MODELING THE MECHANICAL BEHAVIOUR OF A SUBSEA GUIDE BASE SUBJECT TO INSTALLATION AND OPERATION LOADS

José Felipe Bra, jose.braz@c-a-m.com
Luiz Eduardo Nicolini do Patrocínio Nunes, luiz@unitau.br
Valesca Alves Correa, valesca@unitau.br
University of Taubaté, 12060-440, Taubaté, SP, Brazil

Abstract. *Historically, the guide cable system for installation of subsea oil production equipment has proven its efficiency. These cables extend from the rig to the wellhead, which are fixed to the guide base posts and will be subject to loads during the installation and operation of the production equipment. The majority of guide base designs have been calculated by analytical methods, although they allow obtaining exact answers in all infinite points of a structure, they are not applicable to all cases. This motivated the application of finite element analysis method (FEA) in this work, which from the development of approximate procedures can be applied regardless of the structure shape and loading conditions. This paper consists of a structural analysis of the guide base frame model, generated in Autodesk Inventor® software and analyzed in ANSYS® Workbench. The study brings a diagnostic research using a quantitative and exploratory approach. Different mathematical models were generated to obtain the results at the most critical points of the structure in order to determine its resistance to the loads applied during installation and operation, taking into account the criteria laid down in DNV 2.7-3, API 17D 1st Ed and von-Mises equivalent stress.*

Keywords*: Structural analysis, Finite element analysis method, ANSYS®, Subsea guide base.*

1. INTRODUCTION

In the subsea oil exploration many kinds of equipment and operational techniques are used and determined depending on the operating conditions. In the specific case of oil production equipment, the determining factor in this choice is the water depth. At depths around 1500 ft (457 m) guide cable system is primarily used to guide the descent of the equipment from the probe (boat) to the field. These cables extend from the probe to the oil well, which are fixed to the guide base posts that will be subject to the efforts of installation and operation during the running of the production equipment (API, 1992).

The main purpose of structural design in engineering is to ensure that designed equipment is not subject to failures caused by the variety of operational conditions. Although the classical analytical methods for calculating obtain exact answers in all the infinite points of a structure, it is not always possible to implement them due to the fact of these solutions are known only for some cases. Faced with this enormous constraint was developed the method of finite elements analysis, which from the development of approximate procedures can be applied regardless of the shape of the structure and loading conditions (Filho, 2008).

In general, the finite element analysis can be defined as a mathematical method in which a continuous structure is subdivided into a finite number of elements, connected by discrete points (nodes). These elements are described by algebric equations that form the mathematical models solved by means of matrix to obtain the desired results, judging from the number of discrete points chosen is sufficient to represent the full strength roughly (Filho, 2008 e Lotti et al., 2006).

Facing this fact, the goal of this study is to analyze the substructure of the subsea guide base and determine its resistance to efforts made by the method of finite elements analysis. Computational tools are used in engineering, with graphics that help to understand the results, such as Autodesk Inventor ® and ANSYS ® (Kanarachos, 2006).

1.1. von-Mises Equivalent Stress

According to the standard (DNV, 2006), only the primary structures must be included in design calculations. The resistance of structural members shall be calculated by means of manual calculations and three-dimensional analysis of beams or the finite element analysis. This standard also establishes that the von-Mises equivalent stress produced must not exceed 85% of the yield stress of the material used in the structure.

1.2. Finite Element Analysis

To obtain an experimental model for the finite element analysis, it is necessary first to model the object of study in a computer program for 3D modeling, such as the Autodesk Inventor ®. Subsequently, the model must be discretized into a finite number of elements, using computer programs for finite element analysis, such as ANSYS ® (Lotti et al., 2006). The elements most commonly used for representing solid are the tetrahedral and hexahedral. Then the hexahedral are applied to plane geometry and solid, like the beams of the structure of the guide base guide. And the tetrahedral solids are applied to geometries with discontinuities (Filho, 2008 e Documentation for Ansys, 2007).

The choice of the element size in finite element model is a very important issue and is directly linked to the characteristics of the chosen element to the mesh. A widely used technique to obtain a mesh that best represents the behavior of a structure is the test of convergence, where the mesh is made in a progressive way, by applying successive refinements of the mesh points that have significant changes in tension, *"so the fact that the tension is constant within the elements, be overcome with small discontinuities of tensions between the adjacent small elements"*(Filho, 2008). Although in some cases the results may converge after several iterations the mesh refinement usually generates higher stress values depending on model's geometry.

2. METHODS

In this study, the mechanical behavior of the bottom frame of the guide base subject to external loading was considered (Fig. 1), because it is a primary structural element according to the standard (DNV, 2006). Non-linear numerical models were developed to analyze the stress state of the structure during loading (Kanarachos, 2006). A parametric analysis considering a vertical and a horizontal load was conducted using the convergence test for mesh refinement (Filho, 2008, Kanarachos, 2006 e Defilippo, 2007).

In addition, all analyses were performed on programs already established in the industry. Both the modeling of each component and the final guide base assembly model was generated in Autodesk Inventor ®. This model was then introduced and processed by finite element analysis software ANSYS ® to obtain the results of von-Mises equivalent stress, which were compared with the permissible values per standard (DNV, 2006), considering the mechanical properties of the material used.

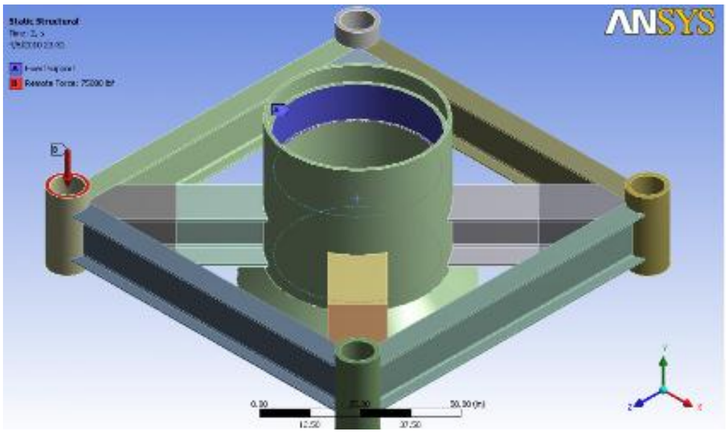

Figure 1. Guide base lower structure model and Boundary conditions

2.1. Vertical Loading

The structure has square dimensions with width L = 113.07" on the x axis, C = 113.07" length in the direction of z axis and height H = 50.75" at the direction of the y axis.

The mesh was generated in ANSYS ®, initially with a relevance parameter $R = 0$ on all elements of the structure. The mesh is shown in Figure 2 and has a total of 10.170 nodes and 20.053 elements with 16.386 solid elements and 3.666 contact elements. Tetrahedral elements were used for solid bodies and hexahedral elements to the beams, except to the diagonal beams welded to the pipes, which due to its geometry, hexahedrical mesh was not accepted. The boundary conditions are shown in Figure 1, all contacts between the components of the structure were defined as bonded as the design specifies full penetration weld between components. The structure was subjected to an external force in the vertical direction (point *B*) *Fv* = 75000 lbf and set as fixed into the inner diameter of the centre section (point *A*).

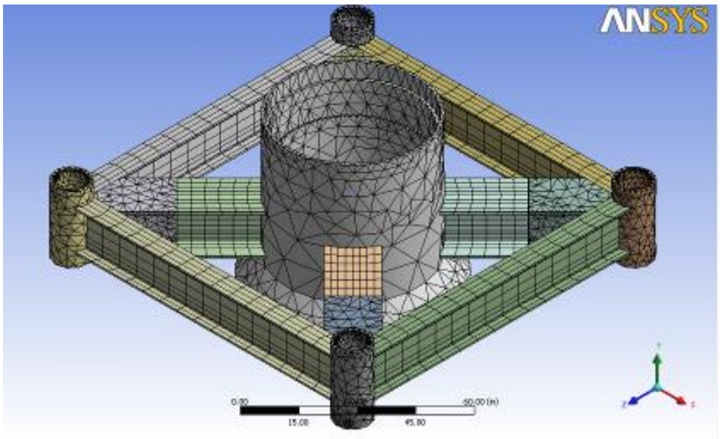

Figure 2. Mesh with Relevance R=0

The bi-linear isotropic hardening (figure 3) of the material was considered to represent the stress-strain diagram of the structural steel used in accordance with its values of yield strength and tangential modulus, specified in Table 1 which also shows the mechanical properties steel.

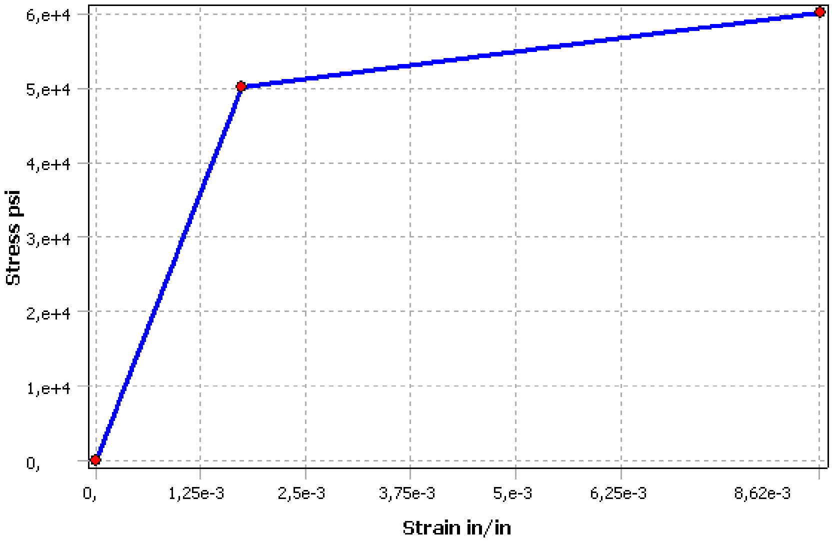

Figure 3. Bi-linear isotropic hardening chart

Table 1. Mechanical properties of the material.

Structural Steel	
Young Modulus	29.008.000psi
Poisson Ratio	0,3
Density	0.28383lb/in³
Yield Strength	50.000 psi
Tensile Strength	60.000 psi
Tangential Modulus	1.450.400 psi

The stress distribution occurred as shown on Figure 4. It can be seen that the maximum von-Mises equivalent stress was 30.422 psi, and occurred at the lower end of the central beam attached to the central cylindrical body. Then a mesh refinement was applied, increasing the relevance parameter for R = 100 on these two solids as shown in Figure 5.

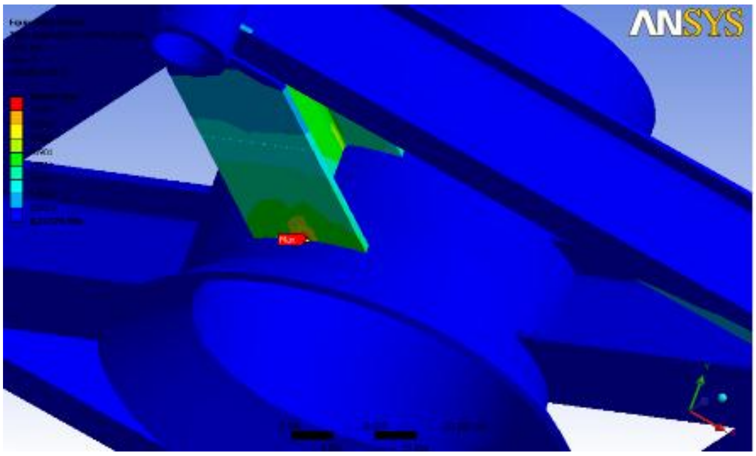

Figure 4. Stress distribution

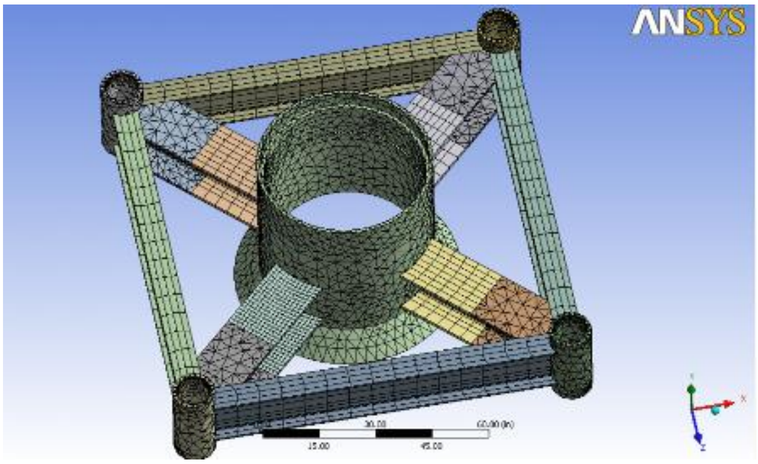

Figure 5. Mesh Refinement

In this case, the maximum value for von-Mises equivalent stress obtained was 35.604 psi still at the bottom of the beam, but the stress concentration occurred at the corners, as shown in Figure 6. Facing this, we applied another refinement in these corners and in cylindrical face, according to Figure 7.

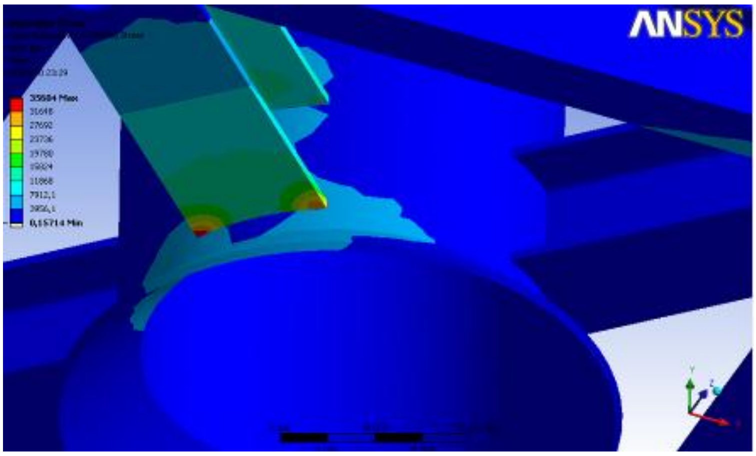

Figure 6. Stress concentration

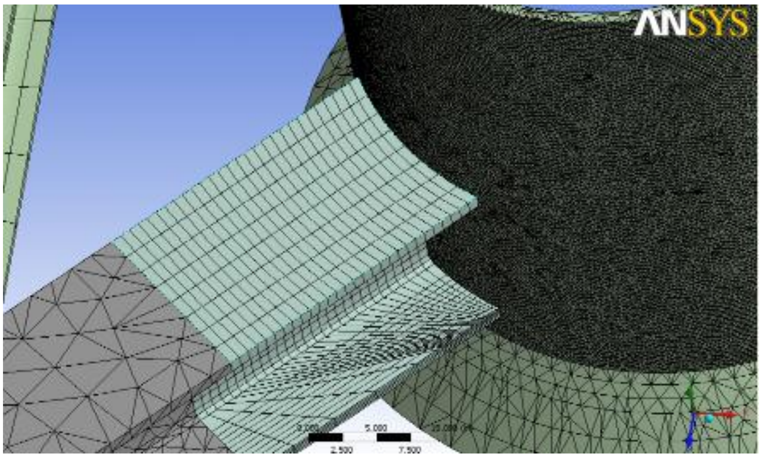

Figure 7. Mesh refinement

After processing this model the maximum von-Mises equivalent stress was 41.753 psi and maintained stress distribution at the corners as illustrated in Figure 8.

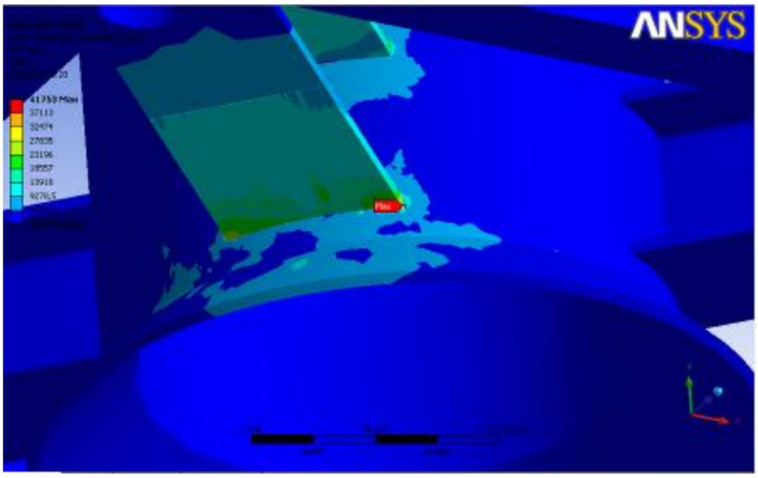

Figure 8. Stress distribution

2.2. Horizontal Loading

The same model was used to analyze the structure. Now under the action of a load in the horizontal direction of magnitude *Fh*=100.000 lbf. However, as it has been identified in the previous analysis the elements of structure that would be stressed, we used the second model (Figure 6) as a starting point. The boundary conditions are indicated in Figure 9.

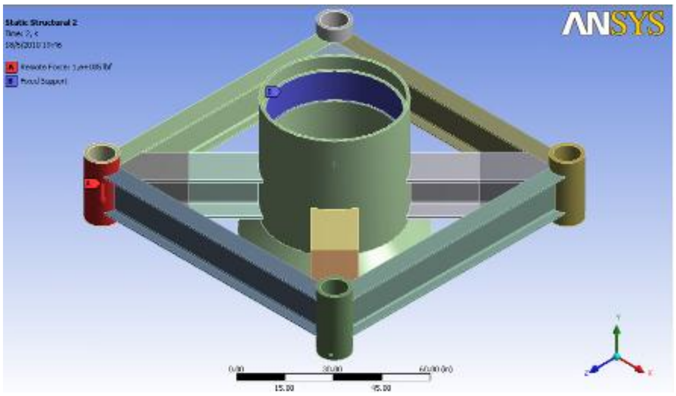

Figure 9. Boundary conditions

The stress concentration due to horizontal load occurred in the upper corners of the beam and reached a maximum value of von-Mises equivalent stress of 25.003 psi according to Figure 10. As in the analysis of the vertical load was applied a refinement to these points of contact and face, in a manner analogous to Figure 7 and, as a new result, we obtained a maximum value of von-Mises equivalent stress of 36.157 psi concentrated at same location (Figure 11).

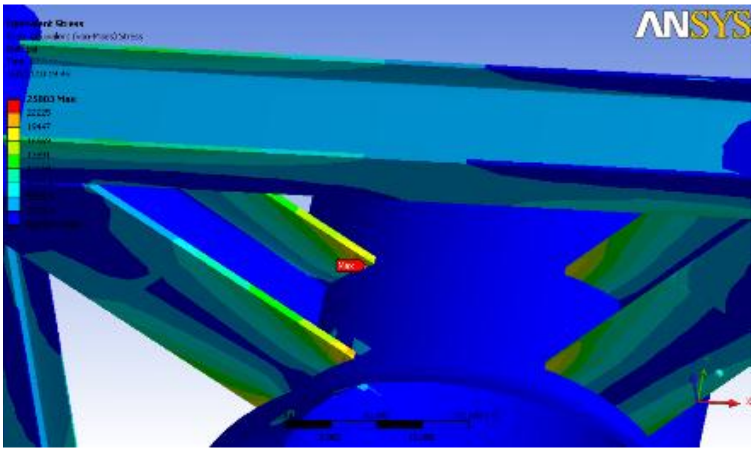

Figure 10. Stress concentration

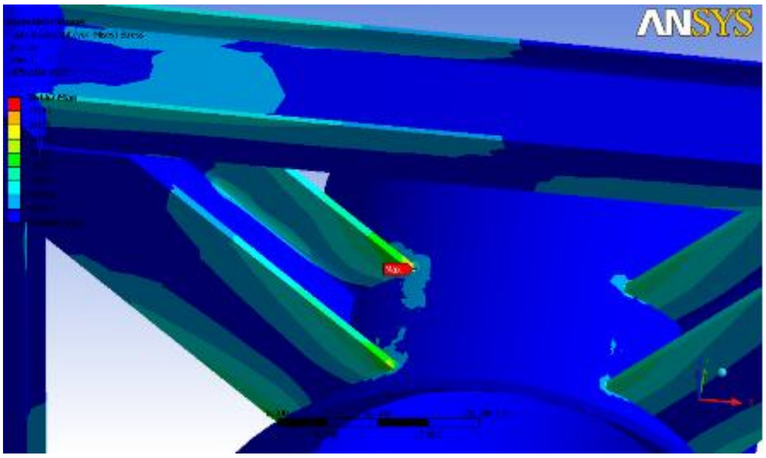

Figure 11. Stress concentration

3. RESULTS

For this analysis, we considered the lower structure of the subsea guide base that was subjected to loads according to the boundary conditions described in items 3.1 and 3.2. As results of this analysis were collected the maximum von-Mises equivalent stress for each one of meshed models according to figures 4, 6, 8, 10 and 11, values which can be observed in Tables 2 and 3.

Table 2. Comparison: Maximum von-Mises Equivalent Stress x Mesh relevance to Vertical Force

Mesh type	Maximum von-Mises Equivalent Stress (psi)
Relevance parameter *R* = 0 (at whole model)	30.422
+ Relevance parameter *R* = 100 (at the most stressed bodies)	35.604
+ Mesh refinement of elements (at the most stressed points)	41.753

Table 3. Comparison: Maximum von-Mises Equivalent Stress x Mesh relevance to Horizontal Force

Mesh type	Maximum von-Mises Equivalent stress (psi)
Relevance parameter *R* = 0 (at whole model)	N/A
+ Relevance parameter *R* = 100 (at the most stressed bodies)	25.003
+ Mesh refinement of elements (at the most stressed points)	36.157

The results of von-Mises equivalent stress obtained in the analysis of models with mesh refinement at the critical points were compared with the maximum allowable stress specified in the standard (Beer e Johnston, 2007) and presented in Table 4 for each of the loading conditions.

Table 4. Comparison: Maximum von-Mises Equivalent Stress x Maximum Allowable Stress

Loading type	Maximum von-Mises Equivalent Stress (psi)	Maximum Allowable Stress (psi)
Vertical load *Fv*=75.000lbf	41.753	42.500
Horizontal load *Fh*=100.000lbf	36.157	42.500

According to the standard (DNV, 2006), the maximum von-Mises equivalent stress should not exceed 85% of the material yield strength. Finally the beam pictures were collected during the most stressed occasion in refined models at critical points to facilitate visualization of the distribution of efforts in the structure, which can be seen in Figures 12 and 13.

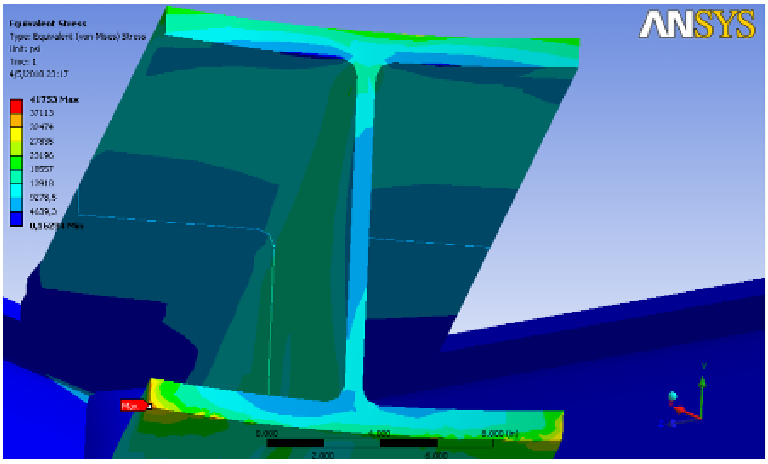

Figure 12. Stress distribution due to *Fv*

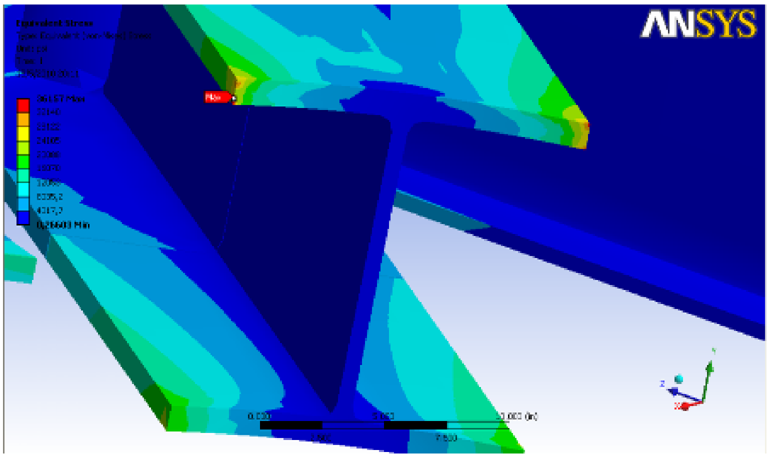

Figure 13. Stress distribution due to *Fh* [7]

4. CONCLUSIONS

It can be concluded, in analyzing the results presented in Tables 2 and 3, when mesh refinement is applied to the model, i.e. decrease of the elements size at the points of interest that the answers are located and higher values for the von-Mises equivalent stress are obtained. This increase of stress occurs due to the sharp edges on the model and might be a concern then is suggested for future works to take weld geometry into account (even though not considering residual stresses) as it might distribute better the stress through out the structure reducing the values.

However based on data from Table 4, it is evident that the lower structure of the guide base resists to vertical and horizontal loads under the boundary conditions described in 3.1 and 3.2.

The finite element analysis enabled us to identify not only the most stressed elements of the structure but also to demonstrate that the structure resisted efforts applied because as quoted in the standard (DNV, 2006), the highest values of von-Mises equivalent stress did not exceed 85% the material yield strength of the structure of the base guide.

Observing figures 12 and 13, note that the stress distribution occurs in an acceptable manner, since a stress concentration was expected in these locations due to the geometry of the beam. Showing that the analysis faithfully represents the behavior of the lower structure of the guide base when subjected to efforts for installation and operation.

5. REFERENCES

API 17D 1st Ed. - Specification for Subsea Wellhead and Christmas Tree Equipment, 1992.

Filho, A. A. Elementos Finitos: A Base da Tecnologia CAE, 5ª Ed., 2008.

Lotti, R. S. et al – Aplicabilidade Científica do Método de Elementos Finitos, 2006.

Kanarachos, S.; Demosthenous, G. Modeling the mechanical behavior of composite metal plastic pipes subject to internal pressure and external soil and traffic loads, 2006.

DNV No. 2.7-3 - Standard for Cerification - Portable Offshore Units, 2006.

Beer, F.P.; Johnston JR, E. R. Resistência dos Materiais, 3ª Ed., 2007.

Documentation for ANSYS® Workbench – Release 11.0, 2007.

Defilippo, F .B. Geração automática de modelos de elementos finitos para análise de estruturas reticuladas na exploração de petróleo offshore, 2007.