Structural Performance's Optimally Analysing and Implementing Based on ANSYS Technology

Na Han[1,4], Xuquan Wang[2],Haifang Yue[3],Jiandong Sun[1] and Yongchun Wu[5]

[1]Structure Department, Shandong Academy of Building Research, Jinan, China
[2]Feicheng City Department for Quality Supervision of Building Engineering, Feicheng, China
[3]Guangrao County Brueau for Garden Greening, Guangrao, China
[4]Shandong Jianke Architectural Design Co.Ltd, Jinan, China
[5]Business School, Shandong Jianzhu University, Jinan, China

Email: hannah_hn@163.com;

Abstract. Computer-aided Engineering (CAE) is a hotspot both in academic field and in modern engineering practice. Analysis System(ANSYS) simulation software for its excellent performance become outstanding one in CAE family, it is committed to the innovation of engineering simulation to help users to shorten the design process, improve product innovation and performance. Aimed to explore a structural performance's optimally analyzing model for engineering enterprises, this paper introduced CAE and its development, analyzed the necessity for structural optimal analysis as well as the framework of structural optimal analysis on ANSYS Technology, used ANSYS to implement a reinforced concrete slab structural performance's optimal analysis, which was display the chart of displacement vector and the chart of stress intensity. Finally, this paper compared ANSYS software simulation results with the measured results，expounded that ANSYS is indispensable engineering calculation tools.

1. Introduction to CAE

Computer-aided Engineering (CAE) is a new discipline which began to be developed in the 1960s. This process is a product which is combined with a numerical analysis methods and modern computer technology. With numerical analysis theory methods maturing and rapid development and wide application of computer technology, there is a broad prospects which commonly used CAE for engineering analysis improving design quality and shortening design circle. At that time, there were two fact that brought a great power in development of CAE [1]. One is basic idea of the finite element method which solves the problem of complex structures. In 1943, Courant, on his first attempt, applied the definition in the triangle area of piecewise continuous function and the principle of minimum potential energy to be solved St.Venant torsion problem. The second is wide application and rapid development of electronic computers. In the 1950's, J.Turner for the Boeing company by the United States, and J.H.Argyris in the university of London , puts forward the structural matrix analysis method, applied electronic computers as a major means of operation. The finite element method was developed on basis of this new method.

A typical procedure to implement CAE generally consists of five steps, which is showed in Figure 1: establishing mathematical model, selecting optimal algorithm, designing program, developing objectives and requirements, and computer automatic screening optimal analysis scheme.

2. The Necessity for Structural Optimal Analysis

Traditional analysis process is generally completed by the structural engineer with continuously testing according to past experience. Although structure can be improved to a certain extent after modification, but this kind of analysis has certain blindness and uncertainty. The final result is different from person to person. Because of the complexity of the structure, how to adjust analysis and design in order to get better, which is often a difficult problem to solve. Structural optimal analysis is effective method which makes a computer to replace human laborious work. Its research content is the combination of numerical theory and mechanics analysis method. It automatically improves and optimizes the bearing structure analysis in various conditions by establishing a set of scientific, systematic, reliable and efficient methods and software [2].

2.1. Lack of dynamic response to a change of parameters relating to design in the process of construction
In the process of structural analysis, there is a change of parameters relating to design in each step of the site construction, such as excavation depth, soil conditions, surrounding environment, the additional load, bending moment and shear force acting on the structure, these factors in affecting pouring quality of concrete, and so on. Computer aided structural optimal analysis can provide internal force analysis under various operating conditions.

2.2. Lack of scientificity and systematicness in traditional analysis method
The traditional analysis was finished generally by the structural engineer according to past experience. The result is different from person to person. In structural optimal analysis, the amount participating in calculation, one is constant, and other is variable, that forms a whole structure analysis solution domain. The analysis organically unites mechanics concept and optimization technology. Practice proves that the structural optimal analysis can shorten design cycle, save manpower, improve the quality and level of structure design. Economic benefit and social benefit is completely remarkable.

2.3. Lack of coordination between the informationization construction and design
We try to establish the concept of work with a computer data process and site data monitor at the core in the process of informatization construction. Traditional analysis cannot provide the data of a whole site construction, unable to form the management value. This make the structural analysis can't realize informatization with the result that its security should be greatly reduced. Structural optimal analysis can provide management data to achieve the design and construction informatization.

3. The Framework of Structural Optimal Analysis Based on ANSYS Technology
There are three typical analysis process including Preprocessor, Solution Processor and Post Processor.
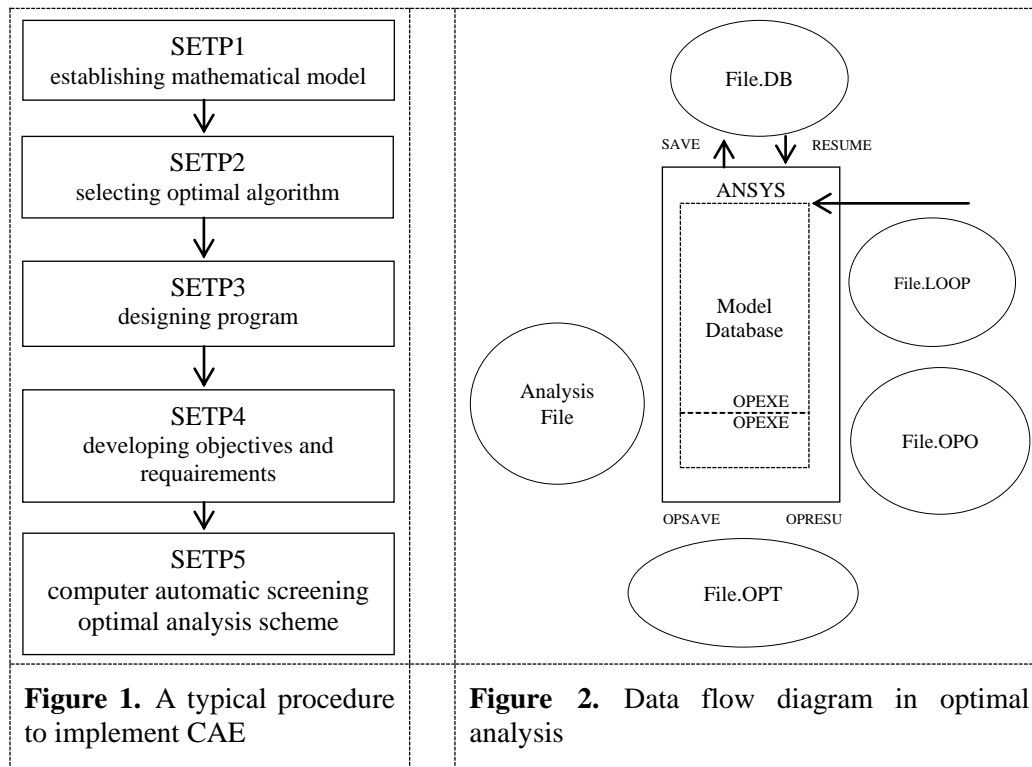
3.1. Preprocessor
The module has two main parts: the entity modelling and meshing. There are two kinds of entity modelling method: top-down and bottom-up. The users define a model primitives, thereby, to directly structure geometry model, such as circular，rectangular, 3D block, ball, taper and column. When Bottom-up modelling entity, users structure model upwards from the lowest figure yuan ,namely, users first define key point, and then in turn   the line, surface, body related to it. No matter using which kinds of method, the user can use Boolean operation to combine data sets, and "sculpture" a solid model. ANSYS program provides convenient and high quality of grid partition function for CAD model. Four meshing are included: extended meshing, image meshing, free classified meshing and adaptive meshing.

3.2. Solution Processor
The user can define analysis type, analysis options, load data and load options, then began to finite element solution. Loading is that the boundary condition data descript the situation of actual structure , and analyze the interaction between structure and outside world. There is the meaning of load such as the constrained displacement of freedom degree,   nodal force load (force, torque), surface pressure, and inertial load (gravity acceleration, angular acceleration).

3.3. Post Processor

A graphic form of the front analysis results can be displayed and output in post-processing. Different lines colors represent different values, different values, and clearly reflect the regional distribution. Different line colors represent different values, numerical areas, and clearly reflect resultant regional distribution. In addition, post-processing can also check results in a time period or a substep process .These results can be viewed by drawing curves or list, in order to help to visualize analysis results [3]. The data flow of analysis is shown in Figure 2.

Figure 1. A typical procedure to implement CAE

Figure 2. Data flow diagram in optimal analysis

4. The Implementing of Structural Optimal Analysis Based on ANSYS Technology

4.1. The experiment survey

There is a reinforced concrete slab of 1000*1000*180mm.Four levels exert concentrated load is applied respectively in the centre of the plate, which is 4kN, 6kN, 8kN, 10kN. Two long sides support slab, and two short sides node are constrained. The node doffs of UY constraints are applied to the slab bottom of Z = 1900 ~ 2000, at the same time, The node doffs of three directions constraints are applied to the slab bottom of Z = 0 ~ 100[4]. The change state of slab 's displacement and stress in the loading stage is observed and recorded. The accuracy of the numerical simulation analysis is verified by compared with the ANSYS software's calculation cases.

4.2. The simulation analysis of deflection and stress in the loading stage

4.2.1. The cloud charts of displacement vector sum.

A concentrated load is imposed on the center of slab. With concentrated load is gradually given an increase of 2kN, slab deflection value begins to expand gradually, however, the deflection value expands greatly slowly. The cloud charts of displacement vector sum are shown in Figure 3. From the figure, the deflection limit distributes the center on the bottom of the slab, whose value is the biggest, while the deflection value decreases in annular state trend on either side of slab [5]. There are four deflection cloud maximum values in the process of per level load, which are shown in Table 1.

4.2.2. The cloud charts of stress intensity.

A concentrated load is imposed on the center of slab. As the concentrated load is given an increase of 2 kN, stress intensity concentrates in the center of the slab bottom. The red part is dangerous stress area, which is also the largest stress intensity area of 1.09304 Pa. And then, you can draw a conclusion that stress intensity satisfies the requirement of the strength of material, if it is in the allowed range which is compared with the allowable stress of material. The minimum value is 0.051724 Pa, which is located on the support constraint of slab. There are shown in Figure 4. There are four stress intensity cloud maximum values in the process of per level load, which are shown in Table 2.
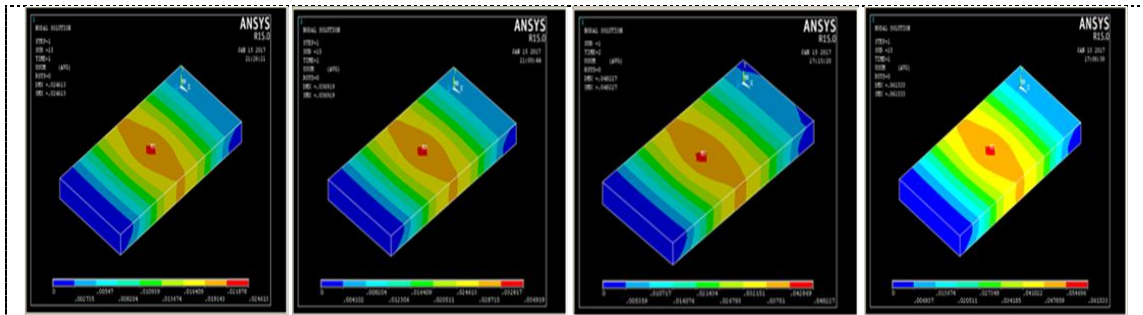

Figure 3. The cloud charts of displacement vector sum

Table 1. The displacement vector maximum of per level load.

Load(kN)	Displacement(mm)
4	24.613
6	36.919
8	48.227
10	61.533

Table 2. The stress intensity maximum of per level load.

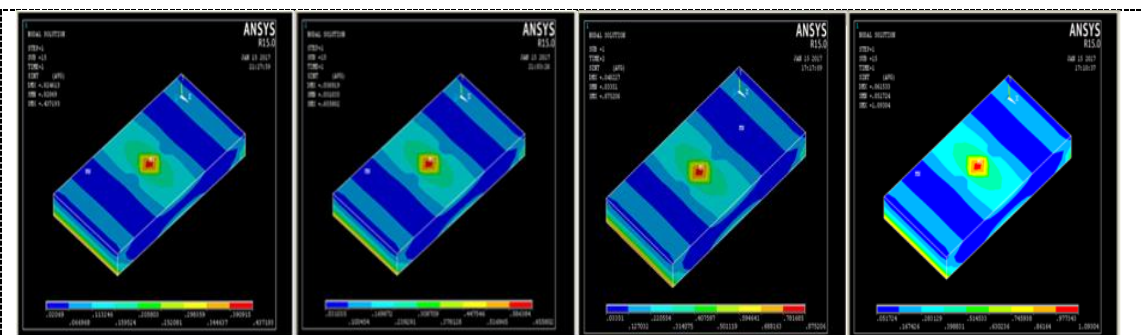| Load(kN)	Stress(Pa)
4	0.437193
6	0.655802
8	0.875206
10	1.09304

Figure 4. The cloud charts of stress intensity

4.2.3. Comparing ANSYS software simulation results with the measured results.

In the laboratory, the reinforced concrete slab is loaded with hydraulic jack. There are measured values of maximum deflection for leach level load which are shown in Table 3.

As using ANSYS software to simulate results, which comparing with the experimental measured results, we can see that the deflection values of computer simulation are consistent with the experimental result, which is shown in Figure 5. The chart is generally linear in the load - deformation curve chart [6].

4

The error in process is mainly manifested these great stress concentration in support of slab , which is with the corresponding concrete node displacement coordination; There are inaccurate dockings on working condition, which affect analysis model incorrect constraint and load.

Table 3. Comparison load with measured deflection

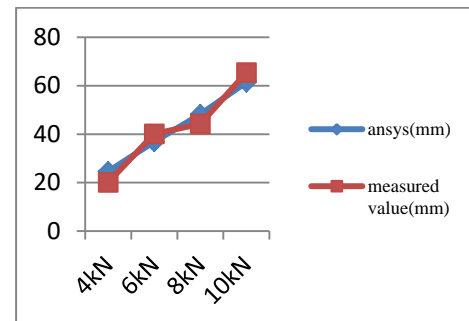| Jack readings (kN)	Measured deflection values(mm)
4	20.13
6	40.12
8	44.24
10	65.42

Figure 5. Comparison ANSYS simulation results with the measured results in deflection

5. Conclusions

The deflection and stress intensity of slab is simulated and analyzed by ANSYS soft. Although there is error in the calculating result of finite element model, it is basically in accordance with the measured results of experiment on the same change trend. The study shows that ANSYS is indispensable engineering calculation tools which simulate the stress process of slab better.

There are some problems in the finite element analysis of this study.（1）Some factors affecting convergence, such as mesh count, convergence criteria, bearing problem, appeared in the process of reinforced concrete slab's nonlinear calculation .（2）the reason of some error in process of finite element software calculation is that material parameters reflecting the calculation model and the working condition of docking is difficult to determine.

6. Acknowledgments

7. Reference

[1]   Wei Li 2007 *ANSYS Civil Engineering Application Examples* Water Conservancy and Hydropower Press
[2]   Mingwei Li 2012 *Based on ANSYS Finite Element Analysis of Deep Excavation Dynamic Optimization Design Method* Master Degree Dissertation of Xihua University
[3]   Hao Liu, et al 2014 *ANSYS15.0 Finite Element Analysis from Entry to the Master* Mechanical Industry Press
[4]   Cimian Zhu, et al 2016 *Strength of Structure* Higher Education Press
[5]   James M.Gere and Barry J.Goodno 2011 *Strength of Materials* Mechanical Industry Press
[6]   China Academy of Building Research 2014 *GB50204-2015 Code for Acceptance of Constructional Quality of Concrete Structures* the Ministry of Construction of the People's Republic of China