Quantitative and qualitative study on the state of stresses and strains of the strength structure of a crane bridge

CAMELIA BRETOTEAN PINCA, GELU OVIDIU TIRIAN, ANA JOSAN, GLADIOLA CHETE
Department of Mechanical Engineering and Management
"Politechnica" University of Timişoara
Revoluției str., no. 5, Hunedoara
ROMANIA
camelia.bretotean@fih.upt.ro, ovidiu.tirian@fih.upt.ro, ana.josan@fih.upt.ro,
gladiola.chete@fih.upt.ro

Abstract: - In heavy-duty material handling equipment, major concern is optimum utilization of material for equipment construction without sacrificing the design parameters. To understand this aspect and also to validate the design, as this kind of equipment are vital part of any manufacturing industry, finite element analysis is one of the best method that can be used extensively. This paper work provides a modeling of the structural strength of a crane bridge, in operation for 15 years in the continuous casting department of a steel plant. Due to the importance of the cranes bridge within the technological processes in which they are involved, either from the point of view of steel consumption or functionality, their resistance calculation, during the design phase and after a certain operation period, constitutes a permanent field of investigation which aims a faithful schematization to be used in further modeling, appropriate for our target.

Modeling will be done with the aid of a COSMOS/M software in two numerical variants using beam3 and shell 3 type finite element. This paper work is meant for checking out the limit state of the structure subject to normal operation, which involved the evaluation of its reaction while under working in terms of quantity and quality. In quantitative terms, we have revealed that the structure had reserves of strength; qualitatively speaking it had confirmed the validity of modeling.

Key-Words: - crane bridge, modeling, strength, structure, finite elements, shell.

1 Introduction

The proceedings used to calculate the cranes bridge strength structures are influenced by the evolution of the optimization concept. The difficulties of the practical use of the deterministic and probabilistic concepts, mentioned in the studies regarding the calculation methods of the strength structures [7], [8], justify the application of the semi probabilistic optimization concept, with its afferent calculation method: the limit state method. Within this calculation method, the determination of the bearing capacity of the structure elements is based on the differentiated safety factors.

The concepts and notions that the calculation method are defined by the norms [11], [12]. The method resides in providing the reasonable elements and the strength structures referring to the limit states who involve either loosing the feature of providing the use conditions or enabling some possible damage for people or goods [4], [8].

According to [4], [8], the limit resistance states are divided into two categories:

- final use limit states (they reach the highest values of the bearing capacity);
- normal use limit states (if the equipments reach these limits, they might damage the working of the equipment or of an element).

If we are able to define the two states, then we are able to calculate the strength structures during the designing and the working, [8].

This situation is explained by the current trend in the calculation of strength structures which is designed to complete coverage of all factors involved in their behavior while in service, in order to ensure the most efficient solutions in terms of strength, durability, and savings. In both situations you need to have a physical and mathematical modeling as accurate as possible. Most of the time, the strength 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 over sizing, because specialists use approximate measurements in order to decrease the number of mathematical calculation. There are many published studies on solid modeling of cranes bridge in exploitation, [1], [2], [3], [4], [5], [6], [7], [8] .

Oversized of the strength structures of the metallurgical equipment requires a through analysis of all factors derived from the characteristic functional role of all technological processes involved. This is a condition for the most fitted working of structural composition resolution during the design stage; meanwhile it is a driver for specific upload that influence the behavior effects of the strength structure during operation.

Metallurgical industry develops some specific processes whose characteristics determine the category of machines classification: heavy duty and hard working.

The cranes are the most used tools in this area. Thus, the general pattern for choosing a crane bridge (geometry, type structure) should be consistent with the requirements of technological process and operating conditions.

In this context, the design features very widely according to their major operational specifications, such as: type of motion of the crane bridge structure, weight and type of the load, location of the crane bridge, geometric features and environmental conditions.

It is difficult and practically impossible to consider all factors involved in the strength structure of a crane bridge in use. Some simplifying assumptions on the behavior of structure element material, geometric design, and loads' action are therefore allowed. Following the acceptance of these simplifying assumptions, we obtain a physical design of the real structure. Both physical pattern and mathematic method define the calculation method. We are then able to define the stresses state due to the mathematical model.

The calculation allows us to analyze the state of stress and strains of the crane bridge strength structure under the action of loads in mining stock. The purpose of modeling is to perform an analysis of the state of stress and deformation of the structure which allows decision making in terms of strength, durability, and money saving. The results depend on the calculation method we are going to use.

On the other hand, the elaboration of the calculation model afferent to such complex strength structures, as the specific structures of the cranes bridge, should be strictly correlated with the chosen calculation method. This correlation is absolutely required, because the model will reflect the chosen simplified hypotheses along with the advantages and disadvantages of the calculation method, whilst the chosen calculation method will substantiate the calculation itself.

The work is carried out for modeling the strength structure of a crane bridge, in operation for 15 years, in the continuous casting department of a steel plant. Modeling was done with the finite element program COSMOS/M, which has some 3-shell advanced finite elements which have a high calculation accuracy and do not limit the size of the analysis by the number of finite element or nodes. They are able to fulfill all conditions of consistency and balance, both within themselves and for the entire structure.

For the same strength structures, we realized a modeling by using finite elements of beam 3 type, existing in the library of the same calculation program, which was to subject of another scientific research, [3]. Comparing the results, we can find conclusions regarding the efficiency of the modeling and the importance of choosing finite elements. In this paper, we approached the semi probabilistic concept, and the afferent calculation method was the limit state method. In this context, we are going to use various measures, some of them bearing a deterministic character, and others a probabilistic one.

It enables the analysis of the state of stress and deformation with greater vigor due to promptness in solving the equations of equilibrium and continuity. The results are particularly close to the real, as the structural modeling and related conditions were well chosen.

2 The technical parameters of the crane bridge

The crane bridge we analyze carries several casting ladles filled with incandescent steel from the steel mill to the continuous casting department within a steel plant. The driving unit is installed on the strength structure.

The main parts shown in fig. 1 are: 1- cab control; 2- electric plant and additional items; 3- strength structure; 4- railings; 5- moving mechanism; 6 – lifting mechanism.

The design values used in the crane bridge analysis is from the F.E.M and DIN standards. The technical characteristics of the crane bridge are: lifting capacity of 100 KN, lifting height of 17,3 m, gauge of 11 m; distance between axes 4,25 m; trolley velocity 10 m/min; crane bridge velocity 63 m/min; total duration of use U4, load spectrum class Q3, appliance group A5, loading type H; dynamic coefficient $\psi = 1{,}15$, dynamic coefficient $\gamma_c = 1{,}11$, [4], [5], [6].

The strength structure of the crane bridge is presented in fig.2.

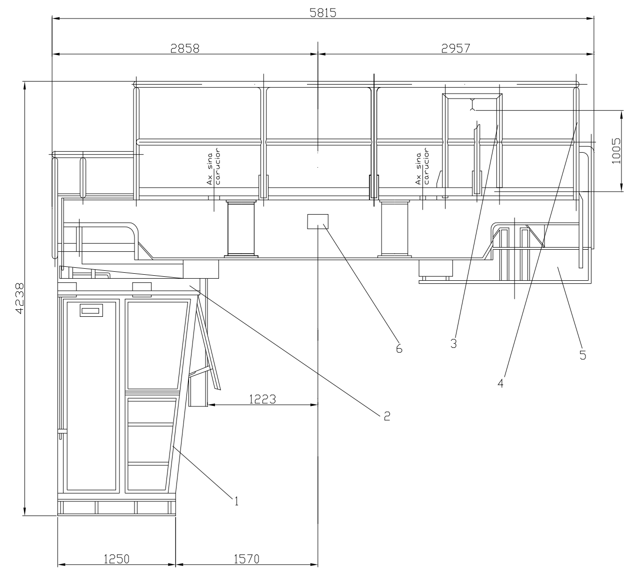

Fig.1 General presentation of the crane bridge

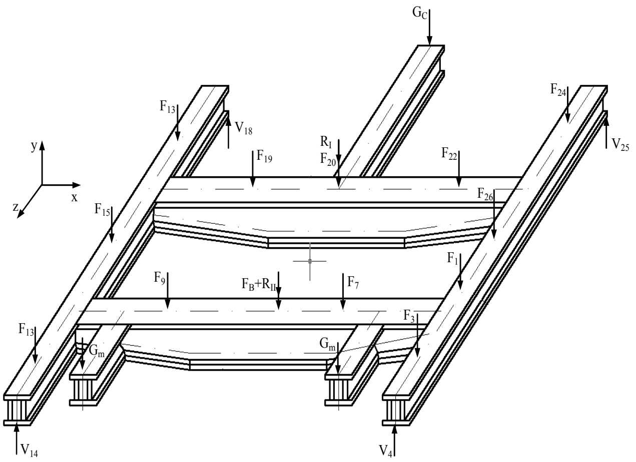

Fig.2 The strength structure of the crane bridge

This strength structure is made of locker-beams - both the two longitudinal beams and the two end beams. They made up a close-plan rectangular area. The longitudinal beams are very long and stressed, thus we make them according to a variable cross-section within the core of the beam, so that they could lift up the highest loads and work highly accurately. The height of the longitudinal beams is reduced at both ends, so they work like the „equal-resistance beams”, [5]. The values of the forces are the following: F_1=F_{26}=2800N, F_3= F_{13} =4375N, F_7=F_9=9700N, F_8=30807N, F_{17}= F_{24}=4125N, F_{19}= F_{22}=3375N, F_{20}=15774,5 N, R_I= 15774,5N R_{II} = 32000N, G_m = 9500N, G_c = 6150N

The values of reactions are: V_{25} = 87048 N, V_{14} = 87048 N, V_{18} = - 13119,304 N

3 Elements about modeling efficiency

The use in the engineering calculations of the specific strength structures of the cranes bridge along with the analytical proceedings known from the Theory of elasticity, rises significant difficulties, on the one hand due to the complexity of the structure geometry an the loading pattern, and on the other hand due to the different mechanical properties of the component elements. For this reason, the calculations are conducted on simplified structures, equivalent with the real ones, in order to replace the integration of the equations that govern the tension and deformation conditions. Because the results obtained on such simplified model are far from the concrete solutions of the analyzed structure, we shall use numerical methods for solving these structures. One of them is the finite elements method, on which was structured the COSMOS program used in this paper. For obtaining results that reflect real-life operational situations, the calculation model should be properly established. To establish the model used to calculate the analyzed strength structure, we followed the steps showed in the diagram presented in fig. 3.

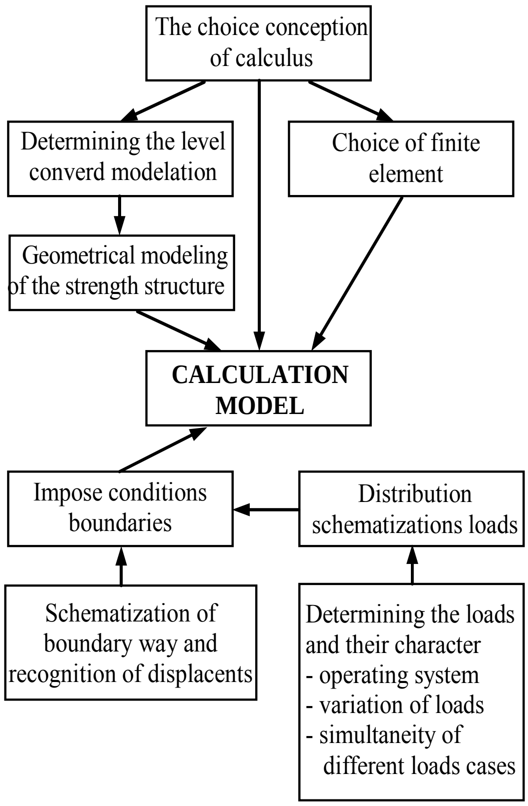

Fig.3 Interdependation between the stage of the calculus model

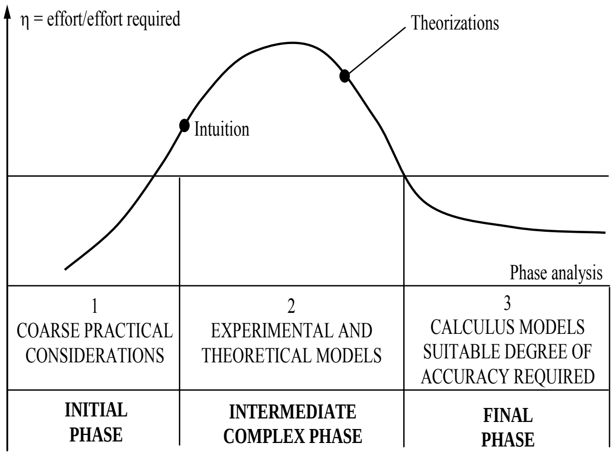

Fig.4 The evolution of the modeling efficiency

The efficiency of the calculation model applied to the analysis with finite elements depends on the finite elements type and the fineness grade of the discretization, both correlated with the necessary accuracy of the results [7]. The efficiency of any modeled of the behavior of a structure depends on the rapport between the actual analysis effort and the effort required to solve the modeled problem. This rapport varies according to the evolution of a common calculation model, fig. 4.

Knowledge of emerging areas of stresses concentration is important for designing the crane bridges. In this regard, it is important to understand the state of stress and deformation of the structure of strength occurring on a calculation model. Its effectiveness depends on the type of finite elements and the degree of fineness of the mesh, correlated with the accuracy required in the problem we study about [7].

4 The modeling of the strength structure using beam 3 finite elements

In the paper [3], there is presented the modeling of this crane bridge, but where the problem solving is done using finite elements of beam type with rigid nodes beam3D. The real constants of the structure component elements, which represent the main geometric characteristics of the cross section, were calculated as follows fig. 5.

Flats component areas:

$$A(x) = 2\delta \cdot [B + H_x] \quad (1)$$

Moment of inertia:

$$I_z(x) = \frac{B \cdot \delta^3}{6} + 2 \cdot \delta \cdot B \cdot \left[\frac{H_x + \delta}{2}\right]^2 + \frac{\delta \cdot H_x^3}{6} \quad (2)$$

$$I_y(x) = \frac{\delta \cdot B^3}{6} + \frac{\delta^3 \cdot H}{6} + 2 \cdot \delta \cdot H_x \left[\frac{162}{x}\right]^2 \quad (3)$$

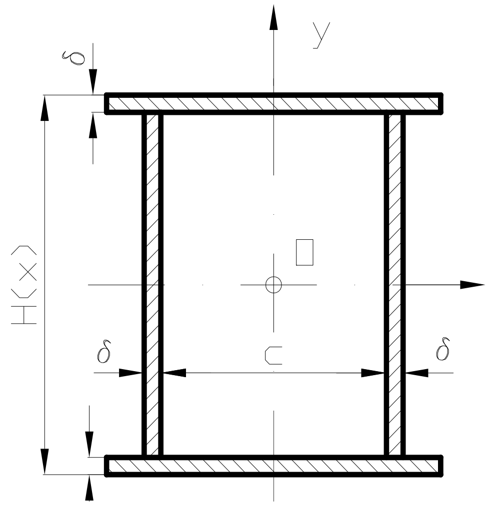

Fig. 5 The dimensions of the cross sections of the resistance structure elements

Twist moment of inertia:

$$I_t(x) = \frac{2 \cdot \delta \cdot (B^* + \delta)^2 \cdot [H_x + \delta]}{B^* + H_x} \quad (4)$$

Resistance mode:

$$W_t(x) = 2 \cdot \delta \cdot B^* H_x \quad (5)$$

In the previous relations, the variable x represents the position of the element in relation to the end of the longitudinal beam, and the width B^* has the values: (290+δ) for cross beams and (240+δ) for longitudinal beams. The variable height segments were assimilated with six elements with constant rigidity per element, which are modifying in steps from one element to another. The geometric characteristics were determined for eight sets of elements (I, II, III, IV, V, VI), of which the set I corresponds to the cross beams, the set II corresponds to the middle segments of the longitudinal beams, and the sets III, IV, V, VI, VII, VIII correspond to the variable height segments (Table 1).

Table 1
The geometric characteristic of the longitudinal and end beams sections

No. of elem.	B mm	H mm	A mm^2	I_z 10^{-8} mm^4	I_y 10^{-8} mm^4	I_t 10^{-8} mm^4
I	425	600	12156	2,3131	2,313	4,168
II	350	762	13200	1,903	1,903	4,441
III	350	614	11568	3,636	3,363	3,451
IV	350	642	11898	1,69	1,69	3,651
V	350	669	12224	1,744	1,744	3,846
VI	350	695	12544	1,796	1,796	4,04
VII	350	722	12863	1,848	1,848	4,235
VIII	350	749	13138	10,90	1,901	4,432

The analysis of the resulting files helped us to determine the forces, the bending moments, the twisting moments and the respective tensions. Based on these results, we plotted the variation of the bending and twisting moments, for the entire strength structure. The program allowed us to determine the normal, tangential and equivalent tensions (σ, τ and σ_{ech}, respectively), based on the von Mises theory of materials resistance.

We have come to the conclusion that the highest stresses occurring in the mid open part of the spars and mid sleepers and the stresses inside the longitudinal beams with variable section, are three times lower than those corresponding to the mid opening, and the ratio of maximum stresses in the longitudinal beams and end beams equivalent is 3.56, [3].

The program allowed the representation of the deformed position of the strength structure versus the non-deformed one, without being necessary to study the structure rigidity. Such an approach allows a general study of the structure mode of behavior, but without making evident the aspects concerning the phenomena of stress concentration or a detailed studying of the stress-strain distribution, [3]. From the stresses state research, we remarked that the maximum equivalent tension values don't exceed the admissible tension values (for the steel grade OL 37, used to make the resistance structure elements, is 150 N/mm^2). The main inconvenient of this modeling is the fact that it doesn't allow us to study in detail the tension and deformation states of the analyzed strength structure, so the chosen finite element doesn't model precisely the behavior in operation of the analyzed strength structure. The finite elements of beam 3 type don't model properly the joint areas of the caisson elements and neither the joints between the longitudinal and end beams sections.

This inconvenient is solved by using evaluated finite elements of beam 3 type. These ones are more adequate for our purpose, from the point of view of the size and form, modeling more precisely the analyzed strength structure under the action of the loads in operation.

Due to the importance of the bridge cranes within the technological processes in which they are involved, either from the point of view of steel consumption or functionality, their resistance calculation, during the design phase and after a certain operation period, constitutes a permanent field of investigation which aims a faithful schematization to be used in further modeling, appropriate for our target.

5 The modeling of the strength structure using shell 3 finite elements

Getting the physical model of the strength structure of a crane bridge also encloses organizing the entry data with the help of the computer software COSMOS/M, [9], [10] as a data file structured into distinct blocks containing information about the composition structure, computing loads and their combinations, position loads on the structure and options for the type of the adopted calculation pattern. Structure topology was shaped up based on drawings of constituents with the help of a GEOSTAR module which allowed us obtain all the space of spars, rails and joints between them, fig.6 [4].

The overview of the strength structure was achieved for the worst position of all operating loads. Specifying structure dimensions for the software was achieved from a total of 42 896 finite elements - advanced thin plate-type – who have 20572 units and 123726 degrees of freedom [4], [5],[6].

The discretization operation has in view the variations of the real structure geometry. The presence of concentrated forces determines their layout around the nodes and nodal lines or surfaces. The calculation is conducted in the elastic domain and is based on linear tension-deformation behavior.

In fig.7 there is a detailed description of the end beam II with 3-shell finite element mesh-type, near the global reference system XYZ, and the entire field of problem analysis reports to that. In fig.8 there is a cross section through the end beam II with finite element, [7].

The geometrical pattern of the strength structure is presented in fig.9.

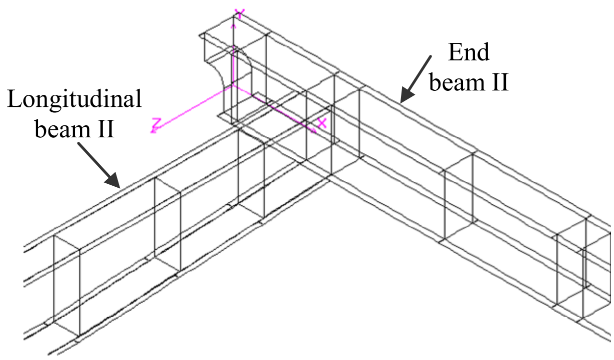

Fig.6 Detail of wireframe type of the strength structure, the combination of end beam II and longitudinal beam II, identifying the global system of axes XYZ, [4]

The large number of shell 3 finite element in which the mesh structure allows the modeling is very close to the actual geometric design of the crane bridge, [4], [5], [6].

When modeling, the difference amongst the aggregate, the loading and the weight of the elements is calculated in condensed mass.

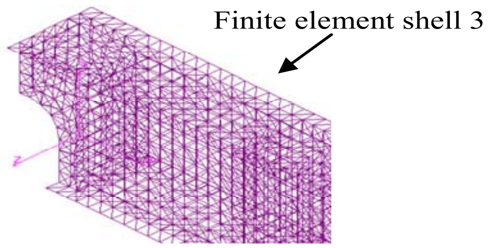

Fig.7 Detail of the end beam II, near the global reference system, with elements of 3-shell type

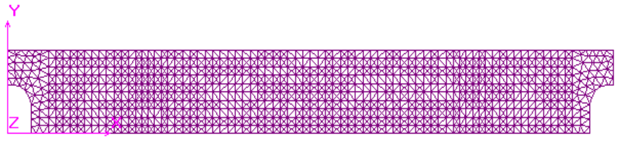

Fig.8 Median section through the end beam II

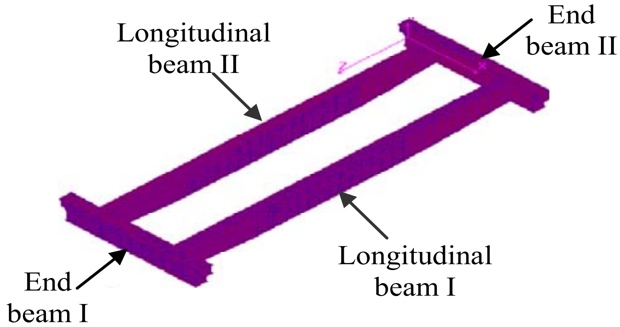

Fig.9 The geometrical pattern of the strength structure and its mesh with finite element shell 3

The loadings have been considered in the elastic field, and therefore the elastic constants have been introduced, corresponding to the material OL 37, Youngs modulus (E) $2{,}1x10^5$ Nmm^2 and the Poisson ratio (υ) 0,3.

The mathematical modeling of the behavior of the strength structure was achieved through the COSMOS/M computer software, due to core sequences, and it allowed the determination of equivalent voltages according to specific energy modifying the shape, as well as of the main and the tangential stresses, [4],[5],[6]. The calculation is enabled by a matrix. In fig.10 the calculation pattern used for the analysis of the crane bridge strength structure and fig.11 is presented calculation model of the structure, near the global system of axes XYZ chosen.

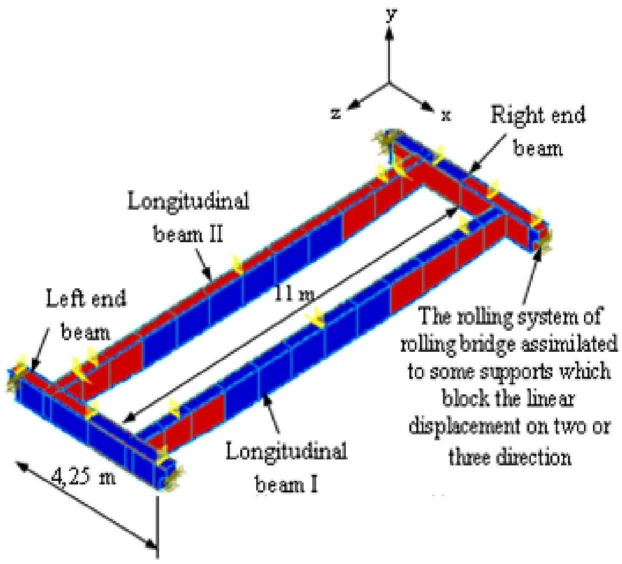

Fig.10 Solid model of the strength structure, [4], [5], [6]

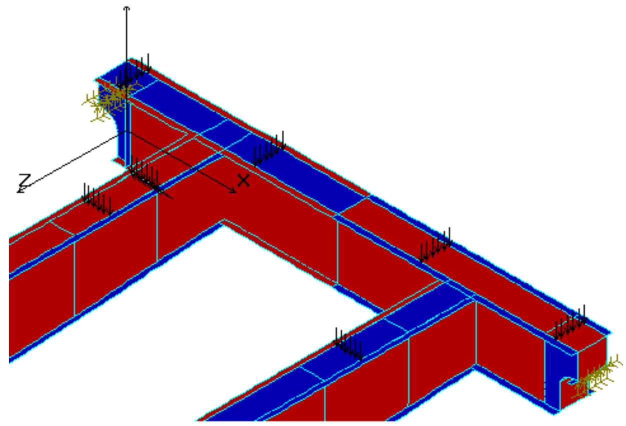

Fig. 11 Calculation model of the strength structure near the axis system

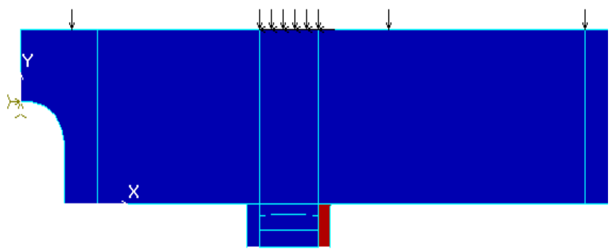

Fig.12 Longitudinal section through the calculation model afferent to the end beam II, near the axis

In fig. 12 is present a longitudinal section through the calculation model afferent to the end beam II, near the axis system.

This has allowed the elaboration of a calculation model where the approximations introduced by geometrical modeling are negligible.

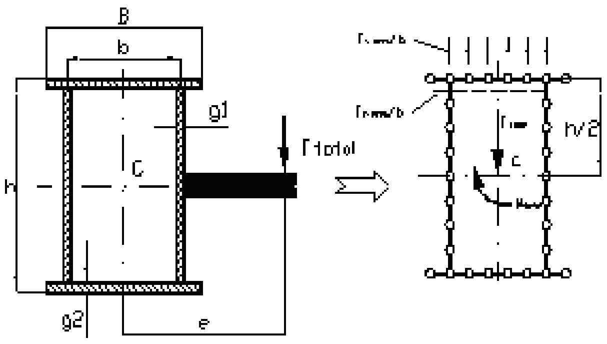

Fig.13 The equivalent loading diagram for the eccentrically applied loads, [4]

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.

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 system beam type with high stiffening. The vertical equivalent forces $F_{v\ echiv}$ and the horizontal ones $F_{h\ echiv}$ are applied to those six nodes placed at the upper surface level of the cross section of the structure in the front of their application area, [4], fig.13. The loadings have been considered in the elastic field, and therefore the elastic constants have been introduced, corresponding to the material OL 37.

The influence of the schematization pattern of a strength structure under operating loads on the calculation of the design space was revealed by calculating the stresses and deformations. Thus, we have been able to establish the stresses and strains of the mesh structure inside all nodes and gravity centers of the finite elements. Both stresses and movements have been studied in a global system of axes XYZ.

6 The analysis of the stresses state

After a thorough analysis of results files, we selected several values we considered significant both for stresses because they generate critical areas within the strength structure of the crane bridge. Thus, in terms of normal stresses, the most requested areas are located towards the boundaries. Therefore, the nodes required and the related stresses values are shown in table 2. Nodes position of strength structure is shown in fig.14.

The calculation software allows us to see the elements of the tension tensor at the level of the whole strength structure of the equipment we analyze: τ_{xy}, τ_{yz} and τ_{xz}, as well as in the case of the critical areas, when all the elements reach their highest values. In fig. 15, we describe the variation spectrum of the tangential tension τ_{xy} for the entire strength structure of the crane bridge.

Table 2

The values of the normal stresses for the most requested areas of strength structure

Analysis component	**No. nod which is recorded the extreme size**	**The extreme recorded size [N/mm^2]**
σ_x	3123	132,164
	3080	-133,064
σ_y	1702	-182,226
	1698	179,63
σ_z	15104	155,315
	14998	-145,691

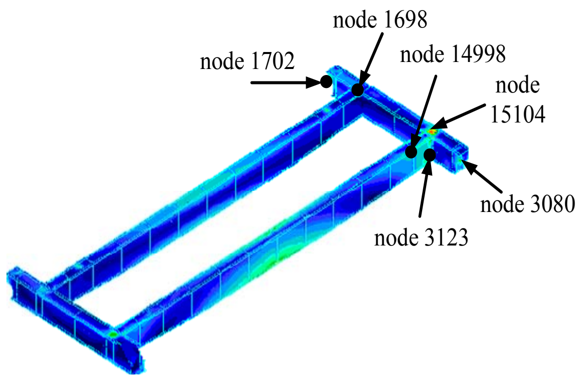

Fig.14 Node position with extreme values of normal stresses σ_x, σ_y and σ_z

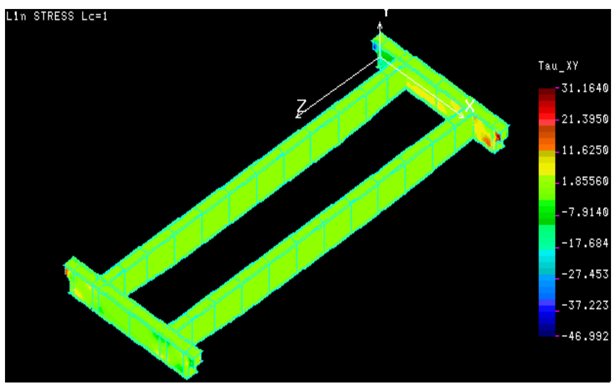

Fig.15 Variation spectrum of the tangential tension - τ_{xy} - for the whole strength structure of the crane bridge

If we analyze the tangential tension τ_{xy}, we see that the variation domain is within 46,992 N/mm^2 – 31,1640 N/mm^2 .

The variation domain for τ_{xz} is within 76,853 N/mm^2 – 57,853 N/mm^2, fig.16, and for τ_{yz} is within 30,067 N/mm^2 –30,732 N/mm^2, fig.17.

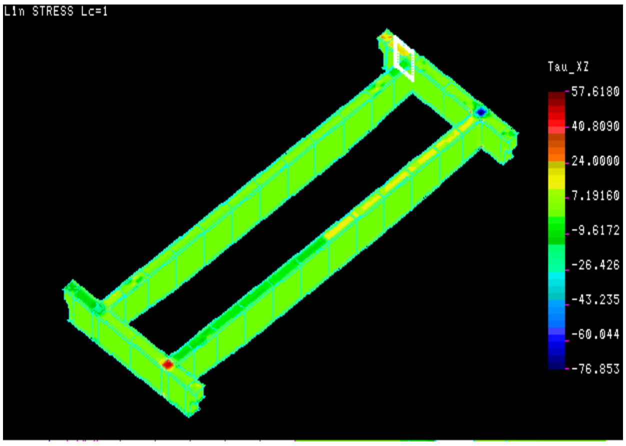

Fig.16 Variation spectrum of the tangential tension - τ_{xz} - for the whole strength structure of the crane bridge

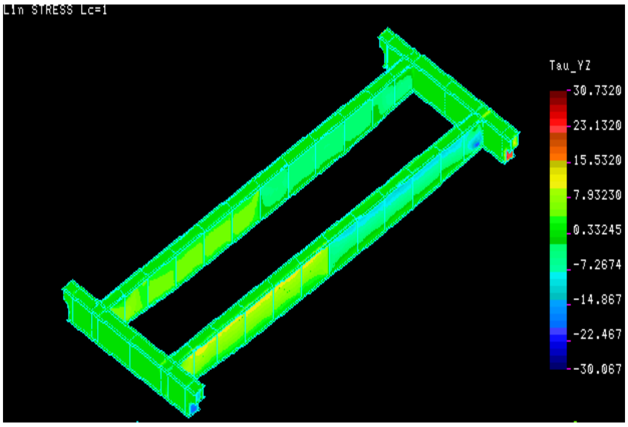

Fig.17 Variation spectrum of the tangential tension - τ_{yz} - for the whole strength structure of the crane bridge

Table 3
The maximum values of the tangential tension for the whole strength structure of the crane bridge

Analysis component	**No. nod which is recorded the extreme size**	**The extreme recorded size [N/mm^2]**
τ_{xy}	1732	- 46,996
τ_{yz}	2517	- 76,85
τ_{xz}	1809	30,73

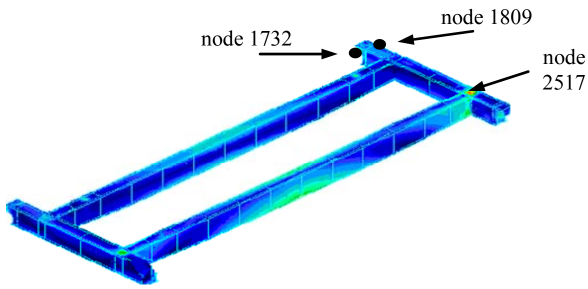

Fig.18 Node position with extreme values of tangential stresses τ_{xy}, τ_{zy} şi τ_{zx}

All elements of the tangential stresses items: τ_{xy}, τ_{zy} and τ_{zx}, for some critical areas where we have obtained high values are presented in table 3 and fig. 18.

Thus, in terms of principle normal stresses σ_1, σ_2, σ_3,, the most requested areas are located in the joint area, between end beam II and longitudinal beams, table 4.

Table 4
The maximum values of the principle normal stresses for the whole strength structure of the crane bridge

Analysis component	**No. nod which is recorded the extreme size**	**The extreme recorded size [N/mm^2]**
σ_1	15104	182,89
σ_2	3123	81,92
σ_3	1702	-192,554

We see that the nodes where the main normal stresses reach the highest values overlap with nodes that reach extreme values of normal stresses.

The analysis of stresses state has been effectively achieved by applying specific shape changing energy theory,(von Mises) as a determining factor in reaching the limit states, [4], [5], [6] .

In fig. 19 are presented the variation spectrum of von Misses equivalent stresses for the entire strength structure of the rolling bridge.

In fig. 19, we see that the variation of the von Mises equivalent tension ranges from -168.98 N/mm^2 to -0.1452 N/mm^2.

Extreme values of equivalent voltages are presented in table 5, and node position related to the structure of nodes is examined in fig. 20.

As can be seen in fig. 14, 18, and 20, joint area between end beam II and longitudinal beam I represents a critical area for the strength structure of the crane bridge under analysis.

To highlight how the variation of stresses in critical areas changes, we have described the components through a variation graphic design for 20 successive nodes and the distances between nodes was considered into parameters, meanwhile the the routes of the successive nodes are marked by presences of contoured lines and background in which the representation began, fig.21, 22, 23, [4], [6].

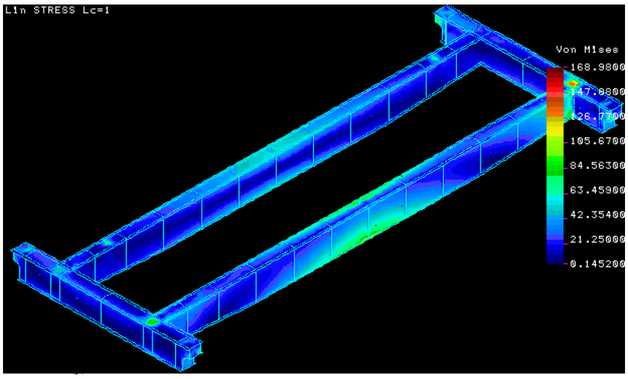

Fig. 19 Variation spectrum of von Misses equivalent stresses for the entire strength structure of the crane bridge

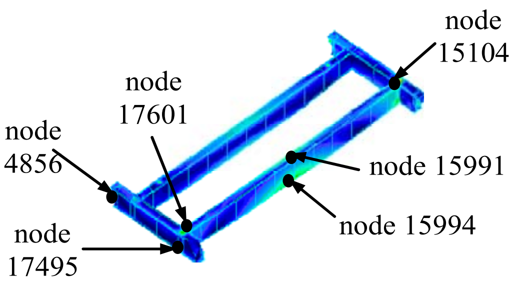

Fig.20 Node position with extreme values of von Mises equivalent stresses

Table 5
The extreme values of von Mises equivalent stresses

The analysis area	No. nod which is recorded the extreme size	The extreme recorded size [MPa]
The middle of longeron I, superior	15991	- 168,98
The middle of longeron I, inferior	15994	- 147, 88
The connection of the longeron I with end beam II	15104	182,89
The connection of the lonjeron I with end beam I	17495	133,99
The connection of superior face of longeron I with superior face of end beam I	17601	87,735
End beam I	4856	168,236

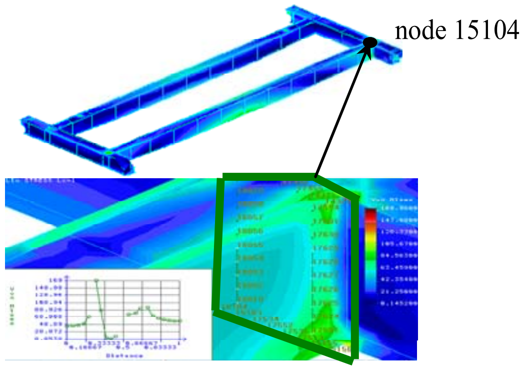

Fig.21 Variation graphic for von Mises equivalent stresses for twenty successive nodes, including node 15104 who has the highest value

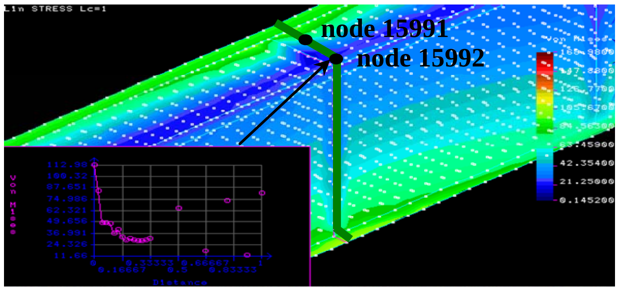

Fig.22 Diagram variation of von Mises equivalent stresses at the middle of the longitudinal beam I

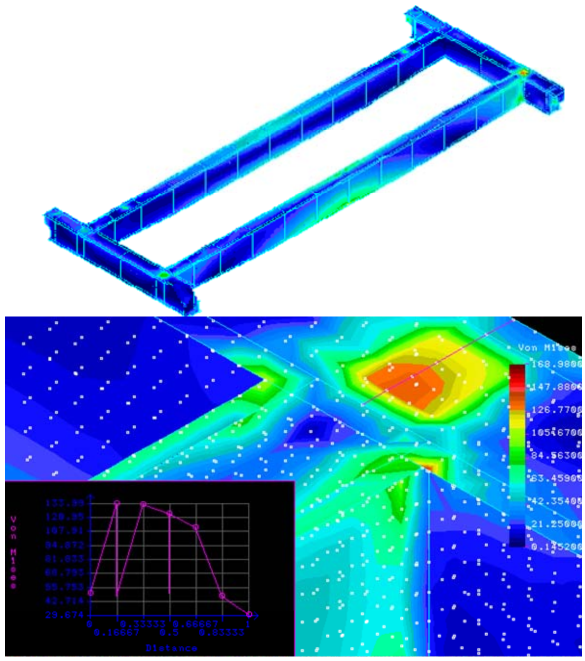

Fig.23 Diagram variation of Von Misses equivalent stresses at the level of the outside edge of the longitudinal beam I and end beam II, [4], [6].

Analyzing the results obtained in terms of stresses and the maximum, fig.20, 22, 23 and table 5, we see that the mid area of the longitudinal beam I is critical for which action will be taken to improve the state of stress if the strength structure is redesigned.

By analyzing the stresses fields, we are able to see that the main critical area of the crane bridge is represented by the connection between the longitudinal beam I and the end beam II, fig.23. 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. Also, the area of the end beam II near the global axis system needs special attention in order to improve the product and to eliminate any possible tension peak.

7 The analysis of the strains state

Analyzing the movements obtained from 3-shell finite element modeling had shown that they fall within the 1,6 mm and 20,87 mm range of values.

After analyzing any displacement we have seen that the highest values are recorded at the middle of the opening of the longitudinal beam I, fig.24 .

We find the highest displacement in case of 15992 node, which reaches u_{rez}= 20,87 mm. This node is positioned in the middle of longitudinal beam I, on the superior side.

These results are possible because the longitudinal beam I is twisted and bent and the shape of the cross section has been designed to take over the bending caused by the vertical loads, [4], [5], [6].

Also noted that neighboring nodes have obtained values close to the maximum values: the 15989 node, u_{rez} = 20.86 mm, the 15994 node, u_{rez} = 20.85 mm and 15988 node, u_{rez} = 20, 85 mm. After we have analyzed all the elements referring to the deformation, we concluded that the longitudinal beam I represents the critical area for the strength structure. The analysis of the deformation state of the structure allows a study on structural rigidity in use.

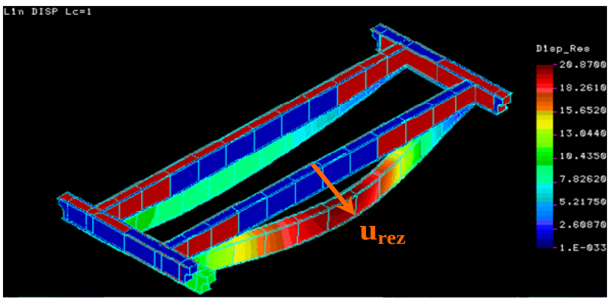

Fig. 24 The rolling bridge in strain state under the action of the nominal loads

Quantitatively, the displacements are, in general, small, so the structure presents a low deformability. This means that no special problems regarding the functionality are envisaged.

The replacement of the eccentric loads, determined by the rigid arms of the structure, with forces applied directly on the structure, has an ignorable influence on the deformation state.

6 Conclusions

The calculation pattern we have obtained by modeling with finite element of beam 3 type has allowed us understand how the strength structure of the crane bridges works, without revealing any aspect of the stresses concentration. Also, the pattern we have obtained does not allow a thorough analysis of the stress state and deformations within the whole strength structure. These drawbacks have been solved by shell-type finite element modeling. They are more appropriate for the intended purpose and strength structure.

The volume of calculations in finite element modeling with shell 3 type is much higher than in finite element modeling beam 3 type, which is directly proportional to the number of nodes and finite elements.

All modeling revealed that the operation mode of the structure varies according to the calculation pattern adopted from a quantitative point of view, and behave identically from a qualitative point of view. Different quantitative behavior revealed that the strength structure has inseminated strength reserve, so it may be subject to dimensional optimization by reducing the thickness of plate beams without exceeding allowable material strength; the same qualitative behavior confirms the validity of the model calculation. In case of strength structure, choosing a certain calculation pattern has a quantitative and qualitative influence on the structure components' state of application, with direct implications on the process of optimization. All results of this study can be conducted as a basis for developing the software used for tracking down the crane bridge strength structure and for redesigning crane bridge strength structure.

Adopt a calculation model is quantitative and qualitative influence on the state of application components of the strength structure of crane bridge in use after a relatively long time.

The possibility to analyze the areas with tension concentration leads to the possibility to calculate a theoretical tension concentration coefficient, with further implications into the durability calculations.

Calculation made on the strength structure of crane bridge which working in the continuous casting department of a steel plant, led on two numerical modeling allows analysis of the stresses and deformation state in its entirety in order to evaluating the operating behavior after a period of 15 years.

The purpose of this evaluation was to give new design solutions on improving areas of tension concentrator's duration to extend life of equipment.

References:

[1] Alkin C., Imrak C.E., Kocabas H., Solid modelling and finite element avalysis of an overhead crane bridge, *Acta Polytechnica*, Vol. 45 No. 3, 2005, pp. 61-67

[2] Ketill P., Willberger N.E, Application of 3D solid modelling and simulation programs to a bridge structure. PhD. Thesis, Chalmares University of Technology, Sweden

[3] Neguț., N.,Hluşcu,M. Pinca B. C.-*The analysis of the strength state of a crane bridge avoiding dimensionale optimization*, 12 th National Symphosium of Mechanical robots, Timişoara, 1994

[4] Pinca B. C., Tirian G.O - *The analysis of the stresses and strains state of the strength structure of a rolling bridge for increasing its solidit*y, 2nd International Conference on Engineering Mechanics, Structures, Engineering Geology, 22- 24 July, Rodos Island, Greece, ISSN 1991-1769, ISBN: 978-960-474-101-4

[5] Pinca B. C., Tirian G.O - *Finite element analysis of an overhead crane bridge*-2nd International Conference on Finite Defferences- Finite Elements- Finite Volumes- Boundary Elements, Tbilisi, Georgia, June 26-28, 2009 ISSN 1790-2769, ISBN: 978-960-474-089-5

[6] Pinca B. C., Tirian G.O, Research on increasing the lastigness of a rolling bridge, Journal of WSEAS transaction on applied and theoretical mechanics, ISSN 1991-8747, Issue 3, Volume 4, July 2009, pp.115-125.

[7] Pinca B. C., Optimization of a strength structure of a metallurgical aggregate, *PhD. Thesis, Politechnica University, Timişoara*, Romania, 2002.

[8] Venetsamos D.T., Magoula E.A.T, Provatidis C.G.- Performance- bsed optimization of the welded box of a single girder overhead traveling crane according to EC3 and EC1, 5th World Congress on Computational Mechanics, 5th European Congress on Computational Methods in Applied Science and Engineering, Venice, Italy, 2008.

[9] Zienkiewicz O.C, Taylor R. L., - *La method des elements finis*, AFNOR, Paris, 1991

[10] COSMOS User`s guide, Structural research analysis corporation, Santa Monica, 1999

[11]CTICM-EUROCODE NO.3- Calcul des structures en cier.Regles generales et regles pour les batiments, V.2-1991

[12] DIN 1800 (Teil 1) - Allegemeine Entwurfsgrrundlagen