Marine Hose Non Linear Analysis Development

R. Guardia, and M.Batista

SMARTTECH Group

Abstract: Some cases of loading were studied through Abaqus/Standard Analysis in order to validate the finite elements model according to the physical tests results made in marine hoses to transport oil considering several situations, such as reeling of the marine hose line in reel during the period of disuse and under hydrostatic pressure conditions. After validating the models, automated process of analysis were developed through the creation of plug-in in the interface Abaqus/CAE and the integration of Abaqus/CAE, Abaqus/Standard and Excel in the software Insight. These automated processes of analysis allow Abaqus to become an important tool in the marine hoses development, and also allow the Development and Research Engineering to analyze virtually several configurations before producing the prototype, reducing the costs of about millions of dollars.

Keyword: Marine Hoses, Plug-in, Isight, Reeling.

1. Introduction

The development of marine hoses to transport oil involves lots of variables, such as the number of layers that must exist for each material to endure the several loading cases, the choice of the material, the foam thickness necessary for the hose marine to keep the necessary flotation during its utilization. It also must be scaled in such a way that it´s possible the reeling of the marine hose in a reel in order to be stored during the period that it´s not used. The condition of the marine hose during the cycle of utilization can be observed in Figure 1.

Figure 1. Marine Hose reeling.

Nowadays, there are tests in laboratory that allow the study of the behavior of marine hoses after the fabrication of the prototype; however, during the phase project phase the structural development is done based on just the practical knowledge and some mathematical calculation estimated. With the customization of the interface Abaqus/CAE through the generation of Plug-in, several configurations can me analysed quickly, allowing a development more precise and less expensive.

To develop this work were considered the reeling analysis of the marine hose in reels, analysis to the evaluation of the behavior of the marine hose under a working pressure, to determine through the analysis the maximum pressure supported by the structure of the marine hose, to evaluate the behavior of the marine hose considering two layers of reeling in reel, determination of the axial stiffness considering the marine hose with and without internal pressure, determination of the axial stiffness and bending stiffness.

The Plug-in created generates the CAD model, the finite elements model with its equivalent properties runs the analysis according to the input data of the user. Until this moment, the Plug-in created runs the hydrostatic pressure analysis, while the others must be setted manually from the element finite model created automatically.

Some simplifications and equivalences were considered in the model in order to reduce the computational time and allow a better performance of the tool in the development phase of the project.

From the flux of information created through the Insight, it´s possible to study the reverse path and from the maximum permissible stress of the materials to determine the quantity of the material required, and the best disposition to the optimized structure of the marine hose.

2. Analysis performed

To validate the element finite model, Flexomarine provided results of the physical tests that were done. From this information, the model was calibrated to obtain the results correlated. The element finite model was generated according to the CAD model about the structure of the marine hose.

- Reeling analysis

  To represent the reeling of the marine hose in the reel to storage, it was considered the marine hose as conformed in the reel and then the stresses and strains for this configuration were analysed.

  The condition for reeling can be observed in Figure 2.

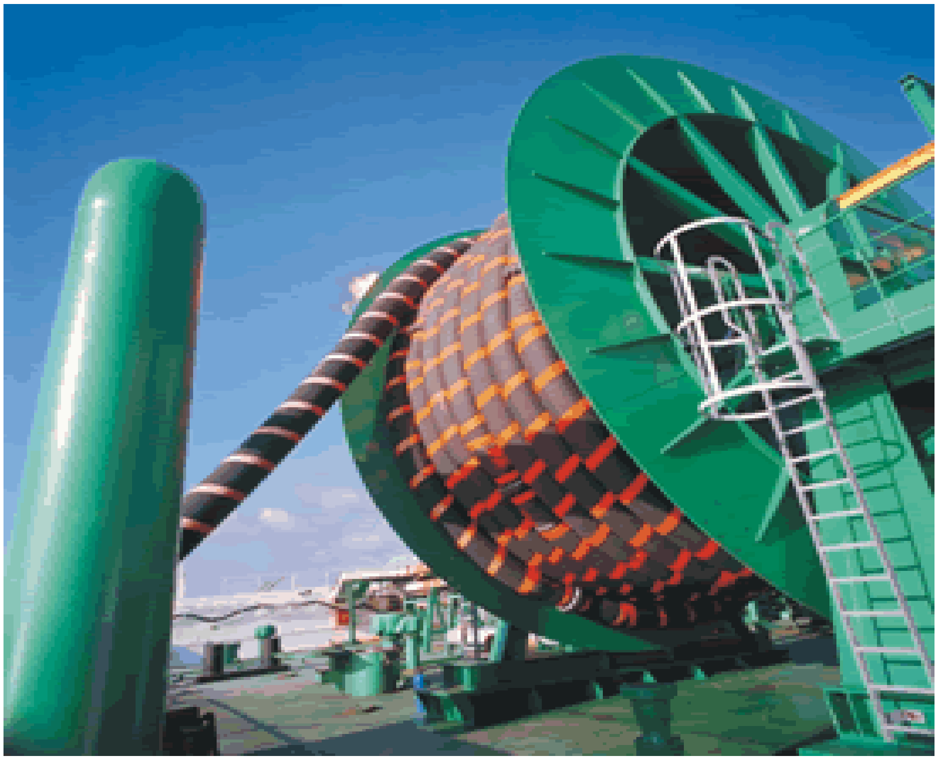

Figure2. Marine hose under the condition of reeling in the reel.

The results for stresses and displacements were obtained to the most critical position of the marine hose, and were analyzed in each step of the reeling for each layer of the material.

Figure 3 shows the distribution of the displacements in the structure during the conformation of the marine hose.

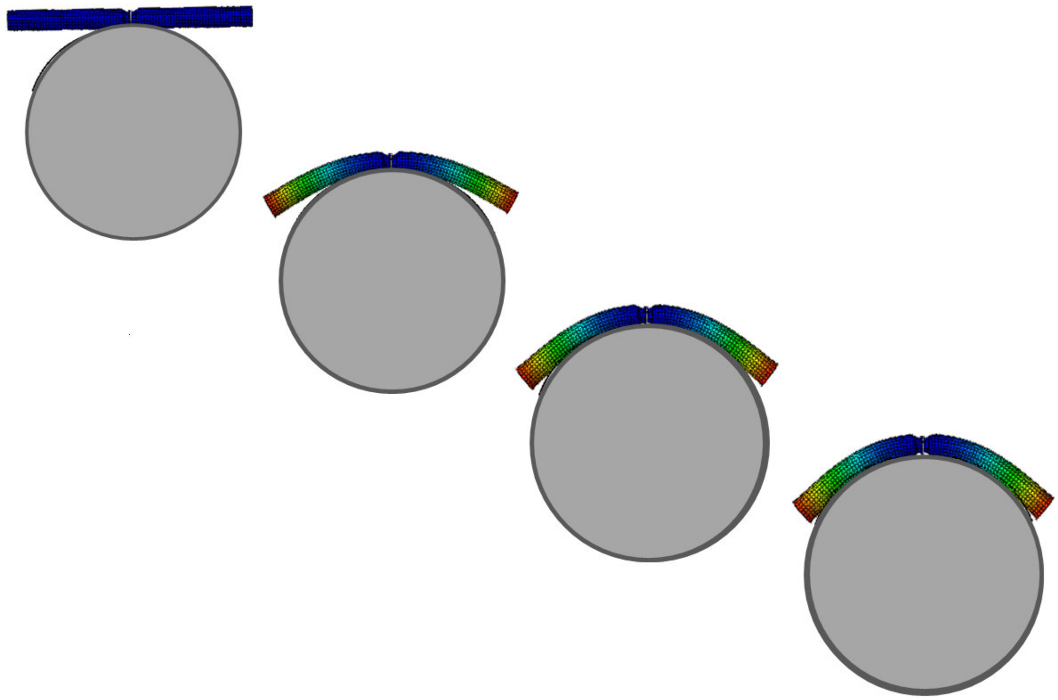

Figure 3. Evolution of the displacement during the conformation of the marine hose.

In Figure 4 it can be observed the evolution of the Von Mises stresses during the conformation of the marine hose. Theses stresses were also analyzed for each layer of the structure of the marine hose.

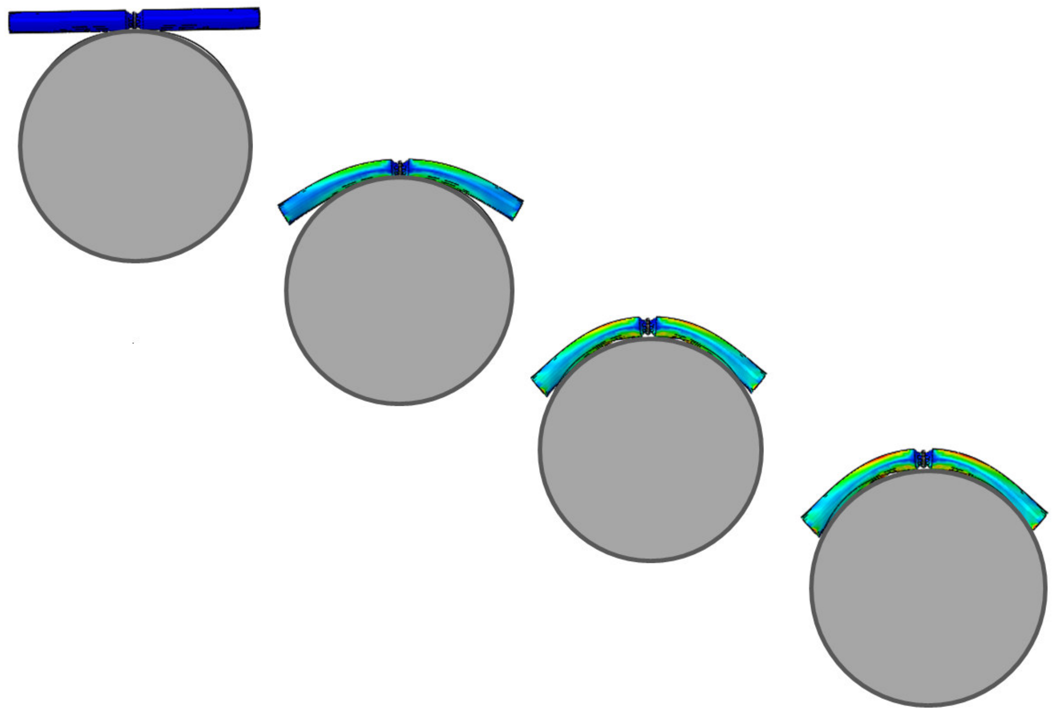

Figure 4 – Evolution of Von Mises Stress during the conformation of the marine hose.

For this analysis, the stresses at marine hose should be less than the material rupture limit. The typical value is about a 30% less than the material limit, for example the von mises stress cannot be higher than 5.4 MPa at the rubber that has 18 MPa as a rupture limit.

- Analysis considering the hydrostatic pressure

To the hydrostatic pressure condition of the marine hose, the structure was simulated considering a working internal pressure determined.

The model used to simulate the hydrostatic internal pressure can be observed in Figure 5.

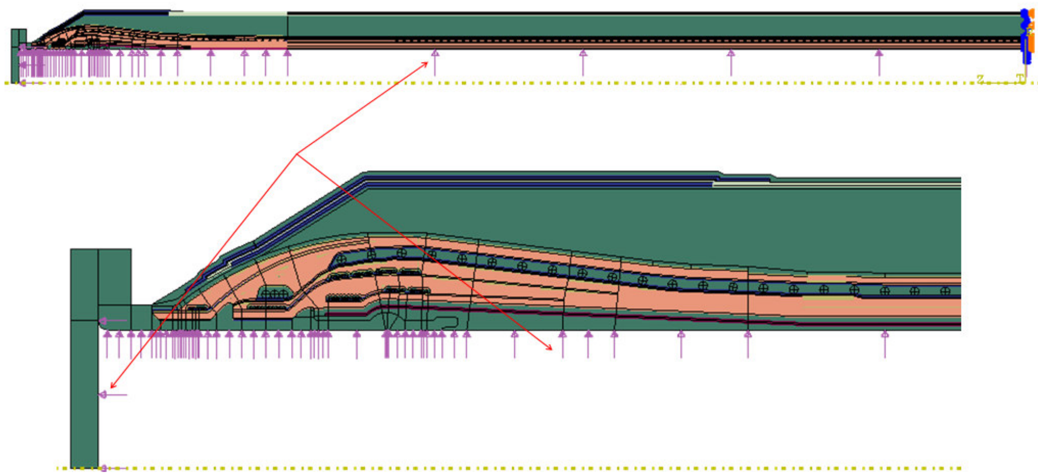

Figure 5. Condition of Internal Hydrostatic Pressure.

For this analysis, the stresses at marine hose should be less than the material rupture limit.

- Axial stiffness analysis without internal pressure

The marine hose was considered without internal pressure, and a tractive force was applied in its end in order to determine the axial stiffness of the structure.

The axial stiffness was calculated according the following formula:

$$EA = \frac{F}{\overline{X}}$$

Where:

EA = Axial Stiffness

F = Force

$\overline{X}$ = Hose displacements

(Source: OCIMF 2009)

For this analysis, the stresses at marine hose should be less than the material rupture limit. For example the von mises stress cannot be higher than 24 MPa at the rubber that has 24 MPa as a rupture limit.

- Axial stiffness analysis with internal pressure

An internal pressure equals to 1.5 times the working pressure allowed was applied in the marine hose. It was also considered a tractive force known in the end of the marine hose in order to determine the axial stiffness of the structure.

The axial stiffness was calculated according the following formula:

$$EA = \frac{F}{X}$$

Where:

EA = Axial Stiffness

F = Force

X = Hose displacements

(Source: OCIMF 2009)

For this analysis, the stresses at marine hose should be less than the material rupture limit.

- Torsional stiffness analysis

To calculate the torsional stiffness is applied a Momentum determined, considering the hose without internal pressure and with one of its end fixed, as it can be seen in Figure 6.

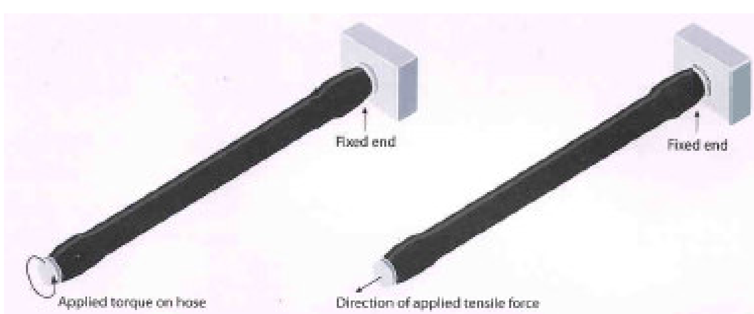

Figure 6. Condition for the application of loadings to anylize the torsional stiffness.

Thus, the torsional stiffness is calculated by the equation:

$$GI = \frac{T}{\alpha}$$

Where:

GI = Torsional Stiffness

T = Torque

α = Rotation in radians per length unit

(Source: OCIMF 2009)

For this analysis, the stresses at marine hose should be less than the material rupture limit.

- Bending Stiffness Analysis

For bending stiffness analysis, was considered the configuration shown at Figure 7.

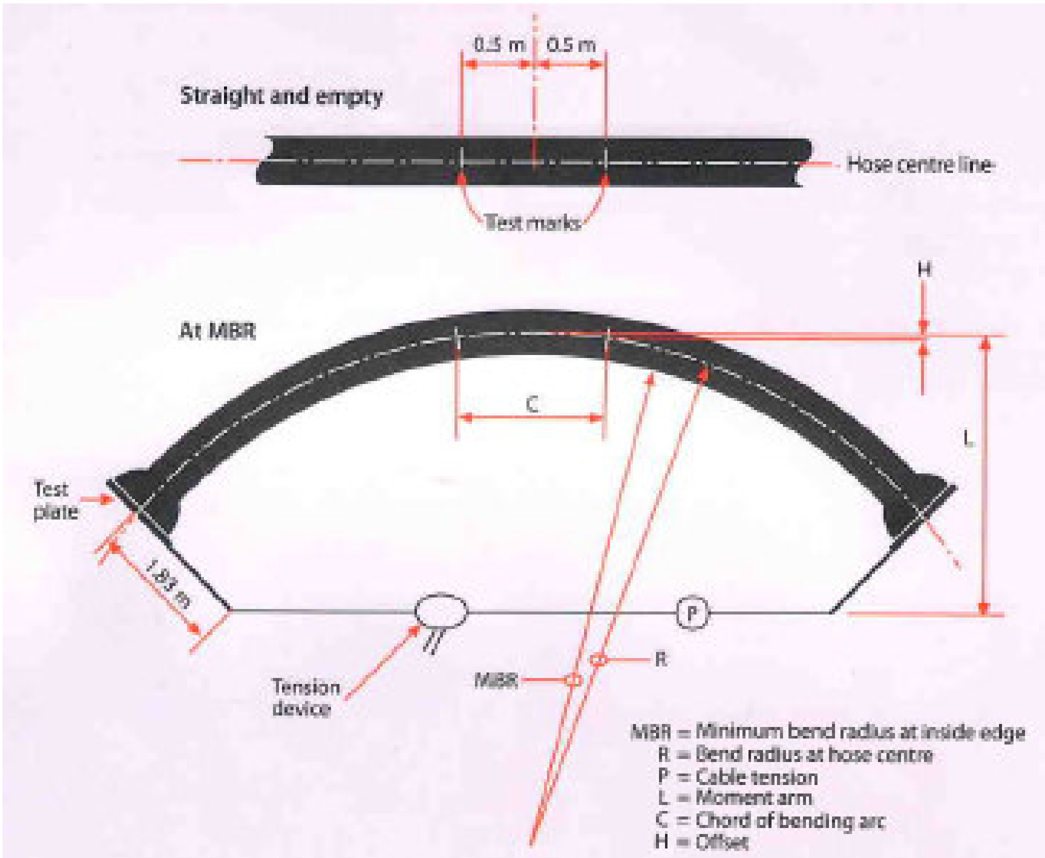

Figure 7. Calculations conditions for bending stiffness.

With the previous results, the stiffness value is calculated using the following equation:

$$EI = M \cdot R \text{, where:}$$

$$M = P \cdot L$$

$$R = \frac{\left(C^2 + 4H^2\right)}{8H}$$

where,

EI = Stiffness

M = Momentum at hose centre

R = Radius at hose centre

P = Cable tension

L = Momentum arm

C = Chord of bending arc

H = Offset

(Source: OCIMF 2009)

For this analysis, the stresses at marine hose should be less than the material rupture limit.

- Two layers reeling analysis

In order to study the behavior of the first layer of the marine hose in the condition of reeling in the reel, under the second layer, a finite element model was created as shown in Figure 7.

Figure 7. Analysis considering two layers of reeling of the marine hose in the reel.

The results obtained allow to evaluate the behavior and influence of each layer of the material, besides other parameters such as the permanent displacement of the floating foam.

An example of a deformation map can be observed in Figure 8.

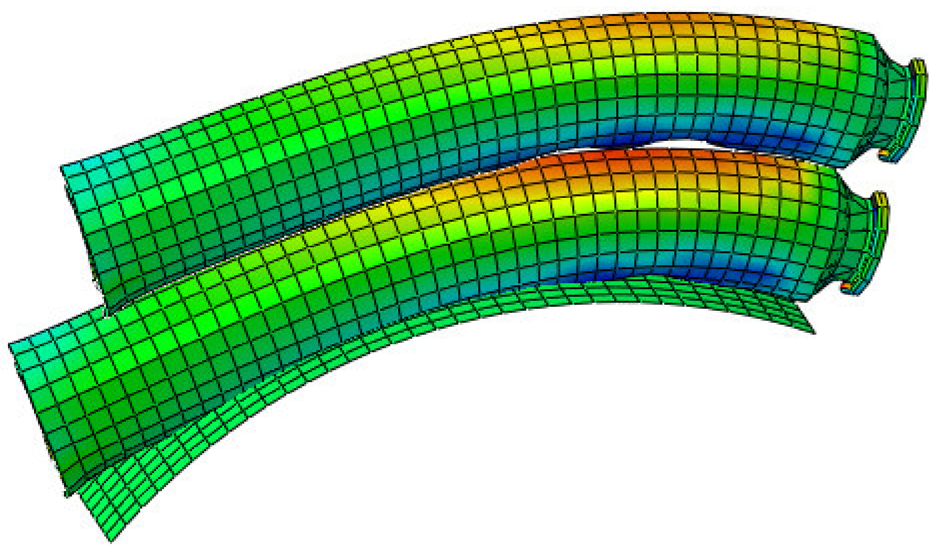

Figure 8. Displacement Plot

For this analysis, the stresses at marine hose should be less than the material rupture limit.

3. Development of the Plug-in

Based on the first results obtained after the calibration of the models with the physical tests results, a Plug-in was developed as shown in Figure 9, where the user just needs to type the constitutive data of the marine hose structure to have the element finite model created automatically so the user can choose which analysis will be run.

The Plug-in was developed based on the customer demand to run a large number of different models to evaluate the hoses marine project, and was located at an engineering consulting office that runs the Plug-in quickly with the data provided by the customer.

With the Plug-in development, the customer can get easily the stress distribution at the marine hose, as well the stiffness and other parameters necessary to evaluate the product.

To create the Plug-in based on the first model analyzed, was required 40 hours approximately.

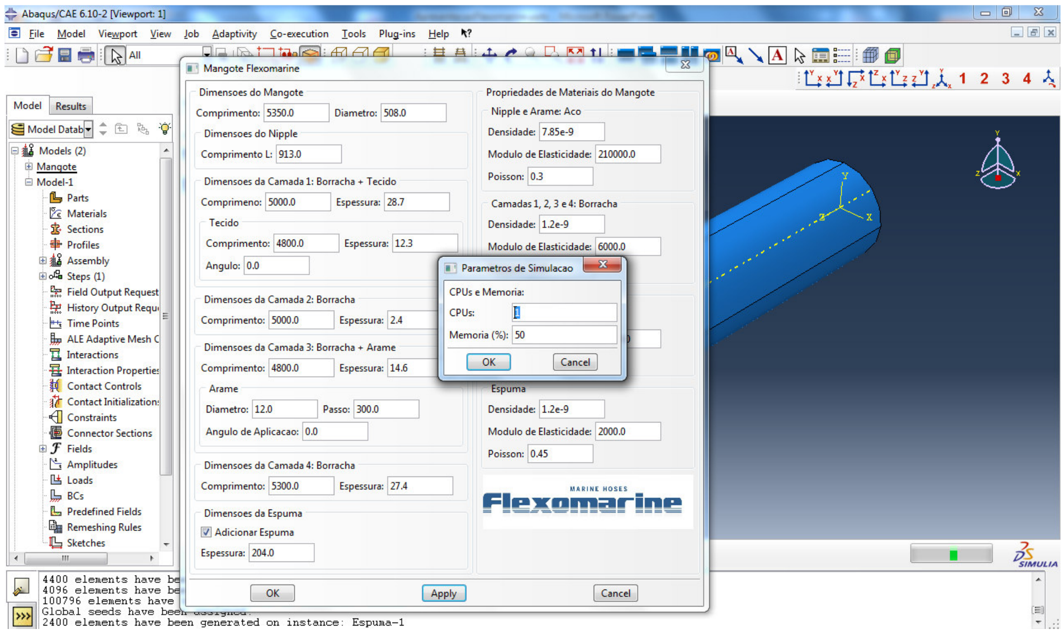

Figure 9. Plug-in Interface

4. Isight Development

The integration between Abaqus and Microsoft Excel allowed the generation of a working flux where the user has the option to execute the same activity done through the Plug-in, or the reverse path which from the answer desired the input data can be obtained to the construction of the structure of the marine hose.

The user interface is shown in Figure 10.

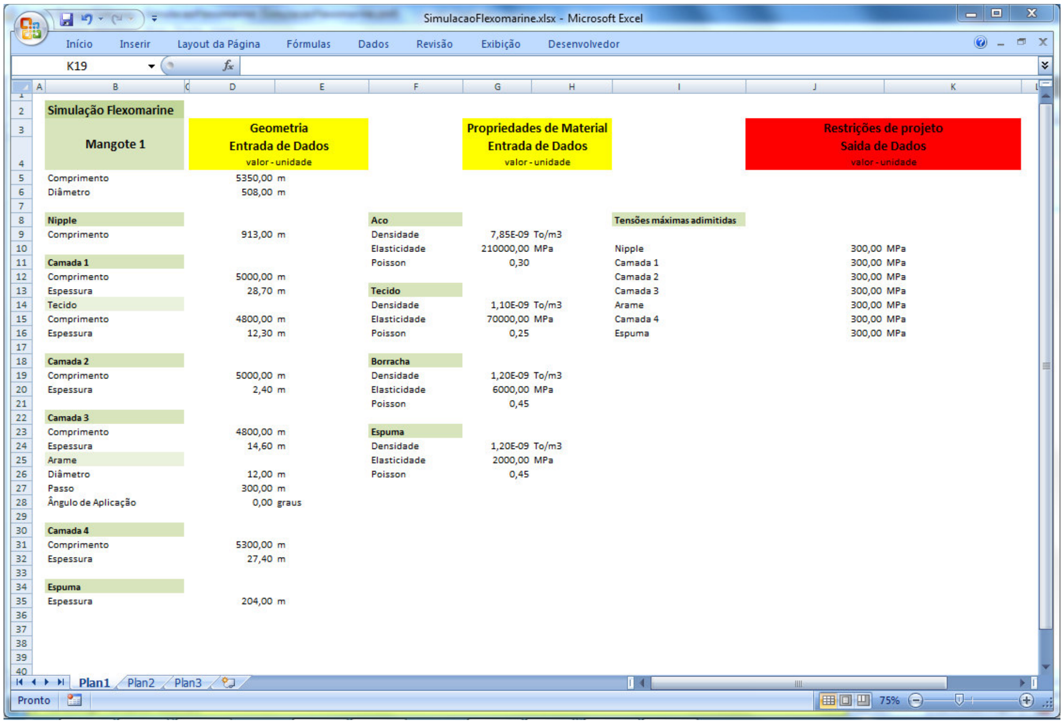

Figure 10. User interface

Figure 11 shows the Isight workflow used to perform the analysis.

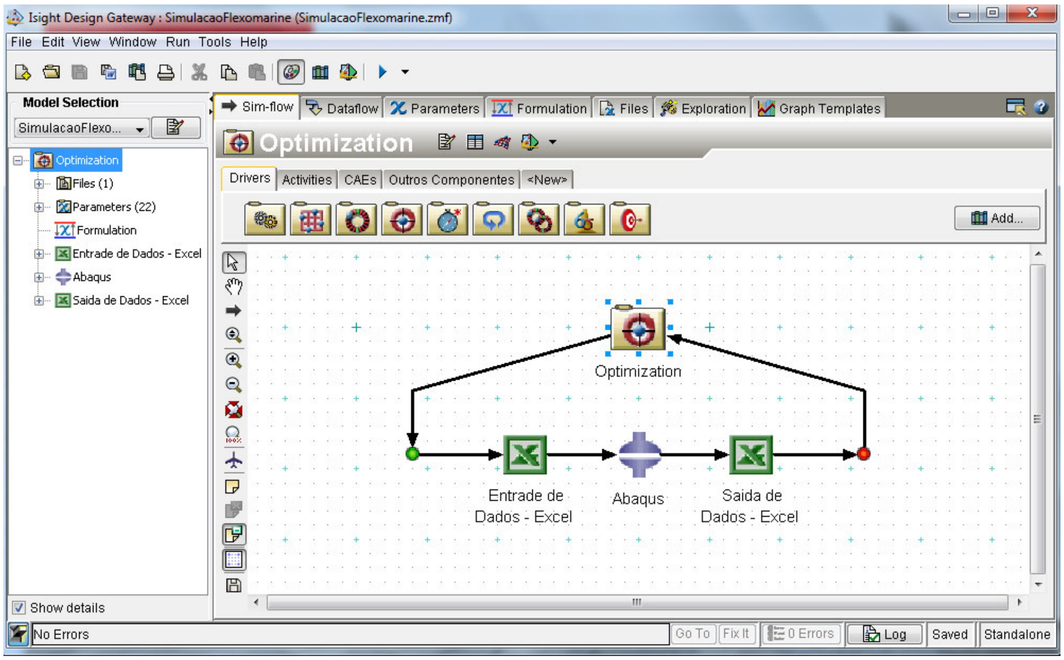

Figure 11. Isight workflow.

With the Isight workflow, the user can get the results the user can get results even if he did not have knowledge of finite elements or software Abaqus.

Just entering the input data via Excel sheet, and after the analysis it is automatically populated with the results.

This makes the process of product development easier and faster.

5. Acknowledgement

The authors gratefully acknowledge Mr. Mauricio Spaolonzi, Mr. Mauricio Bolonhezi, Mr. Rodrigo Soprani and Mr. Antonio Araes.

6. Conclusion

The work showed that the results of simulations in Abaqus were correlated with the results of physical tests performed.

From the correlation of the model with the physical tests, it became possible to create the automation of the calculation process.

With the automation, the future analysis can be performed quickly and easily through an Isight workflow using excel and Abaqus or through a Plug-in generated in Abaqus. The user is not required to have knowledge in finite element method or software ABAQUS to get the results.

This allows the reduction of physical tests, reducing the company costs related to construction of prototypes.

It also provide greater efficiency in product development, since it is possible to know the behavior of the marine hose structure before its manufacture and thus taken the necessary steps until the optimum condition is reached.