Applied Mechanics and Materials *Online: 2012-11-12*
ISSN: 1662-7482, Vols. 217-219, pp 1530-1534
doi:10.4028/www.scientific.net/AMM.217-219.1530

Simulation of the Wind Field of Gantry Cranes Based on FLUENT

Jingbo Hu[1,a], Da Chen[2,b], Shuqing Ding[1,a],Guangwei Qing [1,a], Hui Jin[2,c]

[1]Nanjing Special Equipment Inspection Institute, Nanjing, 210002, P.R.China

[2]School of Civil Engineering, Southeast University, Nanjing, 210096, P.R.China

[a]hubertert@163.com, [b]chenda.seu@live.com, [c]jinhui_seu@163.com

Keywords: Gantry crane, wind field, computational fluid dynamics (CFD), Fluent.

Abstract. The wind load on large cranes with great windward areas cannot be ignored. The simulation of wind field is to determine the accurate wind force acting on different parts of cranes and serves as the basis of the windproof design. Taking 300t shipbuliding gantry crane as the study object, this paper uses computational fluid dynamics, calculates the mechanical response of cranes by FLUENT, obtains the graphs of pressure, streamline, force distribution of cranes under different wind loads. Through the result analysis, dangerous force position is pointed out. And the ideas of structural methods to improve the windproof ability of cranes are also proposed.

Introduction

Nowadays mobile cranes are widely used in open places such as ports where sudden gusts are quite commen. When typhoons or sudden gusts come, cranes are easily damaged or collapesd due to its towering structure, large windward area, and high acting point of wind force. So the calculation of cranes' windproof ability and the design of windproof devices are indispensable. The simulation of wind field is to determine the accurate wind force acting on different parts of working cranes. Taking 300t shipbuliding gantry crane as the study object, this paper uses computational fluid dynamics, calculates the mechanical response of cranes by Fluent, provides the basis and ideas to windproof design of gantry cranes.

Description of the Theory and the Fluent Cade

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. The Fluent cade is a commerical software with good universality. It can be used not only for engineering design, but also for scientific research.

Computational Simulation of the Wind Field

Establishment of the Finite Element Model. Simulation model of the 300t shipbuliding gantry crane is established in the module of Workbench in ANSYS 12.1 with the scale being 1:1, as shown in Fig. 1. Structual details which influence wind force little are ignored, such as wheels, rails, small cars, weights and so on. Meanwhile, this model is just an outline of the crane without internal steel structure. Because only the wind field surrounding cranes is analysed by Fluent and on calculation focuses on the mechanical ability of cranes theirselves.

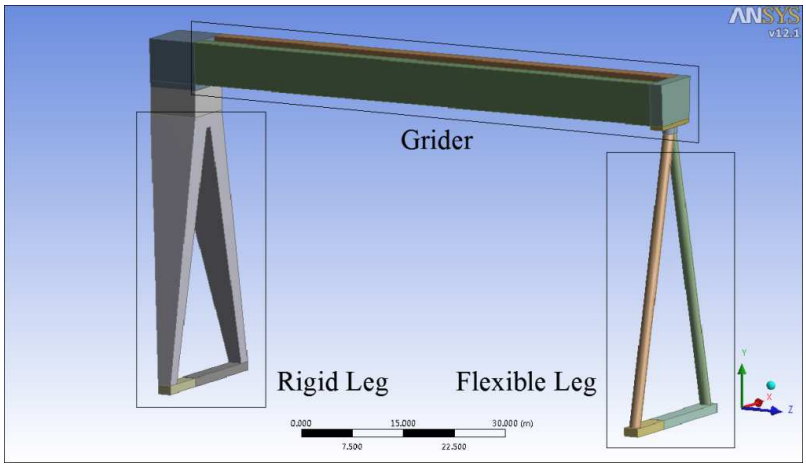


Figure 1 Simulation model of the 300t shipbuliding gantry crane

A rectangular flow field encircling the crane is set up to simulate the wind field (L: 132.400 m, W: 185.603 m, H:112.250 m) , as shown in Fig. 2. The distance between the windward side of crane grider and the air-inlet surface of wind field is 8 m displayed in Fig. 3. The reason for this size is to ensure the total wind force acting on cranes calculated here is close to the results from standards (GB/T 3811-2008) and the error is less than 4 % through verification.

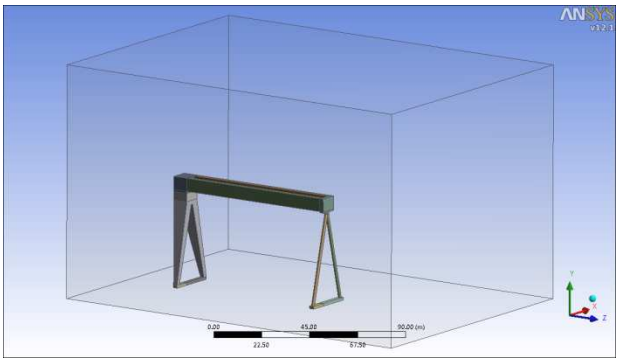


Figure 2 Finite element model of wind field

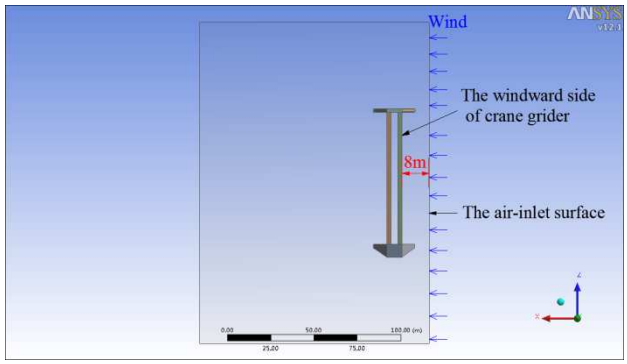
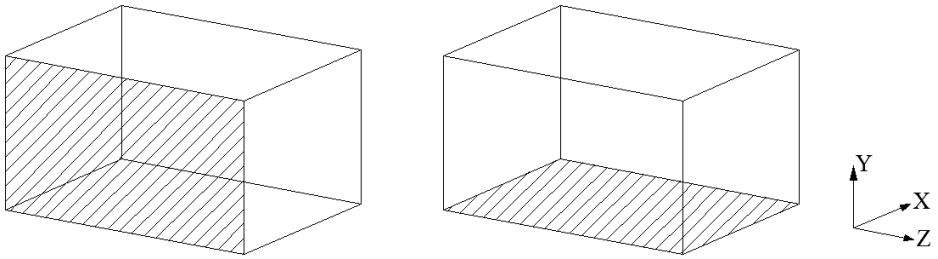


Figure 3 Top view of wind field

Related parameters and the mesh. The wind velocities rank due to the wind scales and according to standards, typical velocities are chosen corresponding to different wind scales among level 4~13. To approach the real, the wind velocity of working state is adopted which means 1.5 times of 10 min average wind velocity.

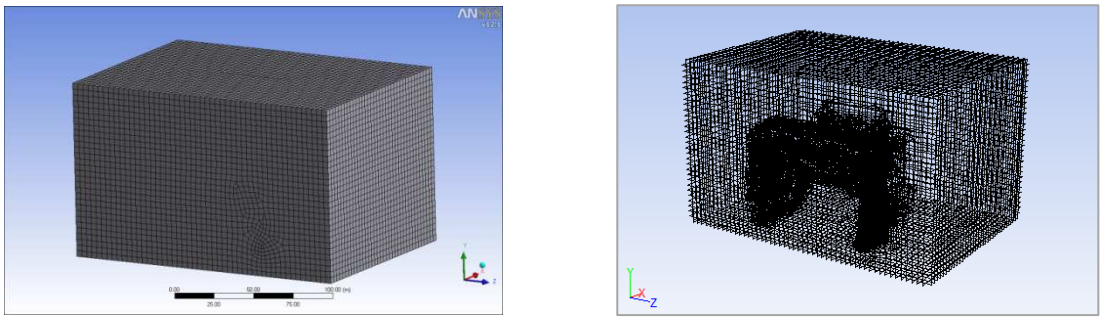
In Fluent, the wind field of cranes is defined as turbulent field and the calculation model is selected as K-epsilon (2 eqn) . This model aims for fully developed turbulence which is most widely used in engineering calculations. When setting boundary conditions, only one side of the field is assumed to be air-inlet , as shown in Fig. 4 (a) . And the type is velocity-inlet with all wind velocities on this side being the same. Synchronously the type of ground shown in Fig. 4 (b) is chosen as wall and other four sides are defined as type of pressure-outlet with gauge pressure to be 0 pa. This boundary conditions can handle problems which have the backflow on exit and calculations can reach convergent more easily.



(a) (b)

Figure 4 Position illustration

The Hex Dominant Method is used to mesh this model which generates hexahedrons on the surface and tetrahedrons in the internal. This method suits block geometries like the rectangular flow field here. And the mesh results for the surface is shown in Fig. 5 (a). To reach an accurate simulation, those nodes enclosing crane are refined as can be seen from Fig. 5 (b).



(a) (b)

Figure 5 Mesh Results

Results of Simulation

Under different wind loads, the pressure distributions and force distributions on cranes are almost the same regardless of the numerical value. The pressure distribution of the windward side in strong breeze (wind of level 6 scale) is diaplayed in Fig. 6.

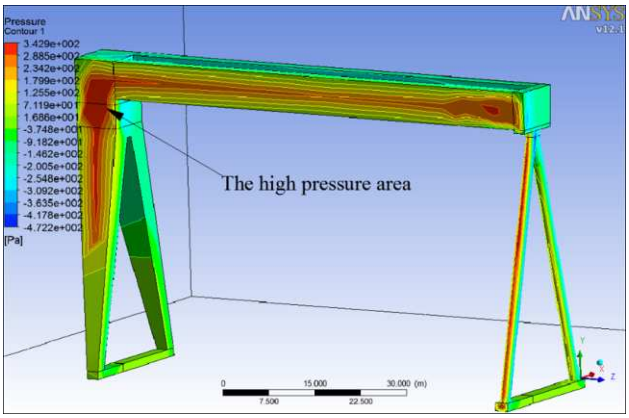
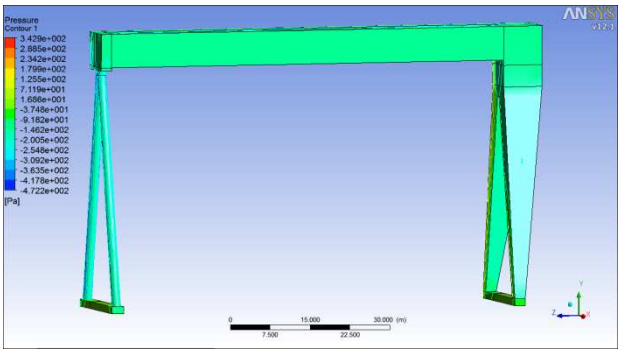
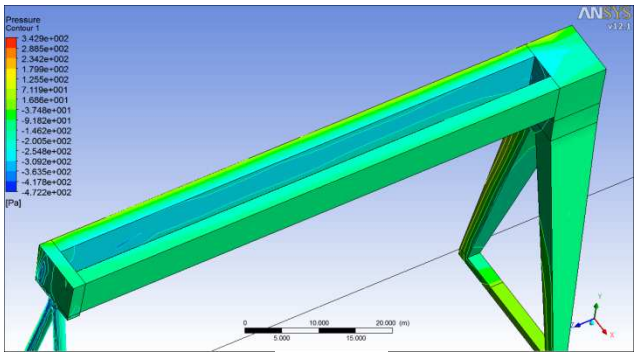


Figure 6 Pressure distribution of the windward side

With load of strong breeze, the high pressure areas of the windward side concentrate on the junction between the grider and rigid leg, as can be seen from Fig. 6. Negative pressure areas are formed on the lee side of crane and internal side of grider, as shown in Fig. 7 (a) and Fig. 7 (b).



(a)



(b)

Figure 7 Pressure distribution of the lee side of crane and internal side of grider

From the followind horizontal streamline chart of grider in strong breeze, massive negative pressure areas cause vortexes mostly emerging at the lee side of junction between the grider and rigid leg. And according to Fig. 6, high pressure areas appear at the other side of the junction. So the dangerous force position of gantry cranes under wind loads is on the junction between the grider and rigid leg. And changing the junction structure to minish the influence of high pressure and vortexes is a very effective way to *improve the windproof ability of gantry cranes.*

The force distribution of crane under the load of strong breeze is shown in Fig. 9. As can be seen, the main wind force acting on crane concentrates on the grider and rigid leg, While the force acting on flexible leg is almost scattered and numerically small.

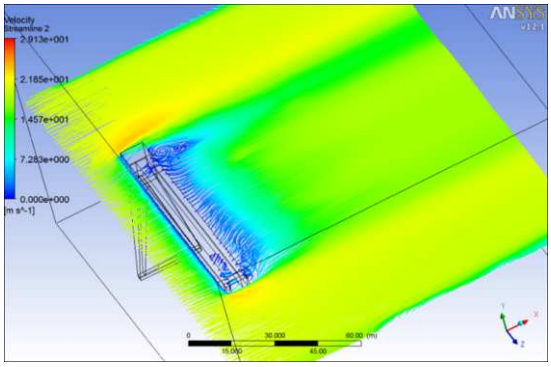


Figure 8 Horizontal streamline chart of grider

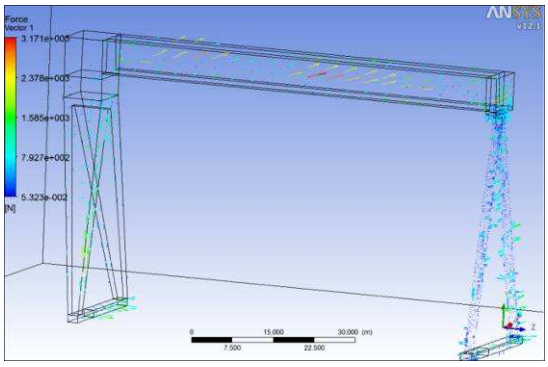


Figure 9 Force distribution of crane

With different wind scales, the total wind force acting on cranes can be summarized as followed table.

Table 1 Total wind force

Wind scales	4	6	8	9	10	12	13
Wind velocities [m/s]	8.25	19.95	28.35	33.15	38.4	48.9	58.95
Wind force [kN]	65.16	385.099	752.064	1024.85	1373.41	2217.29	3211.55

The curve shown in followind Fig. 10 can be achieved through datum in Table 1.

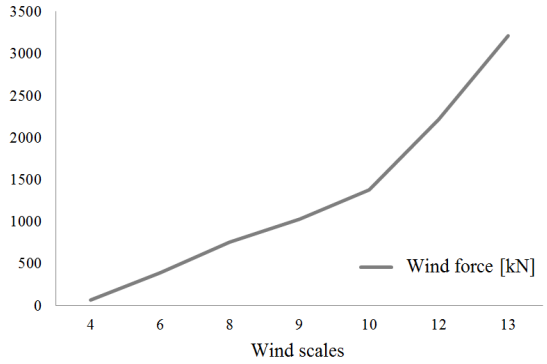


Figure 10 Wind force curve

As can be seen from the figure above, the wind force changes almost in a linear fashion between scales of level 4~10 and 10~13. The amplification of wind force is significantly exacerbated when the scale is more than level 10. So the windproof measure should be adjusted according to different wind scales and different wind velocities. Especially when a storm comes with scale being more than level 10, extraordinary measure must be taken in case of accidents.

Conclusions

To analyse the wind field of 300t shipbuliding gantry crane, this paper uses computational fluid dynamics, calculates the mechanical response of the crane by Fluent, obtains the graphs of pressure, streamline, force distribution of cranes under different wind loads. As the results shown, the dangerous force position of cranes under wind loads is on the junction between grider and rigid leg. And changing the junction structure to minish the influence of high pressure and vortexes is a very effective way to improve the windproof ability of cranes. And when a storm comes with scale being more than level 10, the amplification of wind force is significantly exacerbated. So extraordinary measure must be taken in case of accidents. All these analyses provide the basis and ideas to the windproof design of gantry cranes.

Acknowledgements

This work was supported by the science and technology project of General Administration of Quality Supervision, Inspection and Quarantine of the People's Republic of China (2009QK153).

References

[1] J.P. Wang: Computational Fluid Dynamics (CFD) and Engineering Application. Journal of Electrical Equipment. Vol. 5 (1994).

[2] Seong Wook Lee, Jae Joon Shim, Dong Seop Han, Geun Jo Han, Kwon Soon Lee: An Experimental Analysis of the Effect of Wind Load on the Stability of a Container Crane. Journal of Mechanical Science and Technology. Vol. 21 (2007), P. 448-454.

[3] F.J. Wang: Principle and Application of CFD software. Tsinghua University Press (2004).

[4] Information on http://en.wikipedia.org.