On the Development of 3D Finite Element Method Package for CEMTool

Jung Hun Park*, Choon Ki Ahn*, and Wook Hyun Kwon*

*School of Electrical Engineering and Computer Sciences, Seoul National University, Seoul, 151-744, Korea
(Tel: +82-2-873-4704; Fax: +82-2-871-7010; Email: jhpark, hironaka, whkwon@cisl.snu.ac.kr)

Abstract: Finite element method (FEM) has been widely used as a useful numerical method that can analyze complex engineering problems in electro-magnetics, mechanics, and others. CEMTool, which is similar to MATLAB, is a command style design and analyzing package for scientific and technological algorithm and a matrix based computation language. In this paper, we present new 3D FEM package in CEMTool environment. In contrast to the existing CEMTool 2D FEM package and MATLAB PDE (Partial Differential Equation) Toolbox, our proposed 3D FEM package can deal with complex 3D models, not a cross-section of 3D models. In the pre-processor of 3D FEM package, a new 3D mesh generating algorithm can make information on 3D Delaunay tetrahedral mesh elements for analyses of 3D FEM problems. The solver of the 3D FEM package offers three methods for solving the linear algebraic matrix equation, i.e., Gauss-Jordan elimination solver, Band solver, and Skyline solver. The post-processor visualizes the results for 3D FEM problems such as the deformed position and the stress. Consequently, with our new 3D FEM toolbox, we can analyze more diverse engineering problems which the existing CEMTool 2D FEM package or MATLAB PDE Toolbox can not solve.

Keywords: Finite Element Method, CEMTool, MATLAB, 3D FEM package, 2D FEM package, PDE toolbox, Pre-processor, Solver, Post-processor

1. Introduction

Finite element method (FEM) [1] is well known as a useful numerical method that analyzes a complex engineering problem. It provides a greater flexibility to model complex geometries than finite difference and finite volume methods do. For example, it has been widely used in solving structural, mechanical, heat transfer, and fluid dynamics problems as well as problems of other disciplines. The advancement in computer technology enables us to solve even larger system of equations, to formulate and assemble the discrete approximation, and to display the results quickly and conveniently. This has also helped the finite element method become a powerful tool. There are many kinds of the specialized FEM package for electro-magnetics, structural mechanics, heat transfer, and diffusion. Among the numerical general-purpose packages for scientific computing, MATLAB is one of the well-known package for science and engineering that performs mathematical and engineering computation. As a tool to analyze the solution of Partial Differential Equation (PDE) in MATLAB, the MATLAB PDE Toolbox [2] contains basic tools for the study and solution of PDE in two space dimensions (2-D) and time, using the finite element method. Its command line functions and graphical user interface can be used for mathematical modeling of PDE in some ranges of engineering and science applications.

As another powerful numerical general-purpose package, CEMTool integrates mathematical computing, visualization, and a powerful high-level language to provide a flexible environment for technical computing [3], [4]. The powerful architecture makes it easy to use CEMTool and its companion toolboxes to explore data, create algorithms, and create custom tools that provide early insights and competitive advantages [5], [6]. In CEMTool, similar to MATLAB PDE Toolbox, 2D FEM package has been developed until now. This CEMTool 2D FEM package overcomes some existing disadvantages of MATLAB PDE Toolbox. So, it can guarantee the shorter computational time and the more accurate solution than MATLAB PDE Toolbox. In particular, by using the advanced library of CEMTool 2D FEM package, we can analyze the complex real industry problems such as the motor field analysis [7], [8], [9], [10], [11], [12], [13]. In this paper, we present new CEMTool 3D FEM package which is different from the already developed CEMTool 2D FEM package. Our 3D FEM package in CEMTool can deal with complex 3D models, not a cross-section of 3D models. The pre-processor of 3D FEM package can make and graph a geometric data of complex 3D models by using Parasolid library and OpenGL library. Based on this 3D model geometric data, a new 3D mesh generating algorithm can generate the information on 3D Delaunay tetrahedral mesh of 3D elements for analyses of 3D FEM problems. And, because of the adopted Graphic User Interface (GUI), by using a mouse click, we can assign the boundary and loading conditions in a very easy way. The solver of the 3D FEM package offers the three methods for solving the linear algebraic matrix equation, i.e., Gauss-Jordan elimination solver, Band solver, and Skyline solver. Usually, in FEM analysis fields, Skyline solver is well known as an excellent method for solving the large linear algebraic matrix equation. So, our 3D FEM package can guarantee the shorter computational time and the more efficient memory usage. The post-processor visualizes the results for the 3D FEM problems such as the deformed position and the stress. So, nonspecialists can easily understand the analyzed resultants which are represented as a graphical figure. And, we will discuss the detailed structure and the important features of the proposed 3D FEM package. As a result, the proposed CEMTool 3D FEM package can solve various engineering problems where can be described by 3D geometry in contrast to the existing CEMTool 2D

This paper is supported by the ministry of commerce, industry, and energy.

FEM package or MATLAB PDE Toolbox.
In Section 2, the overall procedure for the FEM analysis is described. In Section 3, the detailed structures and the important features of CEMTool 3D FEM Package are discussed. The conclusion is given in Section 4.

2. Overall Procedure for using General FEM

Generally, for the analysis of FEM, we ask for three processing sequences, pre-processor, solver, and post-processor. Our CEMTool 3D FEM package is conformed by this procedure. Usually, in pre-processing step, we carry out a modeling of an actual structure. After a modeling work, we execute the discretization and mesh generation. Then, we enter the material properties and the boundary conditions. Secondly, in solving step, by using the information inputting in previous pre-processing step, we operate the FEM analysis as a major one. First of all, we select the interpolation function of each element. Next, using this interpolation function, we fulfill the linearization and derive linear algebraic matrix equation at each element. Then, in order to make the stiffness matrix of the overall system, we assemble each element matrix equation. To solve the assembled linear algebraic matrix equation, we can obtain an approximate solution at each node. Finally, in post-processing step, by using the various tools, we check several results of the solved 3D model from the pre-processor and the solver, after the finite element analysis is completed. In order to visualize the result of the finite element analysis, it uses information produced in 3D FEM pre-processor and solutions obtained from 3D FEM solver, i.e., generated mesh data and analyzed resultant data. For easy understanding, these data are represented by mesh plot or flux density plot. The basic procedure for the FEM analysis can be described in Fig 1. The using procedure and the actual implementation of CEMTool 3D FEM package will be introduced in the following section delicately.

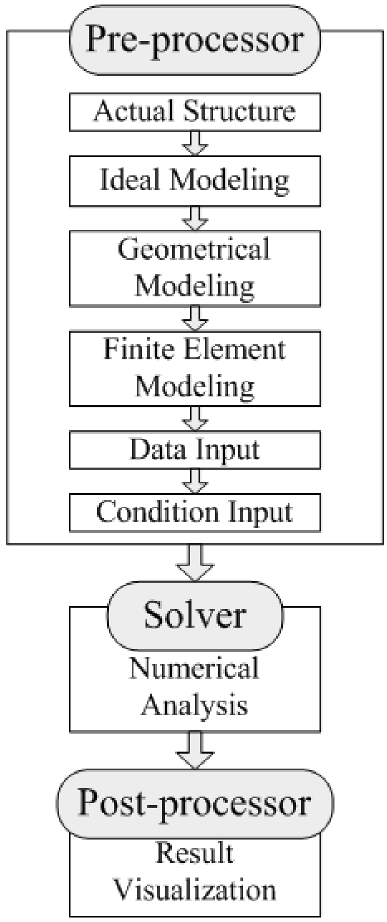

Fig. 1. The basic procedure for general FEM analysis

3. Detailed Structures and Important Features of CEMTool 3D FEM Package

3.1. 3D FEM Pre-processor

CEMTool 3D FEM pre-processor requires the information on 3D models, which will be wanted to analyze in CEMTool environment. Specially, there are the shape of 3D model, the characteristic of material, and the boundary condition of the region, the division of elements as input. Then, by using input data, it produces the information on elements, nodes, regions, and materials for finite element analysis as output. For the purpose of easy input for model shape, it provides various kinds of new and convenient tools and dialogs. We can easily input the basic shape information by clicking the menu button. For example, our 3D FEM package has the functions that make the six 3D standard models (Hexahedron, Sphere, Cylinder, Prism, Cone, Torus). Also we can conveniently enter the complex 3D model which exists in the real world with various existing CAD tools such as SolidWorks which is the famous 3D mechanical design software. Then, we can make the 3D Delauany tetrahedral mesh by using new 3D mesh generator. If we choose the appropriate mesh option (Radius-edge ratio, Volume) according to the analyzed model feature, the developed 3D FEM package is capable to obtain more accurate results. And, we can input the material values and the boundary conditions with convenient dialogs. Also, we can input the boundary condition in a very easy way. Because of the adopted Graphic User Interface (GUI), we can assign the boundary and loading conditions by using a mouse click. These features of CEMTool 3D FEM pre-processor are given in three following figures. In Fig 2, the motivation of the development of CEMTool 3D FEM package is given. As seen in Fig 3, our 3D FEM package has the software compatibility with SolidWorks. And, new high-performance tetrahedral mesh generation algorithm is seen in Fig 4.

3.2. 3D FEM Solver

CEMTool 3D FEM solver uses various information produced in CEMTool 3D FEM pre-processor and performs the finite element analysis. First of all, select the interpolation functions of tetrahedral elements and, using this interpolation function, determine the relationship between tetrahedral elements. With the obtained shape function, in order to make the coefficient matrix of the overall system, it assembles the relationship equations between tetrahedral elements. It solves the assembled linear algebraic matrix equation and obtains the solutions at every node in a pre-defined region. Our 3D FEM solver employs the three methods for solv-

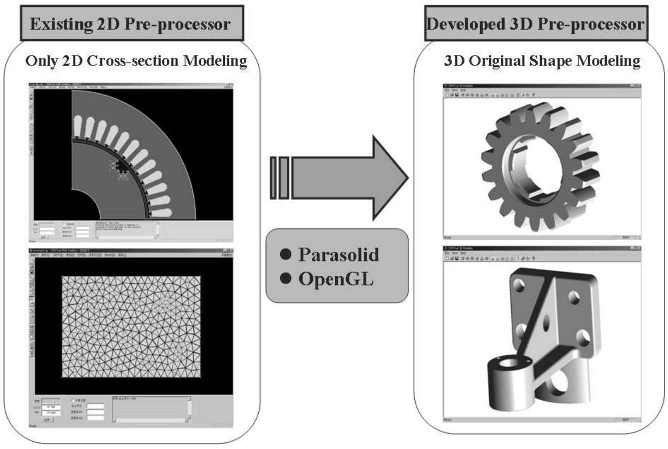

Fig. 2. Motivation of the CEMTool 3D FEM Package

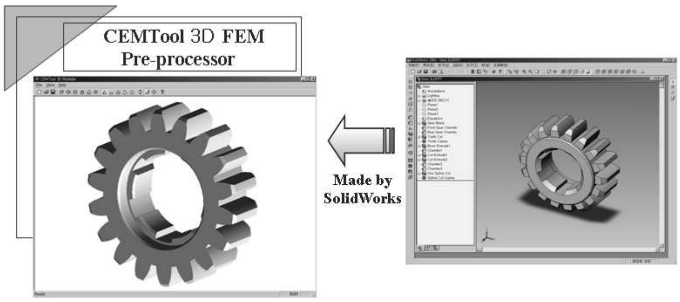

Fig. 3. Software Compatibility between CEMTool 3D FEM Package and SolidWorks

ing the linear algebraic matrix equation, i.e., Gauss-Jordan elimination solver, Band solver, and Skyline solver. Above all, Skyline solver technique is widely used as an optimized method for solving the large linear algebraic matrix equation. It needs the minimum fill-in and saves the memory. Table 1 show the comparison with three kinds of FEM solver, Gauss-Jordan elimination solver, Band solver, and Skyline solver. So, our 3D FEM package can guarantee the shorter computational time and the more efficient memory usage than the existing CEMTool 2D FEM package or MATLAB PDE Toolbox. Also, since CEMTool contains various kinds of the well-known optimization library such as Lapack Library and Intel Library, high-speed and accurate solution are guaranteed.

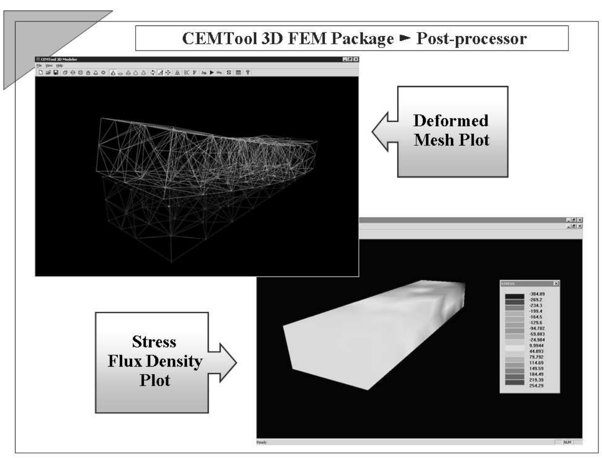

Fig. 4. Tetrahedral Mesh Generation Algorithm

Table 1. The comparison with three kinds of FEM solver, Gauss-Jordan elimination solver, Band solver, and Skyline solver.

	Gauss-Jordan Elimination	Band Solver	Skyline Solver
Computation Time	long time	middle time	short time
Memory Usage	large usage	medium usage	small usage

3.3. 3D FEM Post-processor

CEMTool 3D FEM post-processor represents various information of the analyzed 3D model in a pre-defined region after the finite element analysis is completed. In order to visualize the result of the finite element analysis, it uses information produced in 3D FEM pre-processor and solutions obtained from 3D FEM solver. Our 3D FEM post-processor offers the mesh plot and flux plot function after the finite element field analysis for general structural model is completed. One of the analyzed results, the deformed position of 3D model, can be visualized in mesh plot. And, another analyzed result, the stress in every nodes, can be visualized in flux density plot. So, non-specialists can understand the analyzed resultants which are represented as the graphical figure. Fig 5 shows that CEMTool 3D FEM post-processor can represent mesh plot and flux density plot, after the simple structural model is analyzed by CEMTool 3D FEM solver. Another feature of our post-processing tool is that it is possible to interface with the external software such as Origin, Excel, and other existing engineering packages. The detailed result data can be saved as a text file. So, we can use this data file in the external software as seen in Fig 6.

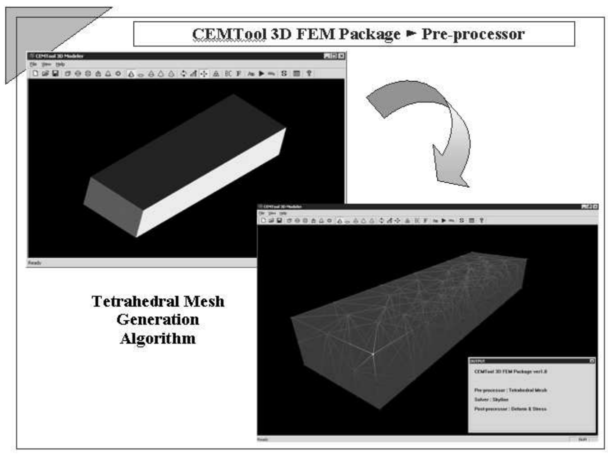

Fig. 5. Two plots for 3D FEM post-processing, Mesh plot and Flux Density plot

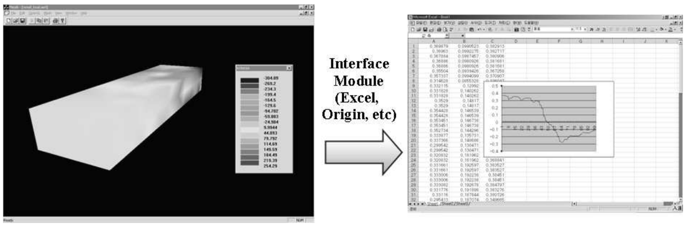

Fig. 6. Interface with the external software

4. Conclusion

In this paper, we present new 3D FEM Package in CEMTool environment. Our 3D FEM package can overcome the existing shortages of CEMTool 2D FEM package or MATLAB PDE Toolbox with the developed new pre-processor, solver, and post-processor. Due to these tools, in contrast to the existing CEMTool 2D FEM package or MATLAB PDE Toolbox, we can analyze more various 3D geometry engineering problems, which the existing CEMTool 2D FEM package or MATLAB PDE Toolbox can not solve. Also, CEMTool 3D FEM package can guarantee the shorter computational time, the more efficient memory usage, and the more accurate solution. So, it can be applied to practical design & analysis engineering problems such as structural analysis. We expect that expert and nonexpert can utilize the proposed CEMTool 3D FEM package in industry and education.

References

[1] Daryl L. Logan, *A First Course in the Finite Element Method*, PWS Engineering, 1992.

[2] The Mathworks, Inc., *Partial Differential Equation Toolbox : User's Guide*, 2002.

[3] S. H. Han, S. K. Choi, K. H. Lee, J. S. Lee, and W. H. Kwon, "The implementation and the application of CEMTool," *in proceedings of the 14th KACC*, 1999.

[4] W. H. Kwon, K. B. Kim, S. K. Choi, H. J. Kim, and S. H. Han, "The technical trend and the practical application of the real-time control system design in CEMTool," *ICASE Magazine*, vol. 5, pp. 18-26, 1999.

[5] W. H. Kwon, K. B. Kim, and S. G. Choi, "Control System Design and Analysis with CEMTool for Control Education," *Transaction on Control, Automation, and Systems Engineering*, March 2001.

[6] W. H. Kwon, K. B. Kim, and S. G. Choi, "Control System Design and Analysis with SIMTool and CEMTool for Control Education," *Transaction on Control, Automation, and Systems Engineering*, March 2000.

[7] C. K. Ahn, S. H. Han, Y. S. Lee, and W. H. Kwon, "On the Implementation of GUI Based Finite Element Method Library in CEMTool," *In the proceedings of KIME Annual Conference*, 2002.

[8] C. K. Ahn, T. I. Lee, S. H. Han, Y. S. Lee, and W. H. Kwon, "On the Implementation of PDE Solver using Finite Element Method in CEMTool," *In the proceedings of KIEE Annual Conference*, 2003.

[9] C. K. Ahn and W. H. Kwon, "Generalized Command-Mode Finite Element Method Toolbox in CEMTool," *In the proceedings of International Conference on Control, Automation and Systems*, 2003.

[10] C. K. Ahn, S. H. Han, and W. H. Kwon, "An equation based approach to the implementation of generalized PDE solver in CEMTool," *In the proceedings of 16th International Conference on Computer Applications in Industry and Engineering, Las Vegas, USA*, 2003.

[11] C. K. Ahn, S. H. Han, and W. H. Kwon, "The Development of Finite Element Package in CEMTool," *In IEEE CCA/ISIC/CACSD, Taipei, Taiwan*, 2004.

[12] J. H. Park, C. K. Ahn, and W. H. Kwon, "On the Implementation of Waveguide Analysis Library using Vector Finite Element Method in CEMTool," *In the proceedings of IEEK Annual Conference*, 2004.

[13] C. K. Ahn, J. H. Park, and W. H. Kwon, "On the Implementation of Waveguide Analysis Library using Vector Finite Element Method in CEMTool," *In the proceedings of KIEE Annual Conference*, 2004.