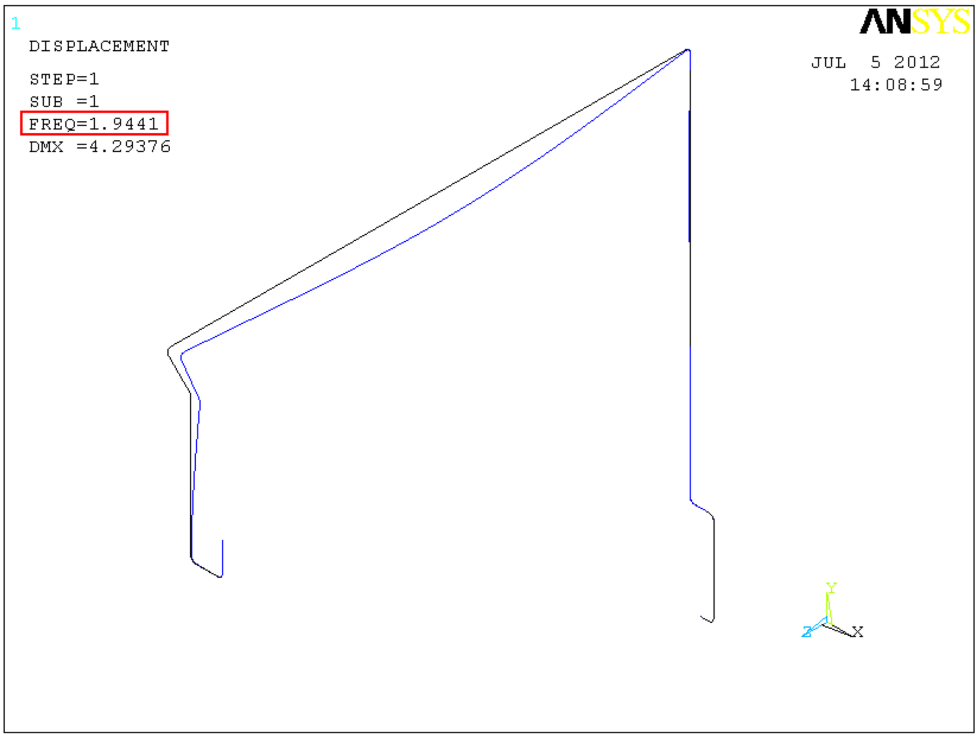

Figure 10: 1st mode shape. f = 1,9 Hz.

Figure 11: 2nd mode shape. f = 4,3 Hz.

Figure 12: 3rd mode shape. f = 9,4 Hz.

One can see that these mode shapes are promoting bending moments in the vertical tube attached to the heat exchanger.

4.2 Harmonic Analysis

The next pictures show the harmonic analysis results. The target frequencies are those obtained from the modal analysis, and the probe point is the one shown in Figure 6 (the point whose displacement was measured by the inspection).

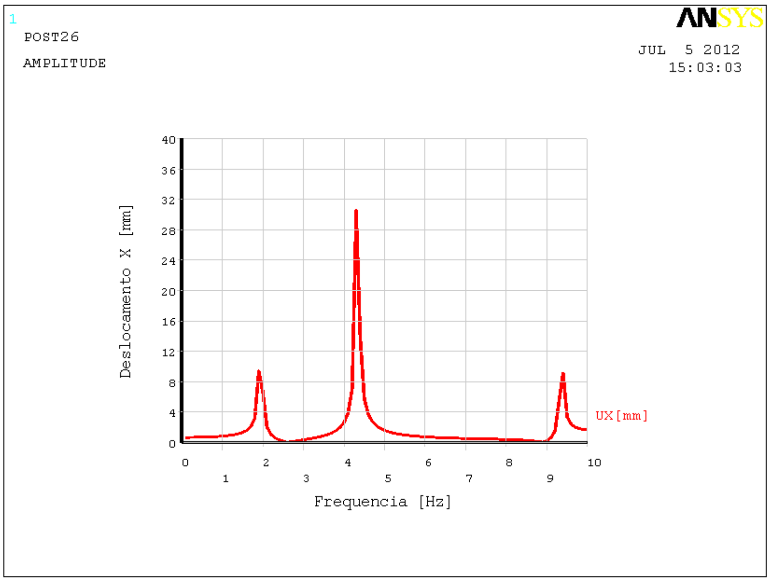

Figure 13: Displacement X frequency plot, showing the resonance range.

Next pictures show the displacement and the stresses calculated for the two first resonance frequencies:

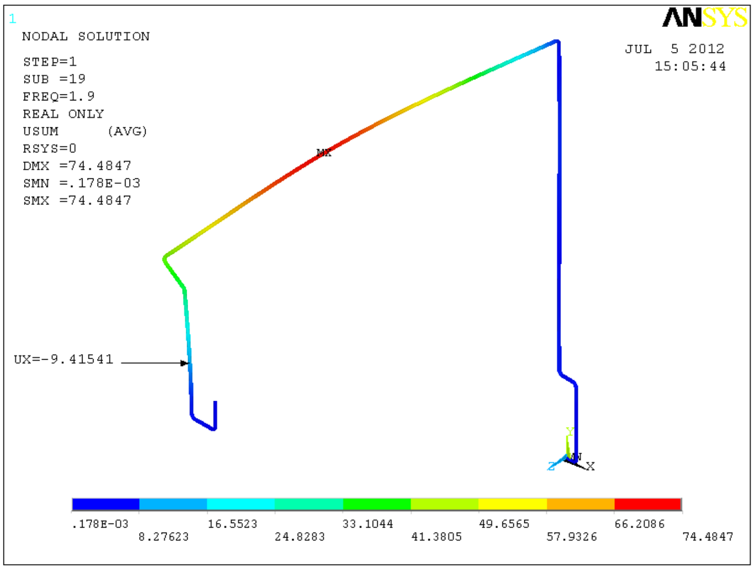

Figure 14: Displacement [mm] in the X direction at the frequency of 1,9 Hz. Notice that the calculated value is in the same order of magnitude of what was measured in the field, indicating that this mode could be active. $ux_{máx}$ = 74 mm.

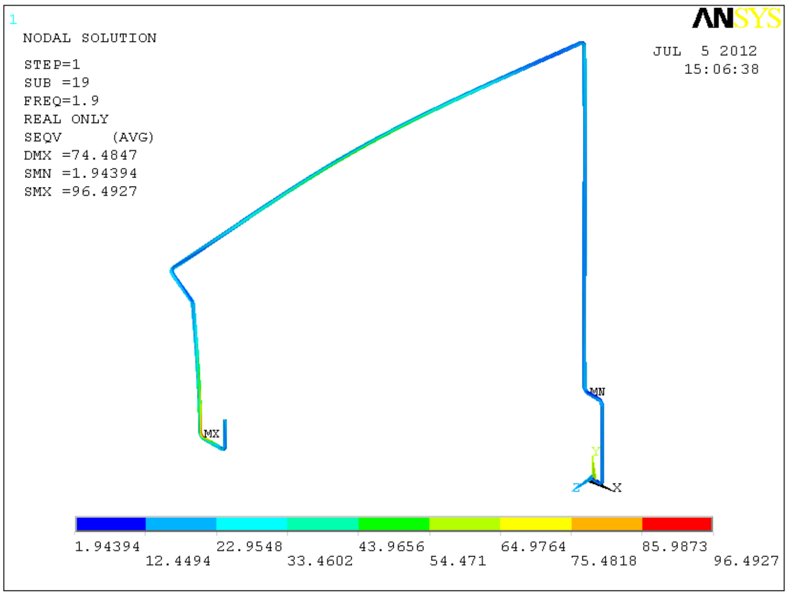

Figure 15: Von Mises stress [MPa] at the frequency of 1,9 Hz. Notice that it is below the allowable design stress (137 MPa). $Seqv_{máx}$=96 MPa.

Although the displacements and stresses are at relatively low levels, activating the 1° vibration mode may cause damage by high cycle fatigue in the pipe system.

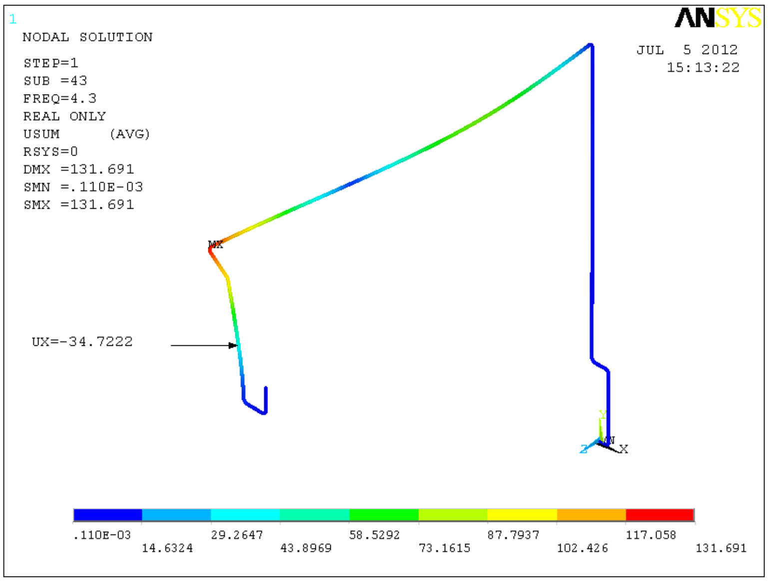

Figure 16: Displacement [mm] in the X direction at the frequency of 4,3 Hz. $ux_{máx}$ = 131 mm.

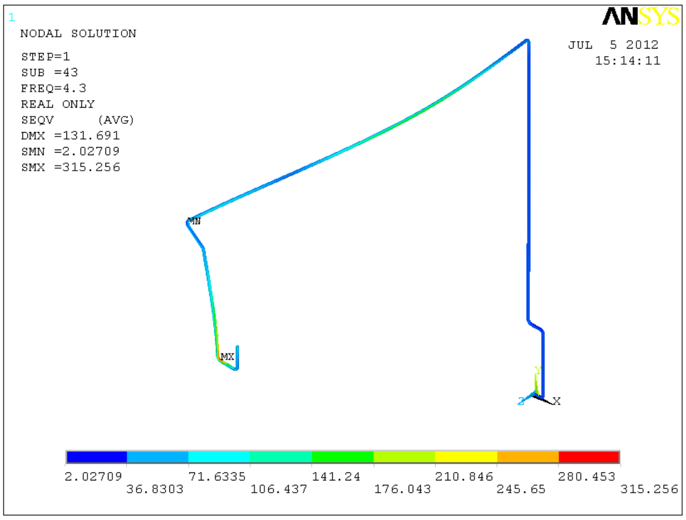

Figure 17: Von Mises stress [MPa] at the frequency of 4,3 Hz. $Seqv_{máx}$=315 MPa.

If the 2nd vibration mode is activated, the stresses are beyond the permissible values and may represent more immediate risk to the pipe system due to the low cycle fatigue damage mechanism.

4.3 Displacement Minimization Design

As one can see, the first resonance frequency was probably occurring during the flow test, since the measured displacement value was very confident to the predicted one at the probe point.

In this case, even the displacement and the stress values being within their allowable range, a simple design of new supports was performed in order to minimize such displacements, by increasing the natural frequency of the system, and enabling further flow test safely. In addition, there is a study of the insurance company DNV ("Det Norske Veritas") entitled DNV-RP-D101 - Structural Analysis of Piping Systems concluding that a good piping design should not have natural frequencies below 5 Hz, which are typical flow frequencies in industrial pipe systems (previous calculations have shown that the harmful resonant frequencies are below this value).

After a series of trial and error, in Figure 18 are shown the positions of new installed supports in the pipe, with the aim to increase its natural frequency by increasing the rigidity of the system. The idea is relatively simple: as the largest displacements were visually perceived in the X-direction, we decided to lock this component.

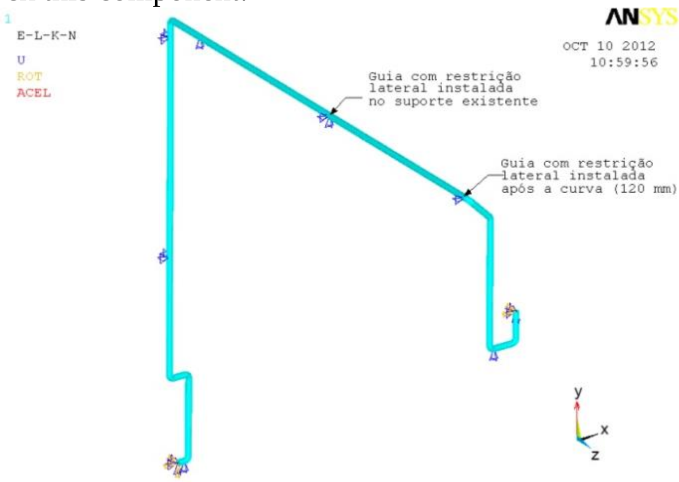

Figure 18: Proposed design, showing the new installed guides.

Thus, a new modal analysis was performed, and the first five results are shown in Table 1:

Table 1 Comparison among natural frequencies of the original system and the modified one.

Mode	**Original System**	**Model with New Supports**
	Frequency [Hz]	Frequency [Hz]
1	1,9441	7,9619
2	4,3307	10,718
3	9,3586	11,631
4	10,857	12,139
5	11,641	13,558

The results from the harmonic analysis, considering the frequency range from 0 to 10 Hz is shown in Figure 19:

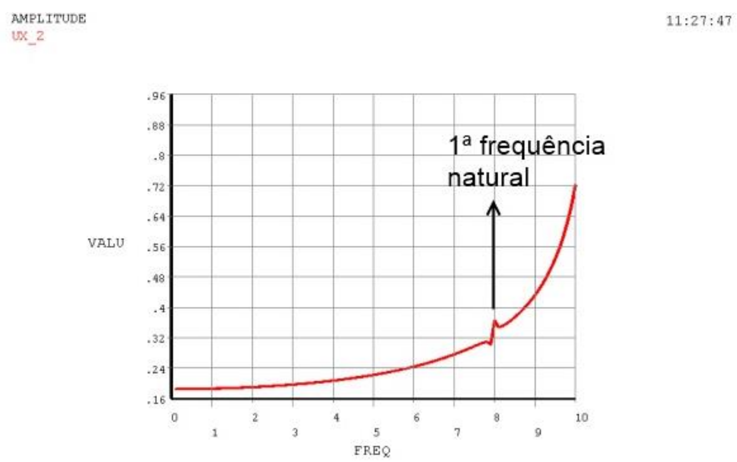

Figure 19: Displacement [mm] X frequency [Hz] plot to the probe point shown in Figure 6.

It is observed that the amplitude of displacement in the X direction, for the 1st mode is negligible, indicating that the use of the guide lead to the expected result. In the figures below are shown the displacements and stresses in the structure in the 1st vibration mode:

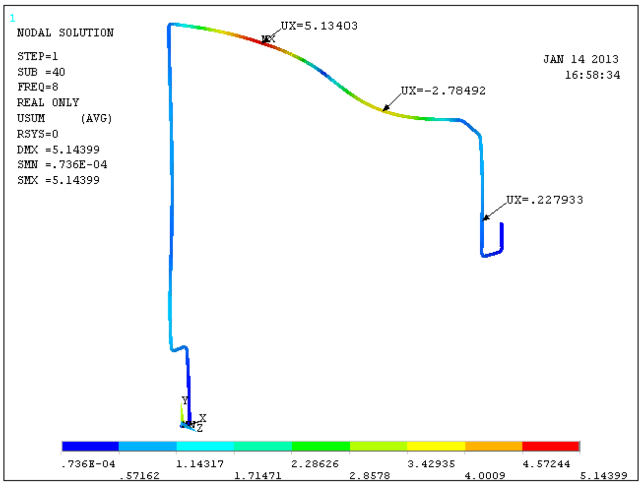

Figure 20: Displacement [mm] in the 1st natural frequency [8 Hz].

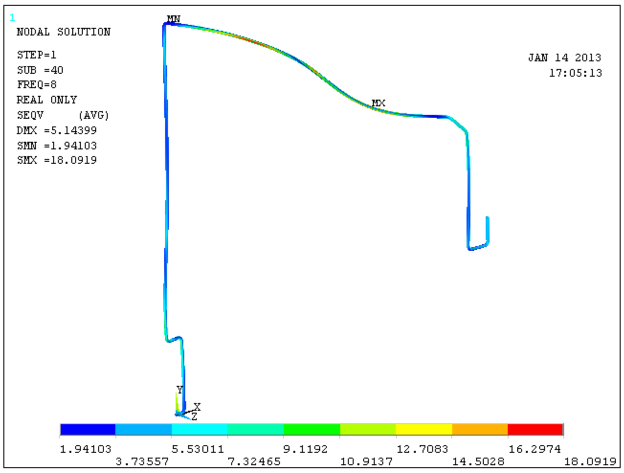

Figure 21: Von Mises stress (maximum = 18 MPa). Remark that the allowable stress and the yield stress are, respectively, 137 MPa and 240 MPa.

4.4 Allowable Displacement Determination

As explained earlier, the flow control is performed indirectly controlling the displacements on the line. An easily accessible point was chosen for measure the displacements. Stresses were calculated, searching what would be the frequency that would lead to the Von Mises value close to the allowable stress design (or to a value lower than the yield stress). Then, the flow that caused this measured displacement would be the maximum permissible flow rate for the system to operate safely. The control point selected for measurement is shown in Figure 22:

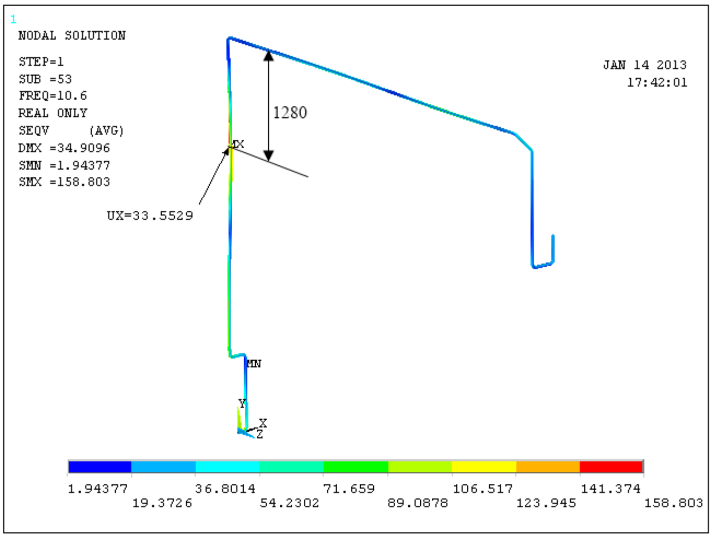

Figure 22: Maximum allowable displacement at the control point. The calculated stress at the most stressed point is approximately 15% above the allowable stress design, but still about 34% below the yield stress.

In Figure 23 is shown the frequency spectrum obtained from the harmonic analysis, proving that the measured point is still in a safe situation.

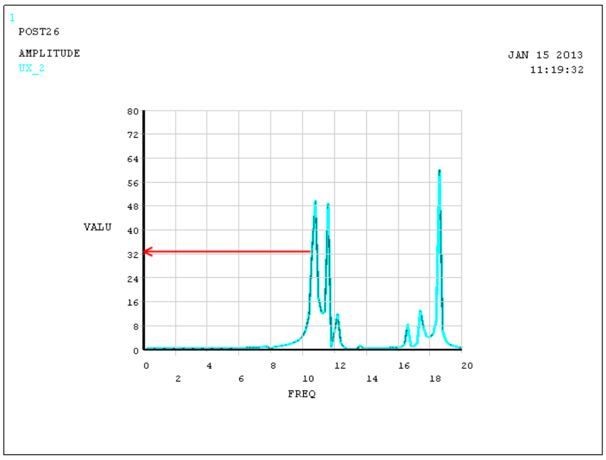

Figure 23: Maximum allowable displacement amplitude at the control point to avoid resonance due to flow.

5 CONCLUSIONS

In this work was carried out the verification of the structural integrity of a pipe system during the execution of a flow test. To this end, an indirect method was devised, using the displacements measured in a control point so that they did not lead the structure to yield stress. This relationship between displacement and stress was carried by a model using the Finite Element Method.

The model has reached to a strong correlation with the initial measured data, indicating that it has been satisfactorily developed. It was also noted that the original design was not in accordance with best practices for design of pipes, according to the specific publication of DNV.

Finally, based on the calculations, it was established that the maximum allowed displacement in structure at the control point, is approximately 33 mm (after installing new supports). Thus, during the flow test, its maximum flow rate value will be the one who cause this displacement value.

6 REFERENCES

[1] A. K. Escoe, Piping and Pipelines: Assessment Guide, Ed. Elsevier, 2006.

[2] N. I. Brasil, M. A. S. Araújo e E. C. M. Souza, Processamento de Petróleo e Gás, Ed. LTC, 2011.

[3] G. Antaki, Fitness for Service and Integrity of Piping, Vessels and Tanks, Ed. McGraw-Hill, 2005.

[4] A. C. Vilaça, A. L. F. Lima e Silva e A. S. Neto, “Modelagem Matemática de Problemas de Interação Fluido-Estrutura,” em *13° POSMEC - Simpósio do Programa de Pós-Graduação em Engenharia Mecânica - UFU*, 2003.

[5] T. J. Hughes, The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Dover Publications, 2000.