Research and Application of Numerical Calculation methods in SCR DeNO$_x$ Reactor & Duct Design

CHEN Haifeng
(Shanghai LongKing Environmental Projection Co. Ltd, Shanghai, PR China. E-mail: chenhaifeng@slep.sina.net)

Abstract: SCR reactor& duct design is the key part for a SCR DeNO$_x$ project to insure the best gas flow and the most reasonable structure. In this paper, gas CFD simulation and structure FEA simulation are used for SCR reactor& duct design based on CAE technology. According to this analysis result, position, dimension and joint of all structure components will be ascertained. These were applied in DeNO$_x$ project for Fujian Huadian Kemen Power Plant 2×600 MW units (stage II).

Keywords: SCR DeNO$_x$, CFD model, FEA model, Reactor& Ducts, Flow distribution

1 INTRODUCTION

SCR DeNO$_x$ projects of coal fired power plant are growing rapid in China at present, which the core techniques are all from developed countries. For most SCR DeNO$_x$ projects all over the world, SCR DeNO$_x$ systems are equipped between boiler ECO and APH. This kind of design is the best for the system operation, but also exerts influence on boiler, APH and dust catcher. So the reasonable designing of SCR DeNO$_x$ system became very significant.

The designing of reactor& ducts is the key part for SCR DeNO$_x$ project execution, which should assure the best flow and the best structure. During SCR DeNO$_x$ system operation, the gas-gas reaction is happened on the face of the solids (catalysts) under the condition of best gas flow. In general, it is difficult for using a simple method to get precise results for the detail design. Currently the best way for reactor& ducts design is numerical simulation including FEA (Finite Element Analysis) and CFD (Computational Fluid Dynamics) modeling.

2 CASE AND MODEL

This paper selects the SCR DeNO$_x$ project of Fujian Huadian Kemen Power Plant 2×600 MW units (stage II) as the case for research, which the DeNO$_x$ efficiency is 80%. Anhydrous ammonia is adopted as the absorbent for DeNO$_x$ system. The main parts of reaction include reactors, inlet& outlet ducts, ammonia injection& mixing system, guide vane devices etc. All these parts compose the whole model for numerical simulation. According to the react condition of catalysts, the main parameters for this SCR reactor& ducts system design are shown in Table 1.

The whole reactor& ducts system is initiated at the economizer outlet and is terminated at the air preheater inlet. It supported by steel structures at the location of 22 m–55 m. The area of this project is located at a coastal county where typhoon comes frequently. The fundamental wind pressure is 0.85 kPa with type A condition. The corresponding Seismic Basic Intensity is 6 degree. The outline of the whole reactor & ducts system including internal structure is illustrated in Fig. 1.

Table 1 The main parameters for SCR reactor& ducts system design

Parameter	Value
Flow (m^3/h), BMCR	4,540,723
Dimension of reactor (m)	15.6*10.8
Design Pressure (kPa)	±8.7
Design Temperature (℃)	371 (max=450)
Pressure drop (Pa)	800
Dust content (g/Nm^3)	22
Flue gas maldistribution, %	15
NH_3/NO_x maldistribution, %	5
Temperature distribution, K	±10

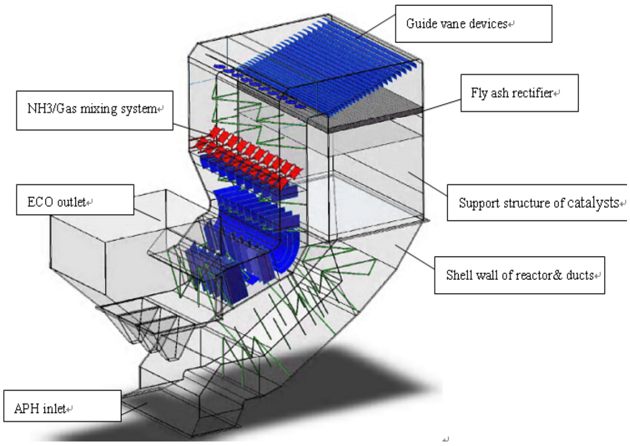

Fig.1 3D drawing of SCR reactor& ducts system

3 NUMERICAL SIMULATION BASED ON FEA AND CFD

In this paper, two kinds of CAE software are taken to do the numerical simulation, Ansys8.0 for structure FEA modeling and Fluent6 for CFD flow modeling. The modeling investigation constitutes the background for the detail design of all the main internal devices and structure members. According to the result of simulation and analysis, the position, quantity, dimension and joint type of all the main parts will be determined.

3.1 Finite Element Model for Structure Analysis

The material of the whole reactor& ducts structure with internal devices is carbon steel. The main structure type is thin shell structure with rib or frame steel structure. The main loads inside the structure are as follows: weight of catalyst (total 500 t for each reactor), gas pressure, deadweight etc. While the main loads outside are the wind, thermal displacement, and all kinds of process equipment like insulation, instrument etc.

The finite element analysis for structure is composed of the following parts.

3.1.1 Geometry Modeling

On the basis of general flow requirements, a preliminary structure outline with internal devices can be established in advanced. The elementary information of the main structure like layout, dimension and gross mass will be determinate. In Kemen project, the thickness of all the shell wall of reactor and ducts are chosen as 6mm, and the outside ribs, support beams are using multifarious of channel steel or H steel, as shown in Fig. 2.

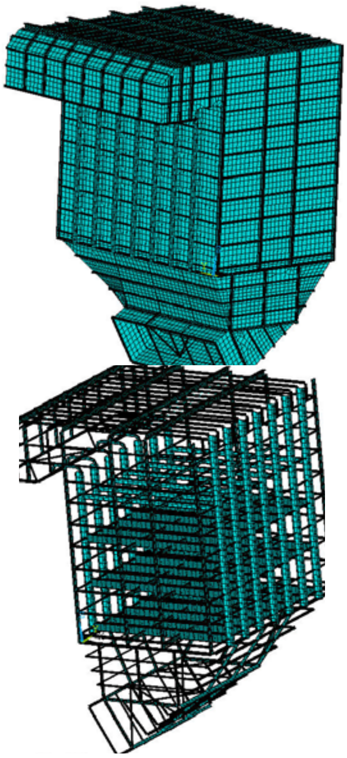

Fig. 2 Structure outline of reactor and duct

3.1.2 Finite Element Model and Analysis

In terms of the above geometry modeling, element type Shell 93 with 8-nodes (each node has six degrees of freedom) to be selected for reactor and duct wall, element type Beam189 which is suitable for analyzing slender to moderately stubby/thick beam structures to be selected for all the ribs and space frames. After the model is meshed, the computational nodes of model can be counted of approximately 250 thousand. The finite element analysis consists of mode analysis and static analysis.

(1) Mode analysis

The purpose of mode analysis is to show the structure dynamical characteristic including natural frequency of vibration and structure stiffness distribution. The theory of calculation program is Block Lanczos method which is especially powerful when searching for Eigen frequencies in a given part of the Eigen value spectrum of a given system. In this paper, 20 steps of mode types are solved. From the animation result, it can be seen that the stiffness of this kind of structure is very fine.

(2) Static analysis in different load cases

According to the Chinese load code for structure design GB50017-2003, five main load cases are listed as shown below.

1	wind□dead load	During installation
2	Earthquake□live load□dead load	Positive pressure
3	Earthquake□live load□dead load	Negative pressure
4	wind□live load□dead load	Positive pressure
5	wind□live load□dead load	Negative pressure

3.1.3 Results Discussion

From above calculation and analysis, it can be observed clearly from the result output, that the structure members' stress distribution, deformation shape and boundary reaction are all well controlled within requirement. See Fig. 3.

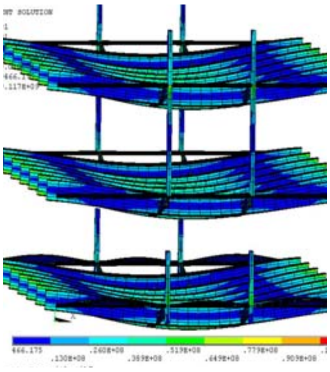

Fig. 3 Results of FEA Modeling

(1) Most of reactor and duct walls are controlled by material intensity and deformation value. The optimization for structure can carried via changing the distance of the outside ribs.

(2) Due to the high temperature during operation, the support gears slide obviously. Then the stress concentration can be found easily. These parts should be reinforced during detail design.

(3) The stiffness of joints for catalyst support structures shall be designed reasonably for the reason that it is disadvantaged for the whole structure when the stiffness is too high or too low.

(4) The internal truss works are mostly pressed members under the condition of negative gas pressure. Therefore the dimension of the truss shall be determined by detailed structure calculation, and the flow optimization shall be considered as well.

(5) In order to keep a better flow condition for DeNO$_x$ reaction, many internal truss works for wide span structure of reactor& duct system are removed. As a result that the values of structure stress and deformation are very high.

(6) The gross mass of the steel used for building reactor and duct can be optimized in evidence by the method of FEA simulation in detail.

3.2 CFD Simulation for Gas Flow

For DeNO$_x$ project, the standards of reaction conditions and environmental indexes are very high, such as flow distribution, temperature distribution, ammonia concentration distribution; pressure drop and ammonia slip etc. The objective of the CFD flow study is to design, test and optimize flow-conditioning devices, as appropriate, to guide the gas flow through the duct-work and DeNO$_x$ reactors of the plant, with the aim of obtaining a homogenous gas velocity profile in front of the first catalyst layer as well as in front of the ammonia injection grid. The process to do CFD simulation work is similar to FEA simulation mentioned above, such like geometry modeling, mesh grid, assumption for input. For KenMen final project, the flow in the full scale SCR unit has been solved at 100 % load by application of steady state, thermal, incompressible conditions with the k-ε turbulence model. Ammonia is injected into the flow through each pipe in the ammonia injection grid. Velocities in the transverse directions are limited by large resistances in these directions. The model is initiated at the ECO outlet and is terminated at the APH inlet. The computational model consists of approximately 2 million computational cells.

3.2.1 Flow Conditioning Devices

(1) Ammonia injection and mixing devices. These devices compose the core part for NH_3-Gas mixing harmoniously. In this project, tens of injection pipes and star-shaped mixers are arranged in two rows located at the inlet ducts, as shown in Fig. 1. The star mixers' angle, dimension, quantity and arrangement shall be determined during flow model simulation.

(2) Guide vanes. Guide vanes have been designed to optimize the flow through the unit with respect to pressure loss and to facilitating dust sweeping in critical areas. The locations of guide vanes are shown in Fig. 1. All guide vanes' dimension, quantity and position shall be determined during flow model simulation.

(3) Fly ash rectifier. The fly ash rectifier rectifies the fly ash in a vertical downward direction in the top of the SCR reactor before the fly ash reaches the first catalyst layer. This helps to minimize fly ash deposits and erosion of the catalyst.

3.2.2 Results Discussion

(1) System pressure loss

System pressure loss has been calculated for the full scale plant at design load. Based on the reference dynamic pressures in the AIG duct, the pressure loss coefficient of the prototype has been calculated. The total system pressure loss between stations 1 and 7 indicated on Fig. 4 as well as between economizer and air preheater is given in Table 2.

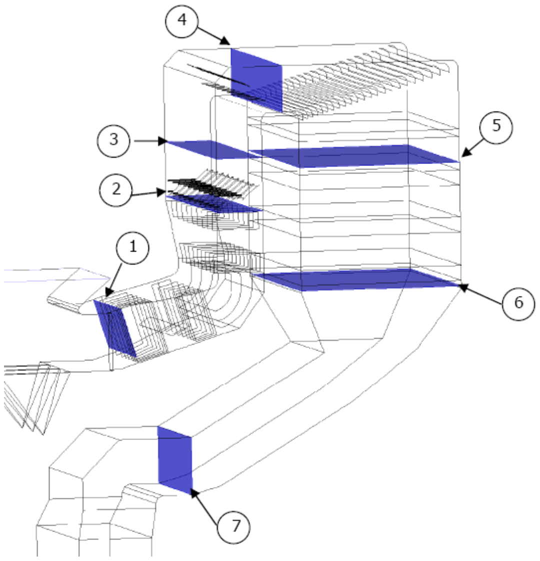

Fig. 4 Stations of pressure loss measuring

Table 2 Breakdown of pressure losses

Duct Segment	Loss coefficient	Prototype Pressure Loss
	[ ]	[Pa]
ΔP_{1-2}	1.87	87
ΔP_{2-3}	2.90	135
ΔP_{3-4}	0.88	41
ΔP_{4-5}	0.45	21
ΔP_{5-6}	5.97	278
ΔP_{6-7}	0.88	41

(2) Flow Distributions

Velocity distribution is an important index to show a flow condition. The fluid flow can be investigated by looking at the velocity distribution at selected 10 cross sections of the ducts. As shown in Fig. 5.

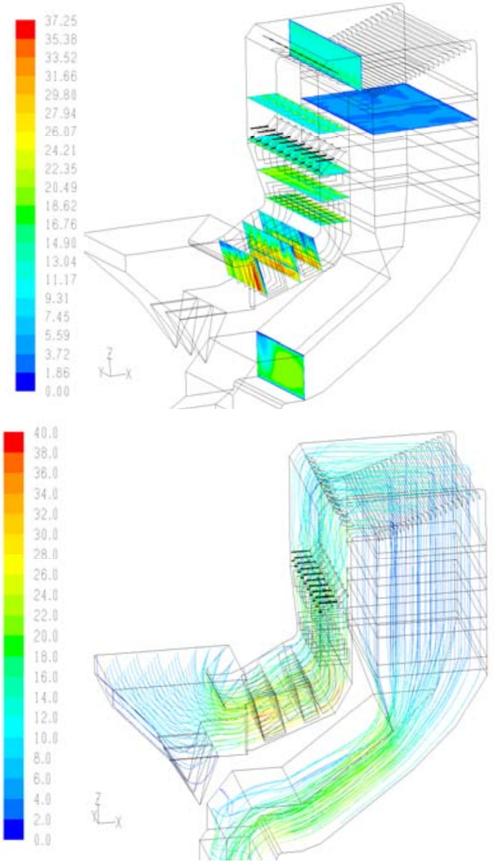

Fig. 5 Flow distribution and ath line of the whole system

Upstream the ammonia injection mixer and upstream the 1st catalyst layer are the most important two stations of the whole system for flow study in DeNO$_x$ plant. The flow distribution upstream the integrated ammonia injection and star mixer system is displayed in Table 3. A path line plot with path lines at the same location is shown in Fig. 6.

The final velocity distribution displayed in Fig. 7, obtained at the reference plane 0.5 m above the catalyst inlet face, has a standard deviation of 13% and a maximum value of 37% above the mean value. The standard deviation fulfils the stated criteria, while the maximum values are outside the stated criteria. However the remaining optimisation through adjustment of homogenisers can with advantage be carried out by model scale tests.

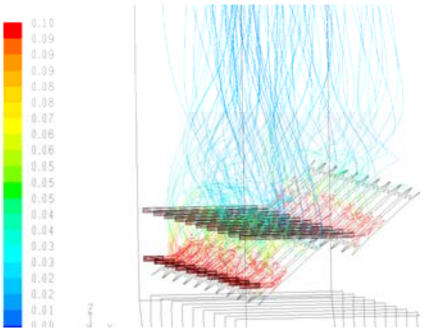

Fig. 6 Path line plot at the AIG

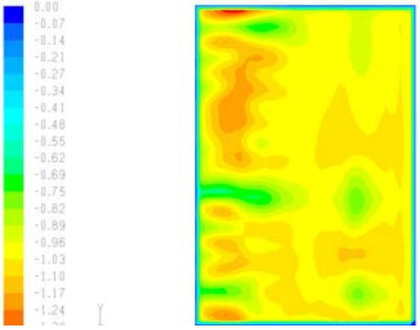

Fig. 7 Velocity distribution upstream the 1st catalyst layer

(3) Ammonia Concentration Distribution

The distribution of concentration of ammonia has been calculated at a plane 0.5 m above the inlet to the 1st catalyst layer. The result is found in Fig. 8. The standard deviation of the ammonia concentration distribution at the reference plane above the upper catalyst layer calculated to 4.7%.This value may be precisely predicted for inspection during operation.

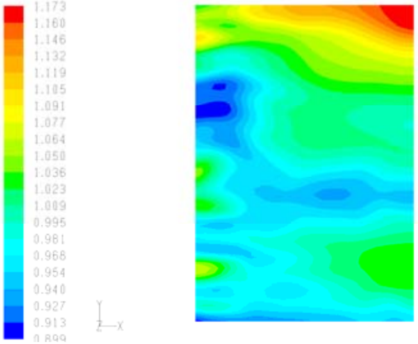

Fig. 8 Ammonia concentration distribution upstream the 1st catalyst layer

Table 3 Flow distribution at plane below ammonia injection mixer

Calculated Flow Distribution Range of Total Flow in Sections and Rows							
		Section 1	Section 2	Section 3	Section 4	Section 5	Row totals
3.1 m	Row 1	0.046	0.047	0.050	0.051	0.051	0.25
	Row 2	0.053	0.055	0.055	0.049	0.046	0.26
	Row 3	0.047	0.047	0.049	0.044	0.044	0.23
	Row 4	0.057	0.056	0.058	0.050	0.046	0.27
	Section totals	0.20	0.21	0.21	0.19	0.19	

15.6 meters

4 KEY POINTS FOR ANALYSIS AND DESIGN

From above discussion, several aspects for further research and investigation shall be considered as follows.

(1) The relationship between structure capability and flow condition

In respect that all the members of SCR reactor and duct structure are the boundary condition for flow analysis, the structure and the flow can infect each other. As a result that the best flow devices for the best flow usually can not be easy to carry out during structure design or construction. So it is necessary to adjust the FEA model or CFD model time and again during simulation analysis. The final objective is to keep a reasonable balance among structure safety, flow condition and the cost.

(2) Scale model and test

It is quit necessary to carry out sale model and test before the conclusion of simulation analysis to be put into practice, especially under the condition of complex structure, bad flow and without rich experience. With regard to Kemen project, the model construction, flow investigation and optimisation for a 1:10 scale model of the DeNO$_x$ extension of one 50% DeNO$_x$ reactor has been carried out in order to test and optimize flow condition.

(3) Deviation of numerical model

Not only for the FEA model but for the CFD model, similarity assumption has been taken in advance when modeling start. It is inescapability that the deviation of numerical model and the practice is exit at a certain extent. So it should be noted that the assumption parameter and the boundary condition for model input shall be selected as veritable as possible. Meanwhile the subjectively estimation and checking by other software for the numerical calculation result are also the same important.

5 CONCLUSIONS

The theory of FEA and CFD numerical calculation are widely used in many field. It is an excellent method for the numerical to be applied for SCR reactor& duct designing, as the complex structure and flow are both can be modeled accurately and expediently. From the report of practice execution and operation of Kemen project, the numerical simulation in this paper has been proved to be reasonable and accurate for practice.

6 REFERENCE

1. Code for design of steel structures (GB50017-2003).
2. W. Nischt B. Wooldridge Recent SCR Retrofit Experience on Coal-Fired Boilers.
3. Dudek S. A , Rogers J., Gohara W. F. Computational Fluid Dynamics (CFD) Model for Predicting Two Phase Flow in a Flue Gas Desulfurization Wet Scrubber [A] . In: Proceedings of EPRI2DOE2EPA Combined Utility Air Pollutant Control Symposium [C]. Georgia: Atlanta, 1999.
4. Huebner K. H and Thornton E. A. Finite Element Method for Engineers. John Wiley &Sons, Inc. 1982.