

Finite Element Analysis of Doorframe Structure of Single Oblique Pole Type in Container Crane

X. F. Cheng^{1a}, F. Q. Wu^{2b}, G. Tang^{1c*}, X. Hu^{1d}

¹Logistics Engineering College, Shanghai Maritime University, 201306, Shanghai China

²Shanghai Institute of Special Equipment Inspection & Technical Research, 200062, Shanghai China

^adafeng1121@126.com, ^b81414458@qq.com, ^cgangtang@shmtu.edu.cn, ^dhuxiong@shmtu.edu.cn

Abstract. Compared with the composite type, the single oblique pole type has more advantages, such as simple structure, thrift steel and high safe overhead clearance. The finite element model of the single oblique pole type is established in nodes by ANSYS, and more details are considered when the model is simplified, such as the section of Girder and Boom, torque in Girder and Boom occurred by Machinery house and Trolley, density according to the way of simplification etc. The stress and deformation of ten observation points are compared and analyzed, when the trolley is in nine dangerous positions. Based on the result of analysis, six dangerous points are selected to provide reference for the detection and evaluation of container crane.

1. Introduction

Container crane is the most important equipment for container special port^[1,2]. Super-speed and large-scale are the future direction of container crane. The rated load under the spreader has increased gradually from 30.5t to 61.5t, and the maximum has reached 65t^[3]. With outreach and lifting height increasing, lifting speed is enhanced from 50 / 120m / min to 90 / 200m / min and so on. These constraints put forward higher requirements for the steel structure of container crane. Finding the weaknesses of the structure is significant for container crane's safety. Optimization and detection of these places are also equally important^[4]. This paper uses ANSYS software to analyze the common single oblique pole type. The stress and deformation of observation points are analyzed and compared when Trolley is at different dangerous positions. Six dangerous points are chosen by the results of the analysis finally.

2. The finite element model of the steel structure is established

2.1. Simplification and hypothesis of model

Container crane is complex generally^[5]. If all the details are taken into account, it is bound to increase the difficulty of modeling and the result is not necessarily well. Therefore, the container crane should be simplified reasonably and ideally. A finite element model is established, which is both beneficial for the calculation and can be accordance with the actual working condition^[6].



(1) The whole structure includes the Leg Landside(LS.) and Leg Waterside(WS.), Frame, Boom, Girder etc. As these parts are welded by the steel plate, so the beam element is used to model [7];

(2) As the Machinery house is fixed on the Girder by several legs, the Machinery house is simplified to the mass points on the Boom. Torque occurred by the Machinery house is also applied on the Boom. The trolley is simplified into four mass points. Torque occurred by the Trolley is considered at the same time;

(4) The Gantry Operating agencies, main Hoist agencies etc, are simplified to mass points imposed in corresponding locations;

(5) Ladders, Rails, Ribs, etc. have no effect on the whole structure, so the density of corresponding components are adjusted. These accessories are not constructed in the model;

(6) As the cross-section of the Boom and Girder are complex, a custom cross-section is chosen. The track is a part of Girder and Boom;

(7) The Gantry is simplified into four legs;

(8) The ear plates are simplified as beam connected Boom, Girder, Forestay and Backstay;

(9) When the Boom is level, Forestay is deemed as a tension and compression to withstand the two-pole, therefore LINK180 is used to model.

2.2. The element type of the model^[8]

(1) BEAM188: Frame, Boom, Girder and other connection systems. The Frame and the connection systems are used rectangular sections; the Boom and Girder use custom sections;

(2) PIPE288: Backstay, the support system between the Frame, Boom, Girder and Pylon;

(3) MASS21: The Machinery house and some other additional accessories;

(4) LINK180: Forestay.

2.3. Material properties of the model^[9,10]

(1) Mechanical properties: Steel structure use generally A709-50-1 steel, the mechanical properties of elastic modulus $E = 2.1\text{GPa}$, Poisson ratio $\mu = 0.3$;

(2) Material density: Frame, Girder and Boom have a large number of internal ribs, the external escalator, Forestay and Pylon have pins, shafts, baffles and other parts etc. These have been simplified as discussed upon. Densities of these accessories should be adjusted to balance the model weight and the actual weight.

2.4. Model Constraint and Condition

Constraint: In the ANSYS model, the Gantry are simplified into four support legs, so that the four bottom nodes' the x, y, z direction of displacement are constrained [6]. According to another papers, 9 dangerous positions (as shown in Table 1, Figure1) be selected where Trolley are on the Boom and Girder to analyze and calculate.

Table 1. Nine dangerous positions

NO.	Dangerous positions
P1	In the Backreach
P2	In the middle of the Tailstay and the Backreach
P3	In the Portal upper beam LS.
P4	In the middle of the Portal upper beam WS. and Portal upper beam LS.
P5	In the Portal upper beam WS.
P6	In the middle of Forestay's hinge and Boom's hinge
P7	In Forestay's hing
P8	In the middle of Forestay's hings

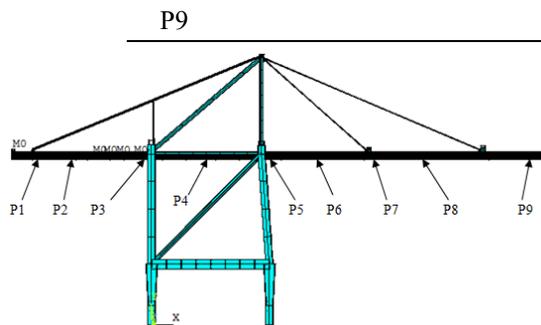


Figure1. Dangerous positions

Max-Outreach

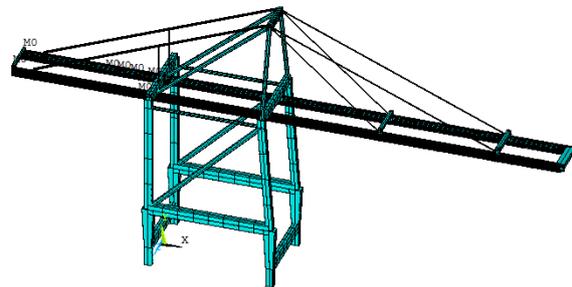


Figure2. Finite model

2.5. Model establishment

Traditional modeling is generally used bottom-up modeling or other software to draw a solid model and then import the ANSYS software. The finite element software analyzes nodes and elements ultimately, so at the beginning of modeling, nodes and elements are chose to establish model, without other objects^[7]. The procedure as follows: select the element - input material properties - create cross-section of beam, pipe and custom sections, link sectional area and mass elements - create node and element - couple the units - adjust the density - load calculation - impose constraints and loads^[11]. The established model is shown in Figure2. It should be defined that length unit mm, mass unit t, force unit N, density unit t / mm [^] 3, the elastic modulus unit MPa, the acceleration unit mm / sec [^] 2.

3. Load calculation

3.1 Main working load.

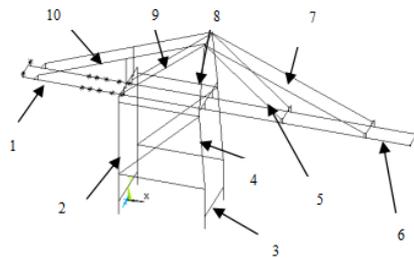
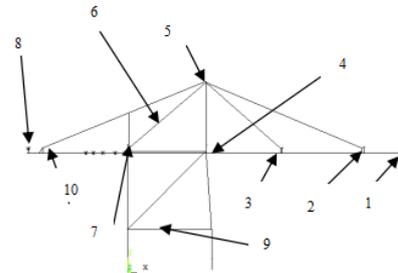
According to the literature, the loads acting on the crane are divided into four categories: normal load, incidental load, special load and other loads in the working state and the non-operating state^[12]. It is assumed that all the moving parts are at their most unfavorable positions, the main working load of the container crane is:

- (1) Container crane overall weight PC;
- (2) Container rated weight PQ, Trolley weight P1, Spreader weight P2;
- (3) The hoisting load caused by the working load: the fluctuating dynamic coefficient ψ resulting from the lifting of the working load is multiplied by the load due to the working load, where $\psi = 1 + \xi VL$ ^[13], $\xi=0.3$. The model comes from Zhen hua Port Machinery Company (ZPMC). Lifting speed (VL) is 1.12m / sec, this formula requires 1m / sec as maximum lifting speed. When the speed is greater than 1m / sec, the dynamic coefficient ψ is no longer increased;
- (4) The level of inertia induced by the movement of Trolley: The formula used for the simplified calculation is $F = 2ma$, where 2 is the load increase coefficient, m is the sum of the Trolley weight, the Spreader weight and the Container weight and a is the Trolley starting acceleration or braking deceleration^[13].

3.2 Load combination.

According to the working characteristics of the container crane, in this calculation, the metal structure of the container crane belongs to the statics characteristic, the load combination is:

- (1) The vertical load (y direction): $DPI = rc \times [\psi \times (P2 + PQ) + P1]$, where rc is the increase factor, according to the European crane design specification value is 1.2;
- (2) Horizontal load (x direction): $F=2ma$.

**Figure3.** Main parts**Figure4.** Deformation observation points

4. Finite element analysis calculation

4.1 Static analysis of steel structure of container crane

(1) It is necessary to analyze the stress of each part of the container crane in order to understand the stress of the various parts of the container crane^[14]. In this paper, 10 important parts of the container crane are selected and numbered, showed in Figure3. The stress analysis results of various components is shown in Table 2. In Figure5, it can be drawn that the Girder, Boom, Pylon, Leg WS. and the inside Backstay are the most sensitive stress places;

(2) Vertical deformation analysis: According to the literature, 10 deformation observation points are selected, and the distribution and number are shown in Figure4. The vertical deformation of the observation points are shown in Table 3, where the negative means downward. In Figure6 ,it can be drawn that the Girder and Boom end point, outside Forestay hinge point, inside Forestay hinge point, Backstay hinge point and the midpoint of the top beam, the six points are the most sensitive deformation points.

Table.2 Stress of main parts in different conditions of container crane steel structure

No.	Part Name	Stresses of various parts of the metal structure of the quayside bridge under load (MPa)								
		P1	P 2	P 3	P4	P 5	P6	P 7	P8	P 9
1	Girder	204.7	209.7	142.7	152.9	128.6	220.4	124.2	124.7	122.7
2	Leg LS.	102.9	95.1	64.8	47.1	40.8	29.8	19.9	15.4	24.8
3	Sill beam WS.	5.7	5.7	5.6	5.1	4.2	5.2	5.4	5.9	5.8
4	Leg WS.	50.9	53.8	59.3	69.2	91.6	89.7	90.8	91.4	105.7
5	Boom	46.7	46.7	46.9	47.1	46.7	222	53	112.1	150.4
6	Inside Forestay	52.9	53	53.3	53.7	52.7	51.8	55	77	135
7	Outside Forestay	57.1	56.8	56	55	57.5	92	123	118	51
8	Pylon WS.	81.5	77.7	74.6	75.3	77.2	87.1	101.7	114.7	129.3
9	Inside Backstay	69.5	73	70.1	63.1	69.9	79.9	88.1	98.4	114.2
10	Outside Backstay	115	92.9	58.7	52.6	57.6	59.9	59.1	59.2	59.8

Table.3 Deformation of transform observation points in different conditions

No.	Observation point	Deformation values of the deformation observation points (mm)								
		P1	P2	P3	P4	P5	P6	P7	P8	P9
1	Boom end	-60	-75.9	-107.4	-123.9	-129.7	-149.7	-174.7	-213.5	-407
2	Outside Forestay hinge point	-85.5	-85.5	-85.5	-85.5	-85.5	-85.5	-85.5	-85.5	-85.5
3	Inside Forestay hinge point	-17.2	-23.6	-35.8	-42.1	-46.3	-68.8	-87.8	-94.7	-95.3
4	Below Portal WS.	-10.3	-10.9	-13.2	-17.4	-14.2	-16.4	-17.8	-16.9	-19

5	Midpoint of the top beam	-12.2	-13.1	-14.6	-15.6	-16.4	-18.8	-21.2	-23.5	-26.3
6	Inside Forestay	-10.2	-10.7	-14.3	-16.9	-14.2	-13.7	-15	-16.1	-17
7	Portal LS.	-25.1	-23.4	-19.8	-16.4	-13	-9.2	-6.6	-3.0	2.7
8	Girder end	-151.3	-116.8	-62.7	-44.5	-45	-29.3	-11.6	9.2	38.9
9	Horizontal beam	-4.9	-4.8	-4.6	-4.4	-4.5	-4.5	-4.5	-4.4	-4.3
10	Outside Backstay hinge point	-161.7	-130.6	-71.1	-52	-54.5	-41.8	-26.1	-8	-17.8

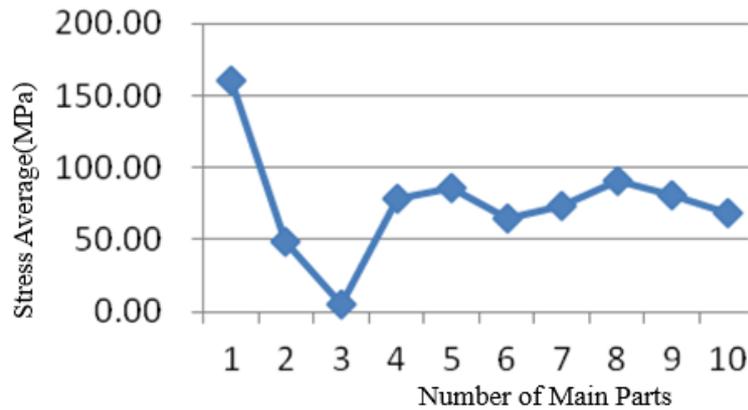


Figure5. The average stress of measuring parts

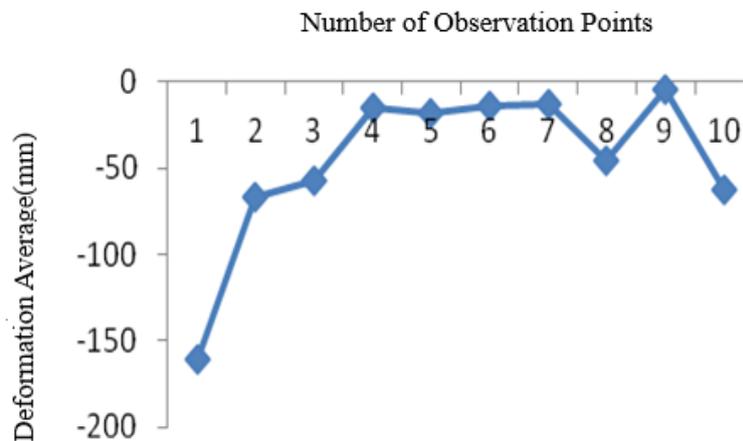


Figure6. The average deformation of observation points

5. Summary

In this paper, the finite element model of single oblique pole type in container crane is established by the ANSYS software, and the stress and deformation are analyzed for the important parts and observation points under different working conditions. We can conclude that:

- (1) The Girder, Boom, Pylon, Leg WS. and the inside Backstay are the most sensitive stress places;
- (2) The Girder and Boom end point, outside Forestay hinge point, inside Forestay hinge point, Backstay hinge point and the midpoint of the top beam, the six points are the most sensitive deformation points;
- (3) In the process of detecting and diagnosing the container crane, we can simply pay attention to these six deformation observation points and these five parts.

Acknowledgements

This work was supported by the National Natural Science Foundation of China (No. 31300783), China Postdoctoral Science Foundation (No.2014M561458), Doctoral Fund of the Ministry of Education

Jointly Funded Project (No.20123121120004), the Shanghai Maritime University Research Project (No.20130474), the Shanghai Top Academic Discipline Project- management science & engineering, and the high-tech research and development program of China (No. 2013A2041106)

References

- [1] Dong Dashan, Mei Xiao, Zheng Su. *Port Crane*.(Shanghai: Shanghai Jiao tong University Press), 2014pp19-35.
- [2] Yang Yuhua, Zhang Qing, Nie Feilong. *Development Trend of Container Automation Terminal*,(Shanghai: Chinese Journal of Construction Machinery),2015p571.
- [3] Zhu Yongjun. *Quayside Container Crane Structure's Parameterize Analysis and the Development of the Software System*, (Wuhan: Wuhan University of Technology),2014pp1-6.
- [4] Du Chao, Wang Jiaoe, Mo Huihui. *Research on the Spatial Pattern and Complexity of Chinese Container Shipping Network*,(Wuhan: Resources and Environment in the Yangtze Basin),2016.
- [5] Huang Xiaojie. *Research on Stress and Fatigue Life Analyze and Online Visualization Technology for Container Crane Component*,(Shanghai: Shang Hai Jiao Tong University),2014
- [6] Deng Guoping, Wu Zhijun, Qiu Huiqing. *Finite element analysis and comparison of the gantrycrane mechanism for the quayside container cranes*,(Shanghai:Chinese Journal of Construction Machinery), 2011pp43-47.
- [7] Cheng Feng. *Based on ANSYS shore crane parameterized simulation and fatigue analysis*,(Wuhan: Wuhan University of Technology), 2007pp1-48.
- [8] Li Guofang. *Optimization Design of Main Girder of Container Crane*,(Beijing: China Science and Technology Information),2014pp152-153.
- [9] Shang Yuejin. *Finite theory and ANSYS application guide*,(Beijing: Tsinghua University Press), 2005.
- [10] Dietrich Braess. *Finite Elements*.Beijing: World Book Inc,2012.
- [11] Li Jing. *Research on Load Spectrum and Fatigue Life of the Container Crane*,(Wuhan: Wuhan University of Technology),2014.
- [12] National Standard of the People's Republic of China GB / T 3811-2008 *Crane Design Code*.
- [13] Federation European Manufacture , *FEM 1.001: European Crane Design Specification*.
- [14] Ta Na, Rao Zhusi, Li Yiming. *Finite Element Modeling and Dynamic Response Analysis of Container Crane*,(Shanghai: Noise and Vibration Control),2008.