Abstract: This paperwork analyzes the tension and deformation estate of the resistance structure of an overhead crane bridge who is used for all the processes performed in the hall of a continuous casting department of an iron and steel plant in order to find out the best sizes. This analysis is made up with the help of the COSMOS software who enables us to make evolved finite items – shell-type with 3 or 4 nodes per element. The shell-type finite items belong to the $C^1$–class items, and they have a field variable and the I-type derivates working continuously alongside the frontier, meanwhile the II-type continuous derivates per item are not continuous alongside the frontier. These finite items allow us to design some complex structures more accurately, such as the resistance structure of the crane bridge. This example is not intended to be the only solution when designing similar structures, but the authors of this paperwork believe that it is providing enough information and useful solutions for the analysis of the tension and deformation state in case of finite items.

Key-Words: crane bridge, analysis, model, resistance structure, overhead, finite elements, shell type.

1 Introduction

Most of the time, the resistance structures who have not been statically determined, cut to the right sized, and checked out by the classic methods of the material strength cause the oversizing, because specialists use approximate measurements in order to decrease the number of mathematical calculation. Modern methods who use automatic data processing allow us to study the tension more accurately, especially due to the operative method of balance and continuity equation calculation, [4],[6],[10]. As far as the validity of the accuracy and operative method is concerned, they are real is the shaping up of the structure and the connection means are as good as possible [4],[6],[10]. This paperwork analyses the tension and deformation state of the resistance structure of a sliding bridge who is used for all the processes performed in the hall of a continuous casting department of an iron and steel plant. This analysis is made with the help of the old method used for producing finite items and the COSMOS calculation software, [14]. The results we have obtained allow us to produce a complete study about the tension and deformation state of the resistance structure of a crane bridge, and to highlight all details based on the operation forces. Based on those results, we are able to produce some details about the lack of resilience of the resistance structure, and to make some decisions about how to improve the bearing capacity of the equipment, and, if there is the case, we are able to perform the best size optimization, [9]. This is possible only if we do not exceed the limits of the acknowledged resistance, in order to redesign the resistance structure of the crane bridge, meanwhile the material consumption is highly reduced.

2 The description of the resistance structure component

The strength structure is made of two longitudinal beams - I and II , as well as two end beams: left and right which make up a dark plane contour. The longitudinal beams have a variable section at their ends, fig.1 and the cross beams have a constant section alongside their lengths, fig.2. The cross section of the resistance elements – caissons – is symmetrical and made of universal iron – they are weld together, fig3. The analyzed rolling bridge have the lifting capacity of 100 KN, lifting height of 17,3 m with a gauge of 11 m and the distance between axes of 4,25 m.
The forces that stress the resistance structure are perpendicular on the structure plane and they make up a space system. The global analysis of the resistance structure has been performed for the best positions, as well as for the less favourable positions of all the forces.

### 3 Application of finite element method to an overhead crane

Among numerical technique, the finite element method is used due to the availability of many user-friendly commercial software.

The finite element method can analyses any geometry and solves both tensions and displacements [1], [2], [6], [8], [12].

Finite elements method approximates the solution of the entire domain under study as an assemblage of discrete finite elements interconnected at nodal points on the element boundaries, [3], [7], [10], [12]. The approximate solution is formulated over each element matrix and thereafter assembled to obtain the stiffness matrix and displacement and force vectors of the entire domain, [1], [2]. Due to the complex structure of the resistance we have analyzed, it is difficult to choose the type of the finite element who we should use for producing a discrete item; and it is related to the way we are able to provide the continuous process amongst the elements. A more accurate analysis needs to use the finite items who are able to approximate the resistance structure as accurate as possible. In this study finite element modeling is carried out by means of the Cosmos package, which has shell type elements with three or four nodes per element and six degrees per node in the finite elements library which secure a very good calculation accuracy, with deviations under 4 % related to the exact methods of calculation, [4] [5], [6], [10], [14]. The availabilities provided by the pack of programmers COSMOS/M, which don’t limit the analysis dimension by the number of elements or the number of nodes used, [4], [6], [10]. This type of finite elements allows us to perform a liniar or non-liniar analysis of the resistance structure of the sliding bridge. In case we shape up and we use such type of finite elements, the elements are compatible if only we should use a complete cubic polynominal, [1], [4], [6], [10].

### 4 The modelling of the strength structure

Because the geometrical model has been elaborated in accordance with the workshop drawings, and the height number of elements of shell type used at meshing has allowed a calculation model very closed to the real geometry of the strength structure of the analyzed crane bridge. In modellation with Cosmos software was used Youngs modulus \( E \) 2,1 x \( 10^5 \) N mm\(^2\) and the Poisson ratio \( \nu \) 0,3 for finite element analysis [4], [6], [11], [13]:

The boundary conditions regarding the supporting and loading way have been introduced as follows, [3], [8], [11]:

a. In the insert of blockings for certain degrees of freedom from the structure nodes, we had in view the presence of those four wheels of the crane bridge. For the nodes placed at the drive wheels level there have been introduced blockings for the linear displacements \( u \), \( v \) and \( w \) according to the three directions of the global system of axes XYZ of the structure, and for the driven wheels there have
been introduced only the linear blocking u and w according to the directions y and z.

b. The forces have been distributed in the nodes in front of their application area according to the loading diagram of the crane bridge. The existence of some eccentric loadings by means of some rigid arms, has led to their replacement with an equivalent system of loads directly applied on the structure, in order to avoid the supplementary use of some finite elements of beam type with high stiffening.

In order to design the shape of the resistance structure we have used two reference systems: a global reference system that we report the entire problem to, and a local reference system which is associated to a sub-domain of the problem we are analyzing. When modeling, the difference amongst the aggregate, the loading and the weight of the elements is calculated in condensed mass.

The solid model of the crane bridge is presented in fig. 4.

In the stage of the structure meshing here have been used a number of 42896 finite elements with a number of 20627 nodes and 12326 degrees of freedom. Fig. 5 describes the discrete resistance structure with shell-type finite elements in detail. The big number of shell-type finite elements allowed us to come up with a calculation method which is almost similar to the real shape of the strength structure we have analyzed. The calculation method we have produced helped us make up a complete study about the tension and deformation state of the resistance structure of the crane bridge and to highlight the detail of the focusing and the division of the tensions, as well as of their deformations caused by the working forces.

5 Results
The analysis of the resistance structure of the crane bridge using finite elements have been calculated all the stresses and strains tensor from the structure nodes and from the centroids of the finite elements. In the post processing stage, for a quick and efficient interpretation of the results, there have been represented under the form of spectrum the tension fields at the whole structure level. Analyzing these data, it results a series of conclusions regarding the behavior of the resistance structure of the crane bridge. We have selected some values we have considered important from amongst the analysis of the folders which contained the results. This values are presented in table 1.

We have selected those values of the stresses and strains which could cause some critical areas where stresses points are gathered (within the resistance structure of the crane bridge). We should consider that the beams of the strength structure of the crane bridge are made of OL 37 then, the analysis of stresses and strains is more effective if we use the theory of the specific form modifying energy (stated by von Mises) as a determining factor for reaching the limit stages, [11].

By analysing the tension fields, we are able to see the main critical area of the crane bridge. Thus, we have to pay all the attention while designing it as well as during the production, in case we want to redesign the structure. The research performed allows the evaluation of the stress state, pointing out the critical areas and measures which are imposed in order to increase the solidity and bearing capacity of the strength structure for the rolling bridge.
<table>
<thead>
<tr>
<th>No.</th>
<th>Analysis component</th>
<th>No. Node which is recorded the extreme size</th>
<th>The extreme recorded size</th>
<th>The location of the node on the strength structure of the crane bridge</th>
<th>Detail of the area which has extreme value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\sigma_x$</td>
<td>3123</td>
<td>132, 164 MPa</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
</tr>
<tr>
<td>2</td>
<td>$\sigma_y$</td>
<td>1702</td>
<td>-133, 064 MPa</td>
<td><img src="image3" alt="Image" /></td>
<td><img src="image4" alt="Image" /></td>
</tr>
<tr>
<td>3</td>
<td>$\sigma_z$</td>
<td>15104</td>
<td>155,315 MPa</td>
<td><img src="image5" alt="Image" /></td>
<td><img src="image6" alt="Image" /></td>
</tr>
<tr>
<td>4</td>
<td>$\tau_{xy}$</td>
<td>1732</td>
<td>- 46, 996 MPa</td>
<td><img src="image7" alt="Image" /></td>
<td><img src="image8" alt="Image" /></td>
</tr>
<tr>
<td>5</td>
<td>$\tau_{yz}$</td>
<td>1809</td>
<td>30,73 MPa</td>
<td><img src="image9" alt="Image" /></td>
<td><img src="image10" alt="Image" /></td>
</tr>
<tr>
<td>6</td>
<td>$\tau_{zx}$</td>
<td>2517</td>
<td>- 76,85 MPa</td>
<td><img src="image11" alt="Image" /></td>
<td><img src="image12" alt="Image" /></td>
</tr>
<tr>
<td>No.</td>
<td>Analysis component</td>
<td>No. Nod which is recorded the extreme size</td>
<td>The extreme recorded size</td>
<td>The location of the node on the strength structure of the crane bridge</td>
<td>Detail of the area which has extreme value</td>
</tr>
<tr>
<td>-----</td>
<td>-------------------</td>
<td>-------------------------------------------</td>
<td>---------------------------</td>
<td>---------------------------------------------------------------</td>
<td>------------------------------------------</td>
</tr>
<tr>
<td>7.</td>
<td>$\sigma_{\text{von Mises}}$</td>
<td>15104</td>
<td>182,895 MPa</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>8.</td>
<td>$\sigma_1$</td>
<td>15104</td>
<td>182,89 MPa</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>9.</td>
<td>$\sigma_2$</td>
<td>3132</td>
<td>81,92 MPa</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>10.</td>
<td>$\sigma_3$</td>
<td>1702</td>
<td>-192,554 MPa</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>11.</td>
<td>$u_{\text{max}}$</td>
<td>15991</td>
<td>20,1832 mm</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>12.</td>
<td>$v_{\text{min}}$</td>
<td>7880</td>
<td>-7,6429 mm</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
<tr>
<td>13.</td>
<td>$w_{\text{min}}$</td>
<td>15343</td>
<td>-1,1586 mm</td>
<td>[Image]</td>
<td>[Image]</td>
</tr>
</tbody>
</table>
4 Conclusions
Analyzing the data presented in table 1, it results a series of conclusions regarding the behavior of the strength structure of the overhead crane bridge. The main conclusions which results from this analysis are:

a. The maximum equivalent stress calculated according to the theory of the specific form modifying energy (the theory of von Mises) is \( \sigma_{\text{von Mises}} = 182,895 \, \text{MPa} \) and is recorded in the node 15104 placed in position No.7, on the lateral external surface of the longitudinal beam I near the connection with the right end beam. This size exceeds the allowable stress for case I of loading according to Bach, for the steel OL 37, \( \sigma_{\text{von Mises}} = 182,895 \, \text{MPa} > \sigma_a \) (\( \sigma_a = 150 \, \text{MPa} \)), with 21.92 %.

b. The maximum component in modulus of the tangential stress is the component \( \tau_{zx} \) which has the size \( |\tau_{zx}| = 76,85 \, \text{MPa} \) and is recorded in the node 2517 placed in table 1, position 6, at the upper part of the right end beam at the connection with the longitudinal beam I.

c. The components of the tangential stresses \( \tau_{xy} \) and \( \tau_{zy} \) recorded size in the nodes 1732 and 1809, as follows: \( \tau_{xy} = -46,996 \, \text{MPa} \) and \( \tau_{zy} = 30,732 \, \text{MPa} \).

These nodes are very closed and are placed according to table 1, positions 4 and 5 on the end of the right end beam near the origin of the global system of axes. In the same area, corresponding to the node 1702 is recorded the extreme value for the normal stress \( \sigma_x = -182,226 \, \text{MPa} \) and the main normal stress 3 with the size \( \sigma_3 = -192,554 \, \text{MPa} \).

d. According to the conclusions from point c, it results that the end beam area right placed near the origin of the global system of axes, is a critical area for which have to be taken measures of improving the constructive solution in order to decrease the peaks of stress which appear. Similar situation appear according to conclusions from point a and b at the connection between the right beam and the longitudinal beam I.

e. The maximum total displacement is recorded in the node 15992, being of 20,87 mm, for which the linear displacement components according to the three directions of the global system of axes are: \( u = 15,18 \, \text{mm}, v = 3,75 \, \text{mm}, w = 0,075 \, \text{mm} \). The extreme values of these components of displacements are recorded as follows: \( u_{\text{max}} = 20,1832 \, \text{mm} \) in node 15991, \( v_{\text{min}} = 7,6429 \, \text{mm} \) in node 7880, \( w_{\text{min}} = -1,1586 \, \text{mm} \) in node 15343,according to the location presented in table 1, positions 11, 12 and 13.

f. The complete study of the issue presented assumes an analysis of the structure behavior in dynamic conditions after the constructive solution improvement in order to reduce the peaks of stress.

References:
[14] COSMOS User`s guide, Structural research analysis corporation, Santa Monica, 1999