

# Numerical Structural Analysis of a Single Girder Crane According to Standard NBR 8400

Samuel Faria Arantes<sup>1</sup>, Ayrton Hugo de Andrade e Santos<sup>2</sup>, José Rubens Gonçalves Carneiro<sup>3</sup>

<sup>1,3</sup>Department of Mechanical Engineering, PUC Minas, Brazil

<sup>2</sup>Department of Civil Engineering, PUC Minas, Brazil

**Abstract**—The use of cargo handling equipment in the industry in general is extremely important for logistics, since their contribution in receipt of material, until the production stages. Since they are subjected to severe mechanical stress, it is necessary for them to resist to these loading. This work aims to verify the structural integrity of a single girder crane designed by “ARPI Engenharia”, according to standard NBR 8400. In order to evaluate the crane structure and compare the mechanical stresses with NBR 8400 yield stress, a 3D model was created in the software Ansys Workbench 17.2 and analyzed through the Finite Element Method. This is one of the most widely used methods by Engineers in order to design or solve engineering problems. This paper shows the solution of the static analysis, presenting the stress outputs from Ansys Workbench 17.2.

**Keywords**—Ansys Workbench, Cranes, Finite Element Method, NBR 8400, Structural Analysis.

## I. INTRODUCTION

Cranes, subject of analysis in the current study, play an important role in the replacement of labor force by the mechanical method, allowing the transport of high loads in situations where manual labor becomes limited [1]. In view of the current engineering scenario of high competitiveness and incessant search for reduction of costs and waste, it is essential that cargo handling equipment be able to resist to several types of loads during their useful life. In order to guarantee operating reliability and safety, as well as optimum performance and cost-effectiveness, is crucial the use of regulatory standards. In this way, the present study presents a numerical structural analysis of a single girder crane subjected to different cases of load combinations required by the standard NBR 8400 [2], using the Finite Element Method through Ansys Workbench 17.2 software. Many engineering problems can be solved by using differential equation. Nowadays, one of the most used method resolution is the Finite Element Method that is among the various numerical methods [3]. It was developed to solve complex engineering problems and is increasingly being

used by many engineers with an aim to simulate components and structures during project, structural reinforcements, among other activities.

In this context, the purpose of this work is to compare stress outputs from Ansys Workbench 17.2 with yield stress requirements established by NBR 8400 [2]. The contribution of this paper is to provide an example of how computational simulation can be useful and effective for a crane structural analysis, in order to simplify and optimize engineering design.

## II. TECHNICAL CHARACTERISTICS OF THE SINGLE GIRDER CRANE

An illustration of the single girder crane in study and its main technical characteristics are shown in the Fig. 1 and in the Table 2, respectively.

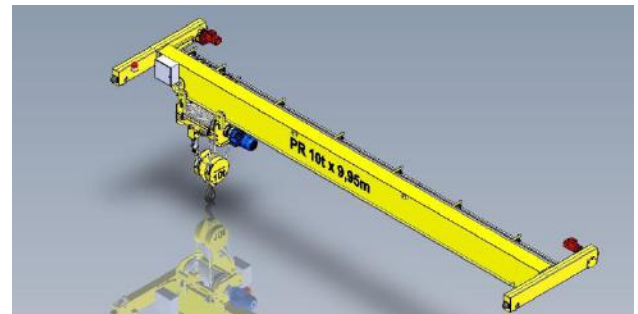


Fig. 1: Scheme of the single girder crane

The material used for the fabrication of the single girder crane is the structural steel ASTM A-36. Its mechanical properties are shown in the Table 2.

Table 1: Technical characteristics of the single girder crane in study

Load Capacity	10 ton
Free Span	9.95 m
Mechanical Group	3M
Translation speed	25 m/min
Electric hoist weight	1028 kg
Single girder weight	1519.5 kg
Braking speed	0.098 m/s <sup>2</sup>
Impact deceleration speed	0.6 m/s <sup>2</sup>
Lifting speed	0.083 m/s

Table.2: Mechanical properties of structural steel ASTM A-36.

Young's Modulus	200 GPa
Shear Modulus	77 GPa
Poisson's ratio	0.3
Specific mass	7850 kg/m <sup>3</sup>
Ultimate Strength	400 MPa
Yield Strength	250 MPa

### III. STANDARD NBR 8400

NBR 8400 [2] provides the guidelines for the correct verification of cargo handling equipment in general. These structures are classified in structural groups, according to their operational functions. In accordance with the technical characteristics of the single girder crane provided by "ARPI Engenharia", responsible for designing the structure, we conclude that the crane is classified as group 3 (as determined by the table 4 of NBR 8400[2]). Based on the selected structural group, we come to a security coefficient ( $M_x$ ) of 1. Due to mechanical loads and shocks caused by vertical movements such as lifting, a dynamic coefficient ( $\psi$ ) is also adopted. According to table 5 of NBR 8400 [2], the dynamic coefficient for a lifting speed of 0.083 m/s is equal to 1.15. At last, another dynamic coefficient ( $\psi_h=2$ ) is adopted for horizontal loads due to the deceleration of the crane.

According to NBR 8400 [2] the evaluation of structures such as cranes is made by determining the stresses during their operation. Three kinds of loads are considered for the present analysis: Main loads, vertical loads, and horizontal loads. The main loads include the self-weight (SG) of the structure (metallic structure and electric hoist weight) and the operating load (SL) of 10 ton. The vertical load is represented by the dynamic coefficient ( $\psi$ ) that multiplies the operating load by 1.15. Lastly, the horizontal loads (SH) represent the inertia effects caused by deceleration during translation movement and impact due to shock effects. The horizontal load caused by braking is applied in the Finite Element model through an acceleration and a force directly in the electric hoist support in the beam. The acceleration applied in the model is equal to 0.196 m/s<sup>2</sup> (0.098 m/s<sup>2</sup> multiplied by  $M_x$  and  $\psi_h$ ). The force applied in the model is equal to 1960 N (0.098 m/s<sup>2</sup> multiplied by  $M_x$ ,  $\psi$ , and SL). The acceleration caused by shock effects is equal to 0.6 m/s<sup>2</sup> (adopted by [4]). Table 3 summarizes all the loads applied in the Finite Element model and their respective directions according to the model coordinate system.

Table.3: Loads and accelerations applied in the Finite Element model

Type	Load	Force/acceleration	Direction
SG	Metallic Structure	14906 N	-Y
	Electric hoist	10084 N	-Y
SL	Operating Load	98100 N	-Y
SH	Direct force	1960 N	-X
	Braking	0.196 m/s <sup>2</sup>	-X
ST	Shock	0.6 m/s <sup>2</sup>	-X

NBR 8400 [2] set three cases of load combinations:

- Case I: normal operation without wind
- Case II: normal operation with wind
- Case III: exceptional loads

For the current study, only case I and III are considered in the analysis. As ASCE Standard (2005) [6], NBR 8400 [2] also approaches normal loads and exceptional loads. Thus, the following load combinations are established:

- Case I:  $M_x(SG + \psi \cdot SL + \psi_h \cdot SH)$  (1)
- Case III:  $SG + SL + ST$  (2)

For each case, there are three different positions of the electric hoist to be evaluated for the single girder crane: left, right and center of the beam. The model and respective positions are shown in the Fig. 2.

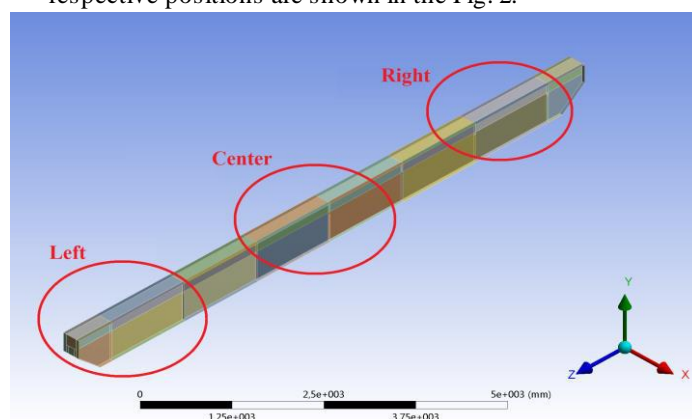


Fig. 2: Three positions of the electric hoist evaluated in the model

Table 4 summarizes the load combinations set for the structural analysis.

Table.4: Load combinations

Number	Combination	Case	Position
1	1.(SG + 1,15. SL + 2. SH)	I	Left
2	SG + SL + ST	III	Left
3	1.(SG + 1,15. SL + 2. SH)	I	Right
4	SG + SL + ST	III	Right
5	1.(SG + 1,15. SL + 2. SH)	I	Center
6	SG + SL + ST	III	Center

According to standard NBR 8400 [2], for Yield Strength/Yield Strength < 0.7, the respective values of allowable stress are:

Table.5:NBR 8400 Allowable stress

Case I	Case II	Case III
$\sigma_a/1.5$	$\sigma_a/1.33$	$\sigma_a/1.1$

The approval criterion for the shell elements used in the Finite Element Model is to present Von Mises stress below the allowable stress in all load combinations. Thus, the allowable stresses for the material of the single girder crane for cases I and III are shown in Table 6.

Table.6:Allowable stress for cases I and III

Case I	Case III
167 MPa	227 MPa

#### IV. STRUCTURAL ANALYSIS IN ANSYS WORKBENCH 17.2

In the Finite Element Method, the geometry of the component or structure under analysis is subdivided into small elements, in a finite quantity, interconnected by nodes, forming a mesh. This process is called discretization [5]. The analysis is divided into three distinct steps: pre-processing, solution and post-processing [5]. The pre-processing step consists of geometry modeling, definition of mesh, material properties, and boundary conditions. At this stage, the geometry was modelled on ANSYS Discovery SpaceClaim and exported to Ansys Workbench 17.2. For the solution step, the linear static analysis was selected in order to obtain stress and strain outputs. Finally, the structural response of the single girder crane was evaluated in the post-processing step. The model was built with SHELL181 elements in Ansys Workbench 17.2. The different thicknesses of the plates are shown in the Fig.3 in a color scale.

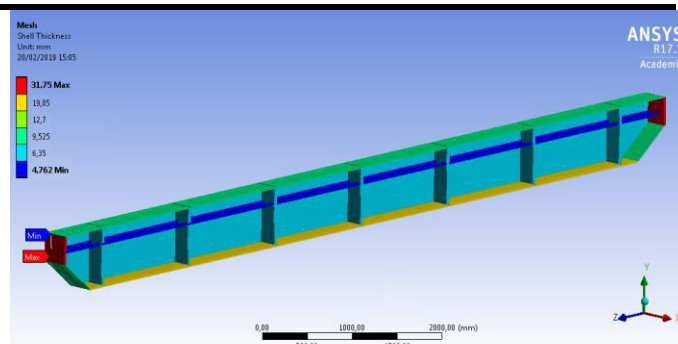


Fig. 3: Different thicknesses of the single girder crane

The model has a total of 49317 nodes and 49635 elements, including Tri3 and Quad4 first order types. Fig.4 shows in detail the defined mesh.

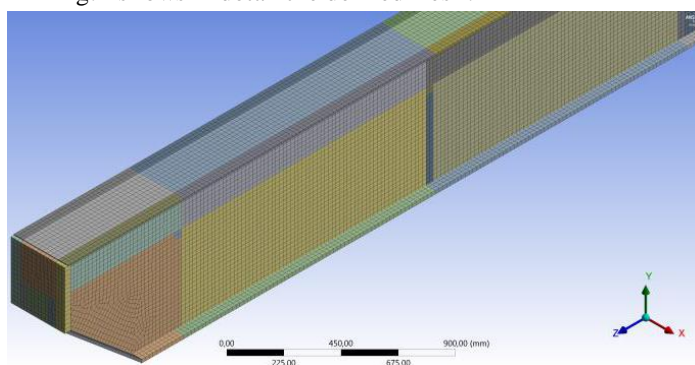


Fig. 4: Details of model mesh

The single girder crane was considered to be simply supported and the most external plates were restricted. One side was considered fixed and the opposite side had Y and Z displacement restricted, while X (longitudinal direction) was set free, according to global coordinate system shown in Fig. 4.

The most critical result of the static analysis occurred in combination 5, where the electric hoist is located in the center of the single girder crane. As previously mentioned, combination 5 includes the following loads: self-weight (SG), operating load (SL), and braking (FH). The Fig. 5 and Fig. 6 show in details, in a color scale, the Von Mises stress.

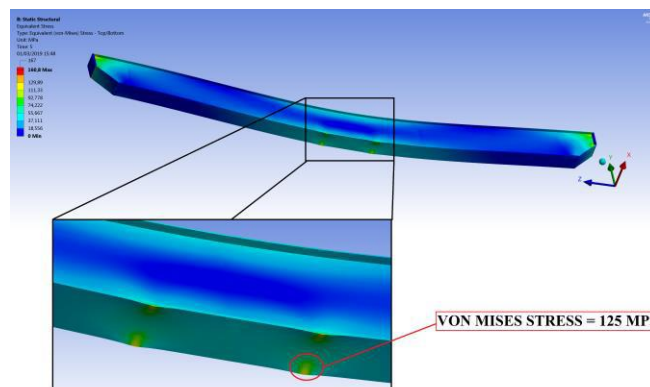


Fig. 6: Von Mises stress for combination 5

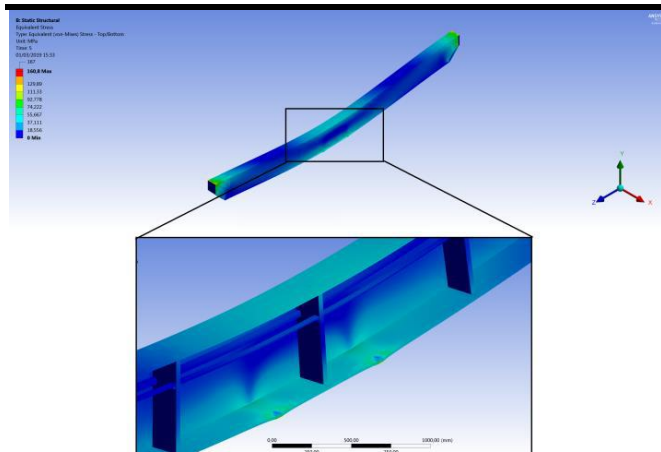


Fig. 5: Von Mises stress for combination 5

Note that the maximum Von Mises stress in the center of the beam (Critical region) is 125 MPa. It can also be noted that there is a stress concentration at the extremity plates, where the constraints were set. It can be viewed in Fig. 7.

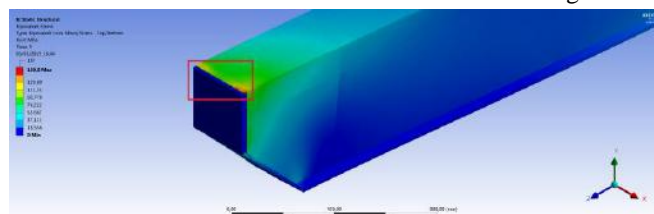


Fig. 7: Stress concentration at the extremity plate

A value of 160.8 MPa is found in this region. It can be explained because of the constraints applied in the area. As the constraints applied simulate a perfect fixed support, consequently it results in a bending moment bigger than the expected in reality, as a rigid joint transfers all the moment thru the joint. Another factor to be considered is that the single girder is connected to the end truck with bolts. It means that the rotational stiffness decreases and actually the connection looks like a semi-rigid joint. Table 6 summarizes the maximum Von Mises stress found for each load combination.

Table 6: Maximum Von Mises stress

Combination	Stress	Allowable Stress
1	124.14 MPa	167 MPa
2	115.85 MPa	227 MPa
3	139.79 MPa	167 MPa
4	115.98 MPa	227 MPa
5	160.8 MPa	167 MPa
6	138.61 MPa	227 MPa

## V. CONCLUSION

The structural analysis of the single girder crane using Ansys Workbench 17.2 and according to standard NBR 8400 [2] showed satisfactory results. The results

demonstrated that the single girder crane structure is able to resist to all load combinations from NBR 8400 [2]. Its structure has enough stiffness to operate with safety and reliability, therefore meets the required criteria of NBR 8400 [2]. The maximum Von Mises stress found was 160.8 MPa and occurred in combination 5. This value is below the allowable stress of 167 MPa. The general objective of the article was reached. The results were able to prove how efficient, practicality, and applicable the Finite Element Method is for a single girder crane structural analysis. Future possible applications and extensions could be a fatigue and buckling analysis of the structure, since the current paper only approaches the yield stress criterion. NBR 8400 [2] also provides the methodology to evaluate fatigue and buckling criteria for cranes and cargo handling equipment, therefore can be useful for possible applications.

## REFERENCES

- [1] SORDI, G. (2016), Dimensionamento da viga principal de uma ponte rolante. Centro Universitário UNIVATES, Lajeado, RJ, Brasil.
- [2] ASSOCIAÇÃO BRASILEIRA DE NORMAS TÉCNICAS - NBR 8400 (1984), Cálculo de equipamento para levantamento e movimentação de cargas.
- [3] SOUZA, Remo Magalhães de. (2003), O método dos Elementos Finitos Aplicado ao Problema de Condução de Calor. UFPA. Belém. PA, Brasil
- [4] MARQUES, Mário Sérgio Silva. (2014), Análise Estrutural de um pórtico rolante conforme as normas NBR 8400 e NBR 10084 usando o Ansys Workbench, São Paulo, SP, Brasil.
- [5] AZEVEDO, Domingos Flávio de Oliveira. (2014), Análise estrutural com Ansys Workbench: Static Structural, Mogi das Cruzes, SP, Brasil.
- [6] AMERICAN SOCIETY OF CIVIL ENGINEERS, ASCE Standard (2005), Minimum design loads for buildings and other structures.