

The Pre and Post-Processing in Finite Element Analysis on the Technology and Programming of Plane Reinforced Concrete Based on AutoCAD Platform

Ying Zhao

Nanchang Key Laboratory of material and structure detection Jiangxi University of Technology

Keywords: Reinforced concrete; Finite-element method; AutoCAD; Pre and Post-Processing

Abstract: At present, reinforced concrete finite element method, as an effective numerical method has been widely applied and is officially listed as one of the design approaches for non-member reinforced concrete structure reinforcement in Design Specification for Hydraulic Concrete Structure. Finite-element pre-and post-processing is a bottleneck for the finite element method application and development so researches on it are of significant importance. This paper adopts the embedded VBA language development tool of AutoCAD to compile finite-element pre-and post-processing application software for plane reinforced concrete based on the nonlinear finite element analysis program conducted by Hohai University's Civil Engineering department, which realizes visualization and finite element analysis program as well as seamless integration of CAD. Users can quickly and reversibly carry out finite element modeling with visual graphs according to the design paper and the result is output based on requirement. This paper sums up with the ideas to take relevant segments automatic k-mesh generation technology to form element topology and improve the flexibility of finite element modeling.

Introduction

Reinforced concrete structure is the main structure for civil engineering, which can be divided into structure of bar system and nonlinear bar structure. The internal force of structure of bar system can be solved by mechanics of materials or structure and the carrying capacity can also be identified by the test. Therefore, we can adopt traditional empirical formula to carry out the designing.

The basic idea of finite element method is to discretize the structure and replace the original continuous structure with discrete structure model based on limited elements in limited nodes and then conduct the comprehensive solution according to deformation compatibility condition. Because the element number is limited, the node number is also limited so we call it finite element method.

Finite-element preprocessing includes: choosing selected element type and division to determine the numbers and coordinates of various nodes and elements, as well as the payload type, boundary conditions and material nature. This work is not only complicated and time-consuming but also will ignite mistakes. So finite-element pre-and post-processing is a bottleneck for the finite element method application and development. This paper is carried out with relevant aspects of finite element method's finite-element pre-and post-processing of reinforced concrete.

Since the birth of finite element mesh generation technology, various methods have been created, which are mainly the following seven approaches:

Firstly, manual mesh generation and semi-automatic mesh generation approach: Manual mesh generation refers to the approach that users input all of the information such as the element information and location etc. Semi-automatic mesh generation is to define and realize an existing reference space grid and then carries out transformation to acquire complicated high-dimensional mesh. The generation of the can be concluded in the following process: 2D network element- mesh generator.

Secondly, topology decomposition approach: Topology decomposition approach is put forward by Wordenwebert which is adopted in two-dimension problem and has been expanded to three-dimension, among which Woo and Thomasma's approaches are representative. In this approach, the division objects are polygon and polyhedron composited by vertices and edges. In the beginning, we divide the original geometric model into triangular or tetrahedron mesh without any nodes and then carry out subdivision so as to acquire precise requirements.

Thirdly, node connection approach: This approach includes two steps: node generation and element generation. Node generation: some mesh generators adopt interactive approach to generate node yet the current development trend is to automatically generate node for finite element model and random generating algorithm and nonrandom node generation approach of Cavendishl are representative methods. The demerit of random generating algorithm is carrying out multiple checks without reasonable node allocation. Element generation: this step links nodes to elements and most of the approaches aim to generate triangle or space tetrahedron and Delaunay's triangulation method is shining out.

Fourthly, grid-based approach: This approach is put forward by Tach, Gozalze and pttl-lad, which divides subjects based on both small and large equilateral triangles to eliminate the triangles outside the area and modify the boundary grid to form the expected mesh.

Fifthly, mapping approach: Its basic steps are: map the physical domain to parameter space so as to formulate rule parameter field; carry out mesh generation of the rule parameter; map the mesh in the parameter filed to the physical domain so as to obtain finite element grid.

Sixthly, geometry decomposition approach: Geometric decomposition method is also known as recursive segmentation method and iterative segmentation method. Recursive segmentation method is proposed by Bykat and its basic idea is to divide the domain into two areas based on the best split shaft or clipping plane and then divide the two areas into four units which cannot be split up.

Seventhly, automatic adaptive mesh generation approach: The adaptability of finite element is a closed circuit circulation process based on the current mesh to evaluate the calculation error and carry out analysis grid and calculation according to calculation results by finite element. If the error achieves the pre-specified value, the adaptive process ends.

Procedure systemfunction introduction

Pre and post processing of finite element analysis is an important step in finite element calculation, the pre processing is always a bottleneck. How to accurately and effectively establish finite element model and carry out data analysis are urgent problems for many technicians. At present civil and hydraulic engineering design generally adopts CAD drawing, for example, technicians directly carry out pre processing in the CAD platform which can take full advantage of the existing material to reduce the input at large. At the same time, the combination of CAD technology and finite element technique is also a development trend of modern CAE technology. Therefore, this paper selects AutoCAD as the operating platform so as to carry out secondary development and compile pre and

post processing to form plane reinforced concrete finite element analysis software based on the AutoCAD platform while combining existing computer program.

The software has a very friendly graphic user interface and users can directly and quickly carry out mesh implementation, information analysis, loading, temperature margin setting, consolidation information processing based on human-computer interaction through visualization to finally generate required data for finite-element analysis. In addition, it can arrange the result show and input the result with color contours or slice. The software structure is shown in table1.

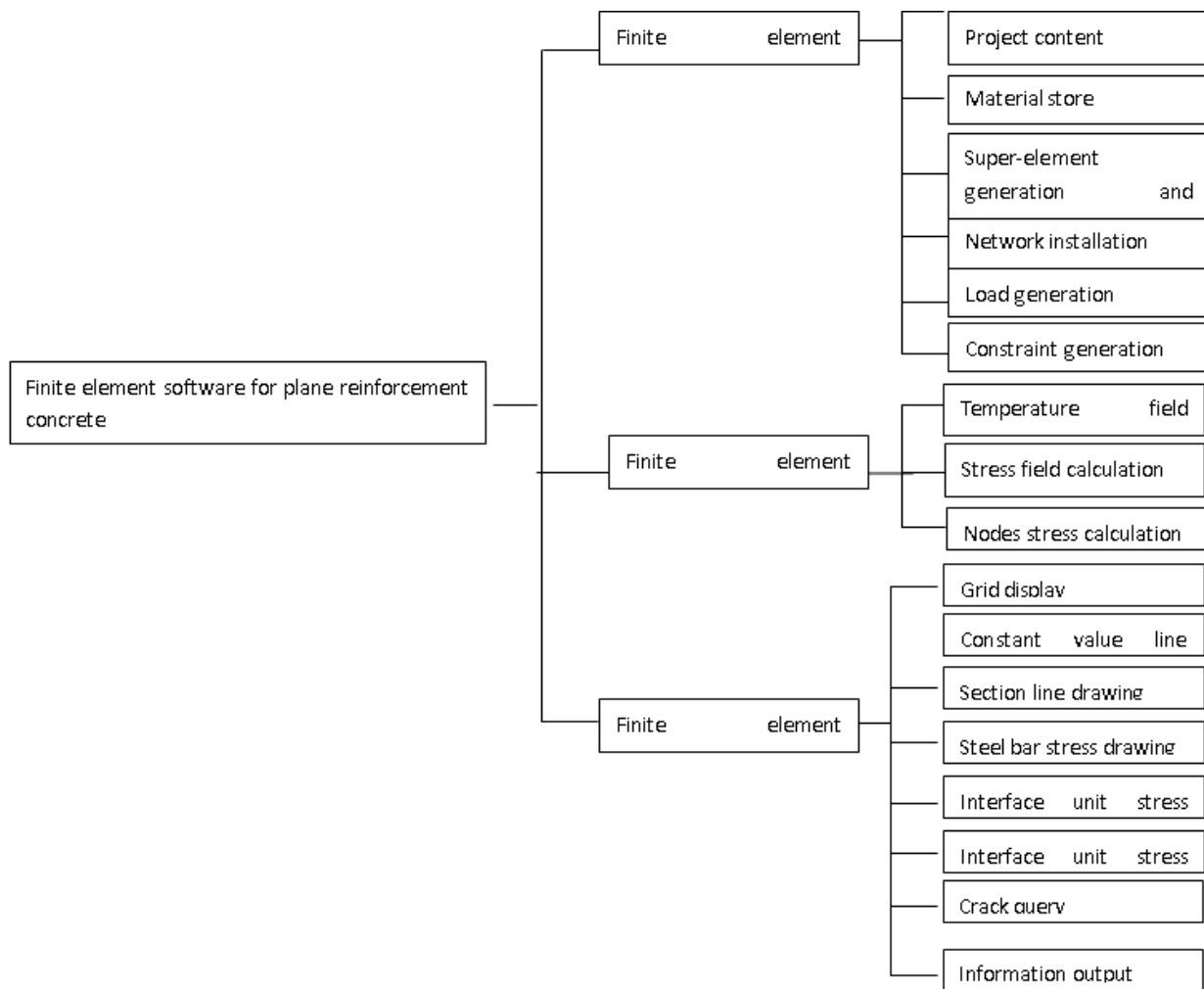


Table 1 Finite element software structure figure for plane reinforced concrete

After the completion of pre processing, the finite element calculation module will complete the finite element numerical calculation. The pull-down menu includes calculation of temperature field, stress field calculation, node stress treatment. In order to treat the post processing, this procedure makes a new AutoCAD menu: click the button to make an order of post-processing system for finite element and the menu has 8 subcommands of grid display, construct deformation, draw constant value line, draw section line, draw steel bar stress, draw interface unit stress, crack query and information output.

Function implementation of program system

The basic idea of this program is to realize the visualization of the whole calculation process, namely, adopt graphic or dialog box to show the whole process from establishment to the result and put the complicated data calculation as well as data processing to the backstage. Users can draw and modify the figures on the screen through multimedia technology. After the formation of calculation module, the input document is automatically generated. Besides, the finite element calculation can help users to acquire calculation results with graphic or document form through interactive approach. In order to realize the visualization for finite element calculation, we have to choose proper development platform and carry out a second development so as to improve the development efficiency as well as cycle. AutoCAD is the most popular graph processing in the engineering, which does not only have powerful and perfect graph drawing, editing ability and friendly interface but also allows users to carry out a second development. In China, AutoCAD is the most popular CAD software for PC application. In most of the designing institute, no matter cartography or machinery industry, AutoCAD can be adopted to draw. Therefore, this paper adopts AutoCAD as the post and pre processing operating platform so as to directly complete the finite element calculation.

VBA is the abbreviation of Visual Basic for Applications. Same to VB, VBA is a program design language facing objects with simple language, powerful function and quicker speed. AutoCAD VBA is embedded in AutoCAD, operating in the same space with more outstanding operating efficiency. It communicates with AutoCAD via ActiveX to operate the object architectures (including entity object, style setting object, organization structure object, graphic display object, document and applied procedure objects, menu and tool bar etc) and acquire the approach and nature of objects. Therefore, it can be concluded that AutoCAD is a perfect development tool. Its advantages show as follows:

Firstly, VBA can provide powerful form creation function. In comprehensive development environment, simple icon dragging can help to establish the GUI as well as the dialogue box and interface for applied procedure in order to carry out the interaction.

Secondly, VBA has development object module. The object module of ActiveX in AutoCAD indicates the approach and nature of each object and users can compile or edit AutoCAD software based on their desire and realize the seamless integration of procedure and AutoCAD.

Thirdly, VBA can help to realize the transformation between data document and graphic. The generated graphic can be adopted to check the accuracy of data document. Besides, it is quick to find out the error which improves the finite element modeling. All in all, AutoCAD, as a development platform can help to develop the finite-element pre-and post-processing to realize the visualization of pre-and post-processing and decrease the development cost. As a result, the developed software shall be granted with widespread mass base and wide prospects for engineering applications.

Program assessment

In entering the system, the first thing is to establish a project. So the relevant document data of both the pre-and post-processing is reserved in the project. Before dividing the super-elements, we should firstly input material information of control parameters, block element, reinforcing bar, double spring element and interface element. Then, sectional view provided by the designing institute is added into the AutoCAD interface along with surrounding rock boundaries and reinforcing bar to get the basic shape shown in Fig. 1.

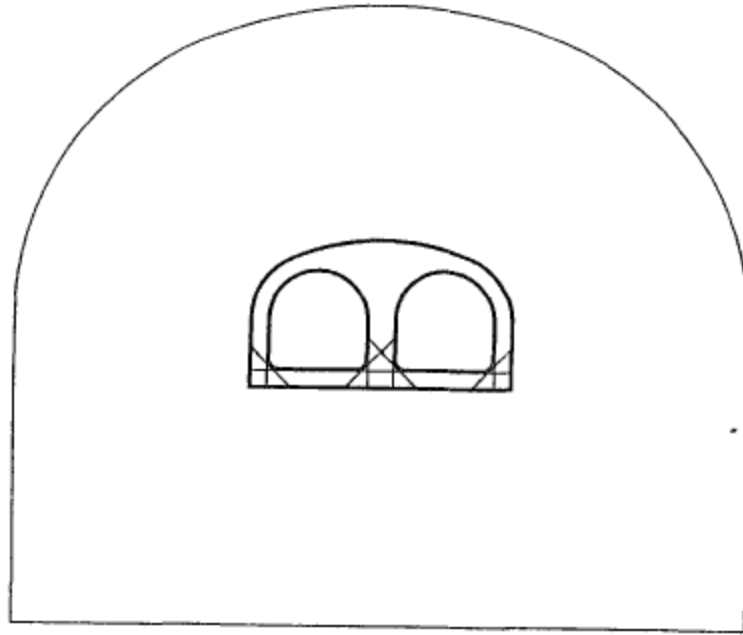


Fig. 1 The basic graphic for central pier water-carrying tunnel's pre-finite-element preprocessing

At present, determine the reinforcement bar from the basic graphic, namely to determine the module and material type after calculation. Then, divide super-element. Considering the different division requirements of materials, 13 super-elements are divided initially. Based on the super-element formation and super-element determination by manual, the topology information is formed and displayed in the AutoCAD interface, the Fig. 6 and 8 are empty.

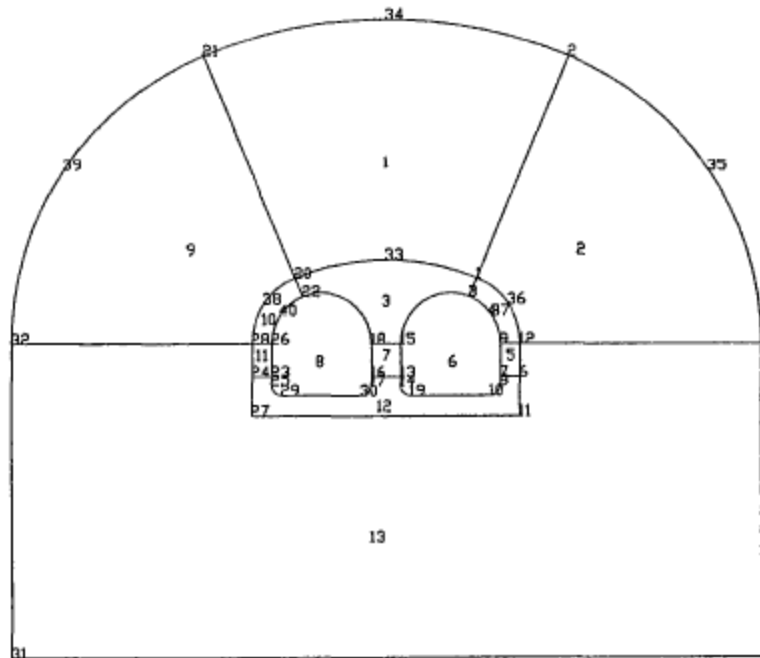


Fig. 2 Super-element topology information display

The calculation model is pier wall intercepting plane, adopting symmetry to calculate the half and set linkage constraint. The concrete takes 4-8 nodes as elements while reinforced bar takes 2-3 nodes as the beam. In addition, the cohesive element adopts double spring element; the concrete

constitutive relation adopts equivalent uniaxial constitutive model and bond slippage relation adopts Houde formula. In order to accurately calculate the crack width and distance, the pier-wall length should be less than 200mm. In addition, in order to accurately consider the temperature and the distribution of constrictive strain, new concrete is divided into 3 layers. Besides, in order to calculate the temperature of concrete placement the construction shall be divided into two stages.

Conclusions

The pre-processing procedure does not only help to directly and quickly carry out the mesh generation between human and computer but also realizes the automatic generation of bar element and interface element. At the same time, it has various finite element modeling such as information given, loading, temperature boundary setting, consolidation information processing etc. Besides, it can directly show the finite element modeling construction process with graphics to make it convenient for users to modify. What is worth noticing is that:

a. In information given, it establishes a friendly interface and input the required formula on the interface so that users can intuitively input relevant data based on the formula.

b. The procedure provides five reinforcement element models which can automatically generate reinforcement element and bond element information.

c. The mesh generation adopts transform subdivision and manual subdivision and users can combine these two methods to divide the super-element or any sub-domains which can quickly and flexibly realize any complicated element refinement.

d. While carrying out loading, temperature function and constraint information input or editing, this procedure can timely feedback the login and change of icon indicates.

e. The overall finite element modeling is visible and reversible. The procedure provides the information checking function for basic management grid, loading constraint. If there should be any inconformity or mistakes, users can timely correct and modify the mistakes to greatly improve the accuracy and efficiency of modeling.

Because of limited time and ability, some work needs further development, especially the following aspects:

1. Starting from the promotion application and integrity of finite element program design, this system lacks help system, including help files and some floating icons. Besides, the menu setting is not comprehensive which cannot realize the sidebar menu and keyboard command.

2. There should be a glittering array of data information in the finite-element pre-and post-processing and instant data exchange and data query shall be carried out. Therefore, the data base can be established to improve the calculation efficiency, which will be beneficial to the data management and query.

3. In post processing, internal force calculation and display, reinforcement calculation and display shall be added to form comprehensive software which integrates analysis and design.

Acknowledgment

This work was supported by Project on professional and characteristic construction of Jiangxi province 2010 (Civil Engineering) and Project on the planning and construction of disciplines in Jiangxi University of Technology (Structure Engineering)

References

- [1] Chi H L, Wang X, Jiao Y. BIM-Enabled Structural Design: Impacts and Future Developments in Structural Modelling, Analysis and Optimisation Processes[J]. Archives of Computational Methods in Engineering, 2015, 22(1): 135-151.
- [2] Mahindru D V, Priyanka Mahendru S, Ganj T. Review of Rapid Prototyping-Technology for the Future[J]. Global Journal of Computer Science and Technology, 2013, 13(4).
- [3] Lu J. Parallel finite element modeling of earthquake ground response and liquefaction[J]. 2006.
- [4] Sun Q. Internet-based distributed collaborative environment for engineering education and design[D]. , 2001.
- [5] Gunnarsson L. A prototype knowledge based system for pressure vessel design[J]. 1991.
- [6] Rahman A B M M. Assessment of Bridge Service Life Using Wireless Sensor Network[D]. Youngstown State University, 2012.
- [7] Mueller C T. Computational exploration of the structural design space[D]. Massachusetts Institute of Technology, 2014.
- [8] Miner S M. Mechanical Engineering Department[J].
- [9] Mensah-Bonsu P O. Computer-aided Engineering Tools for Structural Health Monitoring under Operational Conditions[J]. 2012.
- [10] Khalid R. Component-Based Tools for Educational Simulations[J]. 2013.