International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 6 Issue 8 Ver. II ǁ 2018 ǁ PP. 54-59

Numerical Simulation Of A New LNG Vaporizer Using Flue Gas To Impact Umbrella Rotor

Li Lei-Lei[1], He Fa-Jiang[1]
(1.Shanghai university of engineering science, Shanghai 201600,China)
Corresponding author: Li Lei-Lei

ABSTRACT: *A new LNG vaporizer using flue gas to impact umbrella rotor were designed in detail. Flow and heat transfer models of the LNG vaporizer for modeling combustion and species transportation were built. The models used species transport model to describe combustion of natural gas and species transportation, standard k-ε model to describe turbulent flow, flow and heat transfer of combustion were numerically simulated by SIMPLE algorithm. The effect of flue gas recirculation on velocity and temperature at the outlet of combustion chamber is analyzed and compared. Numerical simulation indicated when 50% of flue gas recirculation flow, high-temperature combustion flue gas and flue gas recirculation can be fully mixed in the conical nozzle area, and the water droplets are fully rolled up by the flue gas and filled with the vaporizer at a higher temperature and speed. The heat transfer of gasification is strengthened.*

KEY WORDS: *vaporizer；numerical simulation；conical nozzle; heat transfer*

Date of Submission: 15-11-2018 Date of acceptance: 29-11-2018

I.INTRODUCTION

LNG (Liquefied Natrual Gas) vaporizer is a heat exchanger which can be used for liquefied natural gas gasification. Therefore, the efficient utilization of LNG is one of the key research targets of LNG vaporizer. Common types of LNG gasifiers include water heating, air temperature, immersion, intermediate heat carrier, combustion heating, etc**[Error! Reference source not found.]**. These vaporizer have their own advantages and applicable occasions, but there are also problems of low efficiency such as icing and gasification. A new type of super open-shelf gasifier developed by Osaka Gas and Kobel Steel in Japan adopts inner and outer double-layer heat exchange tube structure, which can realize the step-by-step operation of LNG gasifier**Error! Reference source not found.**. Bi Mingshu et al. simulated the immersion LNG gasifier and discussed the influence of inclination angle of heat exchanger tube and Reynolds number of jet flow on the heating gasification process of LNG[2]. Huang Xinghua et al. proposed a numerical simulation method for fluid flow and heat transfer in shell-and-tube heat exchangers. The average fluid mass, momentum and energy conservation equations were solved, and the distribution of shell-side flow and heat transfer was obtained**Error! Reference source not found.**. All the above studies have promoted the development of LNG gasifier, but there are still some shortcomings such as poor heat transfer effect and low gasification rate. Therefore, further research and promotion of LNG gasifier are needed.

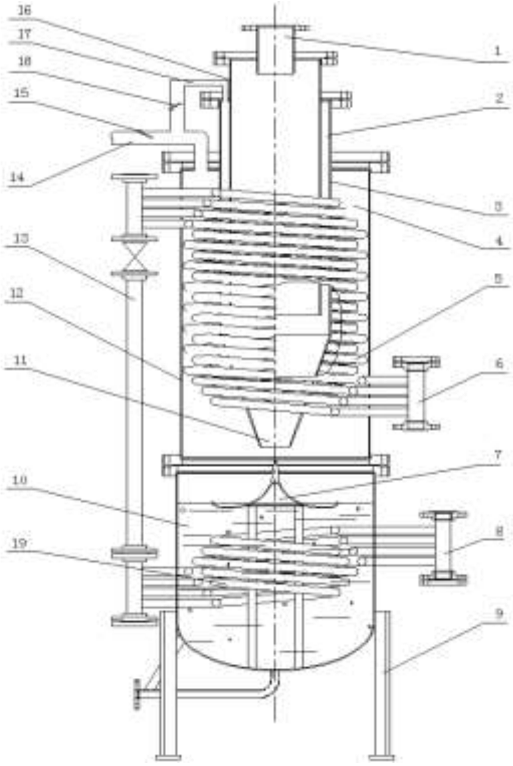
A peak shaving LNG vaporizer with high gas supply rate and high thermal efficiency is developed in this paper. The mathematical model of flue gas impinging swirling water and LNG gasifier combustor is established based on combustion, heat transfer and multi-phase flow theory. The theoretical study and numerical simulation of combustion, flow and heat transfer characteristics are carried out to obtain the circulating flue gas volume with the best impact effect.

1 Structure and working principle of LNG vaporizer

As shown in Fig. 1, high temperature flue gas is generated from combustion of fuel in the combustion chamber, and high temperature flue gas is ejected from the conical nozzle. The lower part of the spinning water is immersed in water, and the flue gas impacts the swirling water at high speed and cuts to the water surface. The annular passage of the gasifier is formed between the outer cylinder of the combustion chamber and the cylindrical shell. The high temperature flue gas enters the annular passage together with the water droplets, water mist and steam from the heated gasification under the pumping action of the induced draft fan. When flowing through the upper coil of the annular passage, the flue gas exchanges with the LNG flowing in the upper coil, and part of the flue gas passes through the annular passage. The chimney discharges into the atmosphere, the other part enters the circulating flue gas jacket, returns to the combustion chamber and mixes with the combustion flue gas, and sprays out from the nozzle together to strengthen the effect of flue gas impacting the water surface, and can make the temperature of flue gas entering the upper coil less than the ignition temperature of LNG, so as to ensure the safe operation of the equipment. The connecting pipe enters the lower

The date of receipt:2018-11-8

coil of the gasifier, and the lower coil is all immersed in water, which is used to heat the gasified natural gas to the required technological parameters, and then applied to industry.



1.burner interface 2. outer jacket 3. outer barrel 4. upload hot surface layout space 5. coil pipe 6. LNG inlet pipe box 7. Cyclone 8. natural gas outlet pipe box 9. bracket 10. Pool 11. conical flue gas nozzle 12. cylindrical shell 13. connecting pipe 14. flue gas pipeline 15. flue gas pipeline air regulator 16. inner cylinder 17. circulation flue 18. cycle flue throttle 19. lower coil

Fig. 1 the LNG gasifier using flue gas hitting the umbrella rotor

2 Model and numerical simulation of flow and heat transfer in gasifier combustor

In this chapter, numerical simulation of combustion, flow and heat transfer in the flow field and temperature field of combustion chamber is carried out. The influence of conical nozzle outlet flue gas on swirling water is calculated when different circulating flue gas volume mixes with combustor flue gas, so as to obtain the optimum outlet flue gas temperature and velocity.

2.1 Combustion chamber structure model

The combustion chamber structure of the LNG vaporizer is shown in Figure 2, and the combustion chamber is cylindrical. In order to facilitate the study of combustion, flow and heat transfer in the combustor, the temperature and flow fields in the combustor are numerically simulated, and the model can be simplified to a two-dimensional symmetric model.

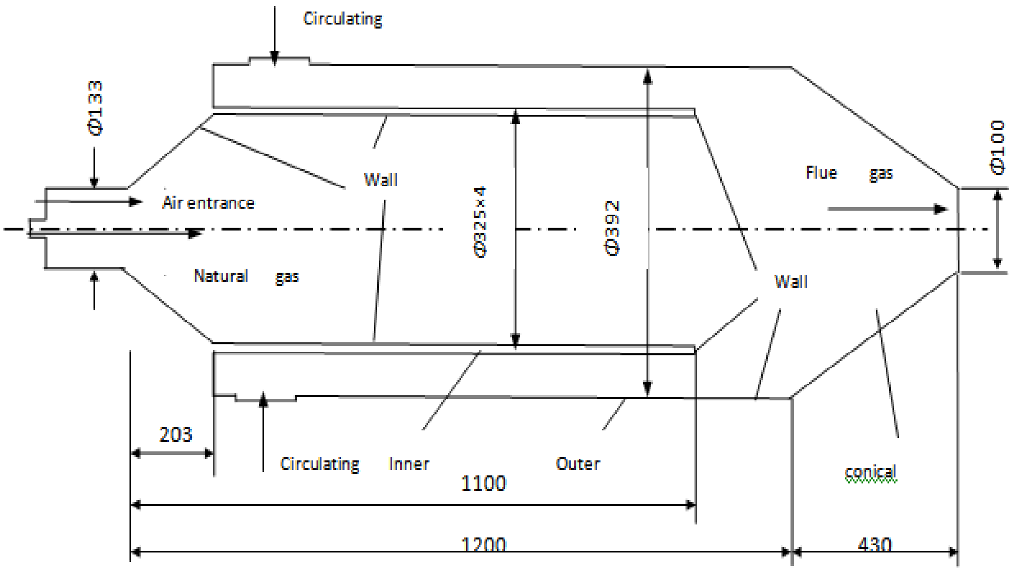


Fig.2 The structure model of combustion chamber

As shown in the figure, the natural gas nozzle is connected with the burner, the fuel natural gas enters the combustion chamber from the nozzle, and the air inlet is an annular inlet, which surrounds the natural gas nozzle. The air and natural gas enter the combustion chamber at different speeds and then burn together. In order to prevent the recirculation of the mixed air, the front end of the combustion chamber is designed as a conical inlet. The two sides of the combustion chamber outer cylinder are designed with a circulating flue gas inlet. The low-temperature circulating flue gas flows through the circulating flue gas jacket between the inner and outer cylinders and is mixed with the high-temperature flue gas generated by combustion at the conical nozzle.

2.2 computational model

In this paper, a general finite ratio model is used to simulate the component transport and combustion of natural gas in combustor. The eddy dissipation model is used to simulate the turbulent chemical interaction, and the standard k-e model is used to simulate the turbulent flow of fluids[Error! Reference source not found.].

2.3 Numerical simulation and analysis of combustion chamber

The structure model of combustor shown in Fig. 2 is simulated with 50% circulating flue gas. The velocity vector and temperature distribution of fluid flow in combustor are obtained.

2.3.1 Flow field and temperature field distribution

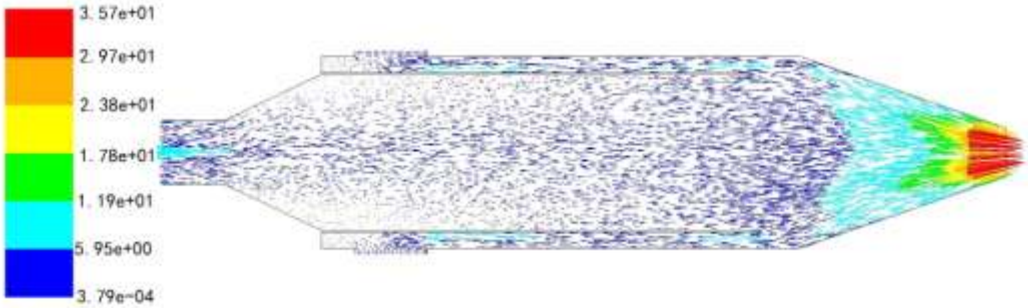


Fig.3 Velocity vector diagram of fluid flow in combustion chamber (unit:m/s)

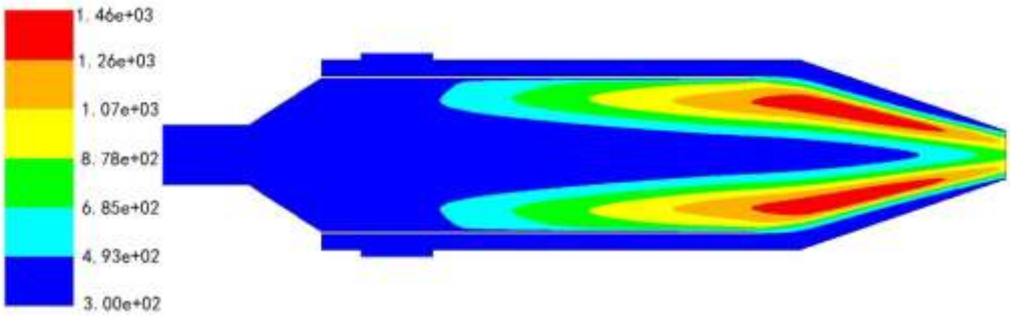


Fig.4 Temperature field distribution in combustion chamber (unit: K)

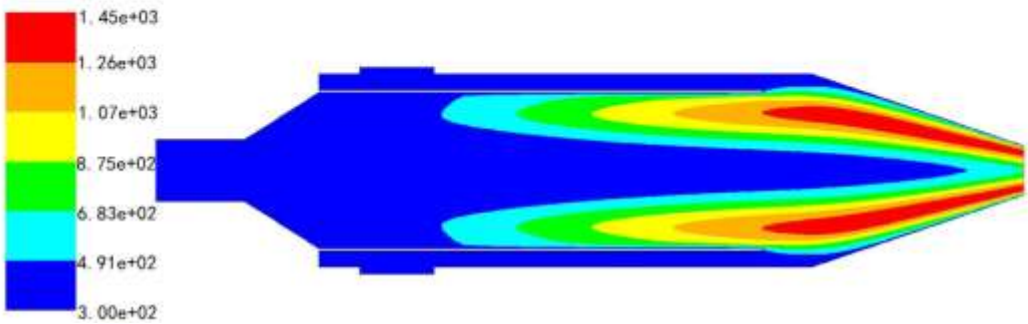
From Fig. 3, it can be seen that the gas just entering the combustion chamber is jet-like and flows at a high speed along the central line. According to the design of the burner nozzle, air is injected around the gas and is gradually absorbed by the gas jet, which mixes with the high-speed gas and combusts rapidly to form smoke. The smoke flows downward along the combustion chamber tube.

From Fig. 4, it can be seen that in the process of gas self-injection from top to bottom in the combustion chamber, the gas jet is continuously entrained and mixed with the surrounding air, so the gas jet becomes narrower and narrower in the shape of a sword. The air flow outside the blade-shaped blade burns in the air and warms up rapidly. The maximum temperature in the combustion chamber can reach 1000-1200K. The circulating flue gas with a temperature of 330K is injected downward from the annular jacket between the inner and outer cylinders of the combustor to effectively cool and protect the inner cylinder, and mixes with the combustion flue gas at the conical nozzle. After mixing, the flue gas temperature is about 920K, and there is always a layer of low temperature air flow close to the wall of the nozzle to protect the nozzle from burning.

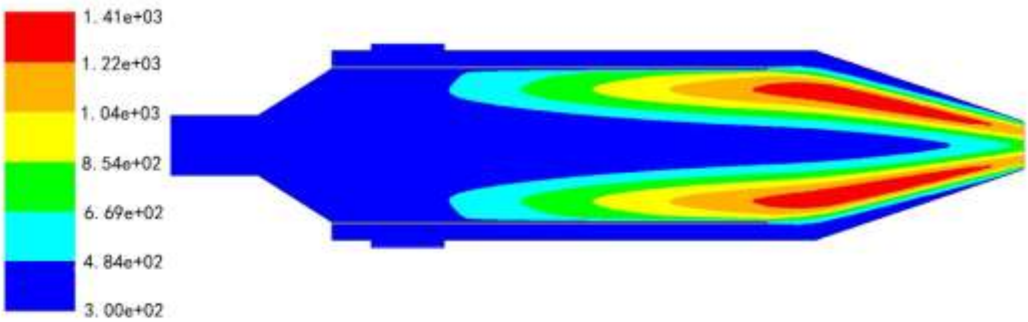
2.3.2 Effect of circulating flue gas volume on combustion chamber flow and temperature field

The temperature of circulating flue gas is basically unchanged, so the flow field and temperature field in combustor are mainly affected by the amount of circulating flue gas. In this paper, the amount of circulating flue gas is calculated to be 20%, 35%, 50%, 65% and 80% respectively.

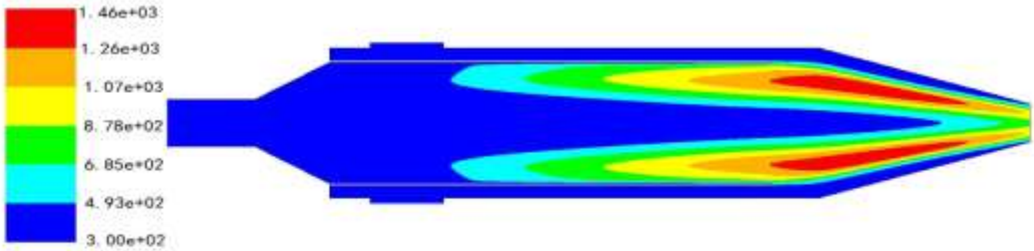
The distribution of temperature field in combustion chamber under different circulating flue gas volume is shown in Figure 5.



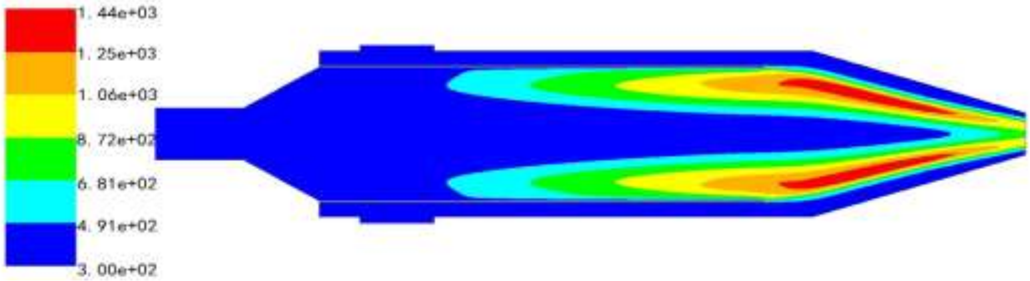
(a) 20% circulating flue gas volume



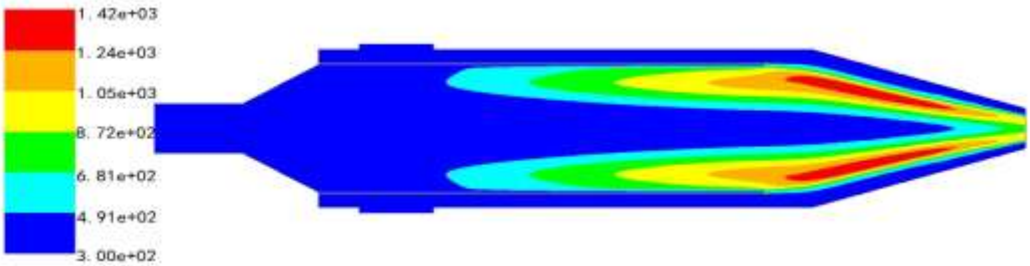
(b) 35% circulating flue gas volume



(c) 50% circulating flue gas volume



(d) 65% circulating flue gas volume



(e) 80% circulating flue gas volume

Fig.5 Temperature distribution of combustion chamber in different flue gas recirculation (unit: K)

As shown in Figure 5 (a), the flue gas quantity is 25%, it can better cooling the combustion chamber in the cylinder, but through the conical nozzle, high temperature gas entrainment is rapidly mixing nozzle wall is easy to be burned. In addition, due to a small amount of recycled flue gas, the mixed flue gas injection speed is low, can not effectively impact on water jet, water droplets, mist, water vapor entrainment effect is poor, can not achieve the purpose of cooling wet flue gas. The mixed outlet flue gas temperature and 1010K is higher than that of natural gas ignition temperature (about 923K), the security can not be guaranteed. When the cycle of flue gas volume rose to 35% of the amount of combustion flue gas, such as shown in Figure 5 (b), the phenomenon has improved, but still cannot reach the ideal requirements.

As shown in Fig. 5 (d) and Fig. 5 (e), when the circulating flue gas volume is larger, reaching 65% and 80% of the combustion flue gas volume, the mixture of high temperature gas and circulating flue gas is slower, and the non-uniform mixed flue gas needs to continue to mix after it sprays out from the nozzle. In addition, there is a thick layer of low temperature circulating flue gas on the wall of the nozzle, which is not conducive to the release of mixed flue gas to the water pool. Heat, and the weighted average temperature at the exit of mixed flue gas is also low. Fig. 5 (d) is 905K at 65% of the circulating flue gas volume, while Fig. 5 (e) is lower at 867K at 80% of the circulating flue gas volume, which is not conducive to the subsequent heat transfer of flue gas impacting cyclone.

As shown in Fig. 5 (c), when the circulating flue gas volume is 50%, it can be fully mixed with the combustion flue gas and ejected from the conical nozzle at a higher temperature and speed. It can not only effectively roll up water droplets, mists and vapors to achieve the effect of cooling with moisture, but also avoid the effect of low average temperature at the outlet on heat transfer. And there is always a layer of cryogenic air flow close to the nozzle wall to protect the nozzles from being burned.

Based on the analysis and comparison of the temperature distribution of combustion chamber calculated by simulation when the circulating flue gas volume is 20%, 35%, 50%, 65% and 80%, this paper considers that when the circulating flue gas volume is about 50%, the high temperature flue gas and circulating flue gas generated by combustion can be mixed in the conical flue gas nozzle, and the wall of the nozzle can be well protected. The average temperature of flue gas at nozzle outlet is 921K, and the average velocity of flue gas at nozzle outlet is 30.3m/s. At this time, the temperature and velocity of mixed flue gas at nozzle outlet of combustion chamber are more reasonable. The humid air flow is formed when swirling water is impacted by air flow, the proper temperature difference between mixed air flow and coil heat transfer is maintained, and the safety of LNG heating and gasification device is ensured. All operations are favorable.

II.CONCLUSION

In this paper, a novel and efficient LNG heating and gasification device has been developed, and a jacketed combustor structure has been designed. The flue gas circulation mode has been adopted to increase the flue gas flow rate of the conical nozzle outlet of the combustor impacting the swirler, increase the impact intensity and reduce the flue gas temperature. The number of fuel combustion and flow in the simulated combustor has been established. The general finite ratio model is used to describe the combustion and component transport of natural gas, and the standard k-e model is used to describe the turbulent flow of fluids**Error! Reference source not found.**. The effects of different circulating flue gases on flue gas injection were numerically simulated, and the results showed that when the circulating flue gases were too small, the injection speed of mixed flue gases was too low, and water droplets, mists and vapors could not be rolled up after impact on the water surface of the swirler, thus the effect of humidity and cooling of flue gas could not be achieved; on the contrary, when the circulating flue gases were too large, the mixing would occur. The low temperature of flue gas affects the heat release of the mixing flue gas to the pool. The calculation shows that when the circulating flue gas volume is 50% of the combustion flue gas volume, the effect of flue gas impacting cyclone is the best.

REFERENCE

[1]. He Fajiang, Cao Weiwu, Yan Ping. Design and experimental study of flue gas impact cyclone LNG gasifier [J]. Journal of Shanghai University of Technology, 2012, 34 (03): 298-302.
[2]. He Fajiang, Cao Weiwu, Kuang Jianghong. Numerical simulation of two-phase flow and heat transfer in umbrella-shaped gasifier [J]. Journal of Power Engineering, 2013, 33 (03): 199-204+217.
[3]. Chang Ling. Experimental investigation of flow and heat transfer characteristics of finned tube heat exchangers [D]. University of Science and Technology Liaoning, 2012.
[4]. Yu Guo Jie LNG immersive combustion gasifier numerical simulation [D]. Dalian University of Technology, 2009.
[5]. Yan Ping, Cao Weiwu, Qian Shangyuan, Liu Weijun, Yang Liyun. Research and design of a new type of flue gas self-swing wet LNG gasification unit [J]. Natural gas industry, 2011, 31 (01): 86-89+117.
[6]. Wang Jun. Flow and heat transfer analysis of hydrostatic pump based on computational fluid dynamics [D]. Huazhong University of Science and Technology, 2009.
[7]. ANSYS FLUENT Fluid Analysis and Engineering Case Matching Video Course [M]. Beijing: Electronic Industry Publishing House. 2015:195-208.
[8]. Li Pengfei, Xu Minyi, Wang Feifei. Familiar with CFD engineering simulation and case practice FLUENT GAMBIT ICEM CFD Tecplot [M]. Beijing: People's Posts and Telecommunications Publishing House. 2011:67-75.
[9]. Dou Xinghua LNG numerical simulation of flow and heat transfer process of immersive vaporizer [D]. Dalian University of Technology, 2007.
[10]. Yang Meng. The simulation of two-phase flow in computational fluid dynamics and the study of two-phase flow model [D]. Tianjin University, 2005.

Author's brief introduction: Li Leilei (1989-), male, Handan, Hebei Province, is a master's degree student, mainly engaged in the application of liquefied natural gas.（Kevin_lill@163.com）.

Author's basic situation table

Full name	Li Leilei	Sex	male	Birthplace or place of birth	Handan, Hebei
Nationality / nationality	Chinese	Date of birth	1989,10	Title	Postgraduate student
University one is graduated from	Shanghai University of Engineering Science		Research direction	Applied research of liquefied natural gas	
Speciality / degree	Energy equipment and process control			E－mail	Kevin_lill@163.com
Work unit	Shanghai University of Engineering Science			post	Student
Contact number	15117948711				
Zip code - Address	201620 Room 4122, building four, Shanghai University of Engineering Science, 333 Longteng Road, Songjiang District, Shanghai.				

Li Lei-Lei "Numerical Simulation Of A New LNG Vaporizer Using Flue Gas To Impact Umbrella Rotor " International Journal of Research in Engineering and Science (IJRES), vol. 06, no. 08, 2018, pp. 54-59