2004년도 대한전기학회 하계학술대회 논문집 2004.7.14-16

CEMTool환경에서 GUI 및 명령어 기반 유한요소법 패키지 개발에 관하여

안춘기, 박정훈, 권욱현
서울대학교 전기컴퓨터공학부

On the Development of GUI and Command Based Finite Element Method Package in CEMTool

Choon Ki Ahn, Jeong Hoon Park, and Wook Hyun Kwon
School of Electrical Engineering & Computer Science, Seoul National University.

Abstract - CEMTool 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 FEM (Finite Element Method) package in CEMTool environment. In contrast to the existing MATLAB PDE Toolbox, our proposed FEM package can deal with the combination of the reserved words. Also, we can control the mesh in a very effective way. With the introduction of new mesh generation algorithm and NDM (Nested Dissection Method), our FEM package can guarantee the shorter computational time than MATLAB PDE Toolbox. In addition, using the advanced electromagnetics library of CEMTool FEM package, we can analyze the practical problems such as the motor field analysis. Consequently, with our new FEM package, we can overcome some disadvantages of the existing MATLAB PDE Toolbox.

1. Introduction

Finite element method (FEM) has been widely used in solving structural, mechanical, heat transfer, and fluid dynamics problems as well as problems of other disciplines. There are many kinds of the specialized FEM package for electromagnetics, 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 PDE (Partial Differential Equation) in MATLAB, the MATLAB PDE Toolbox [1] 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 special ranges of engineering and science applications. However, MATLAB PDE Toolbox has some disadvantages as follows: (1) It can deal with only constant coefficients. (2) It requires much computational time for mesh generation and solving for the finite element analysis. (3) It has no mesh control and refinement algorithms. (4) It cannot be applied to the practical analysis and design problems such as motor field analysis.

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 [2, 3, 4, 5]. In this paper, we present a new FEM package which overcomes the some existing disadvantages of MATLAB PDE Toolbox. Our FEM package in CEMTool contains the lexical analyzer and the parser to deal with the general combination of reserved words. That is why we can solve the PDE represented by the combination of some reserved words and mathematical operators [4, 5]. Also, it contains mesh refinement and control algorithms by the dense factor in GUI (Graphic User Interface) FEM Package. So, it is possible to obtain the accurate results in a pre-defined region. In addition, it guarantees the high speed for finite element analysis with new mesh control algorithm and NDM (Nested Dissection Method). Finally, since our new FEM package has the advanced electromagnetics library, we can solve the practical analysis and design problems for electromagnetics.

In Section 2, GUI based CEMTool FEM package is presented. In Section 3, we discuss command-mode FEM package of CEMTool. In Section 4, we introduce the advanced electromagnetics library. The conclusion is given in Section 5.

2. GUI FEM package of CEMTool

2.1 GUI FEM Pre-processor

CEMTool GUI FEM pre-processor requires the information on the shape of model, the characteristic of material, and the boundary condition of the region, the division of elements as input. Then, 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 and boundary condition, it provides various kinds of new and convenient tools and dialogs. We can input the shape information with various existing CAD tools such AutoCAD easily. Then, we can input the material values and the boundary conditions with convenient dialogs. It is possible to make partial modification

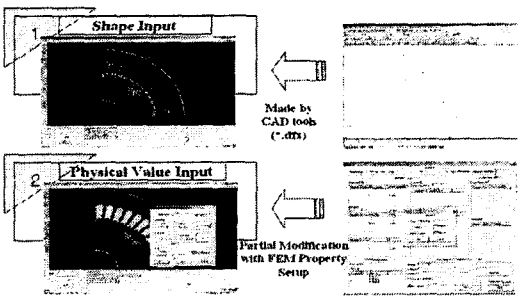

Fig 1. Some features of GUI FEM pre-processor

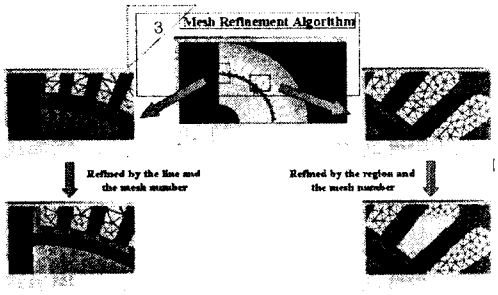

Fig 2. Mesh refinement algorithm

using FEM property setup program. These features of GUI FEM pre-processor are given in Fig 1. The new feature of our FEM package is that it is possible to refine the mesh using some parameters. With new high-performance mesh refinement algorithm, as seen in Fig 2, we can make the mesh refinement by assigning the lines and specifying the region together with the mesh number. Using this technique, more accurate result can be obtained in pre-defined lines and region. For demonstration of high-speed mesh generation, if we consider the dielectric analysis problem in equi-fields, our package guarantees about 10 times shorter computational time than MATLAB PDE Toolbox for the cases of nodes 3600 and nodes 5000, respectively.

2.2 GUI FEM Solver

GUI FEM solver uses various information produced in GUI FEM pre-processor and performs the finite element analysis. Our GUI FEM solver employs NDM (Nested Dissection Method) which is one of direct solving technique. This technique needs the minimum fill-in and saves the memory. So, it requires the shorter computational time than MATLAB PDE Toolbox. Also, since it contains various kinds of the well-known optimization library such as Lapack Library and Intel Library, high speed and accurate solution are guaranteed.

2.3 GUI FEM Post-processor

CEMTool GUI FEM post-processor represents various information 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 GUI FEM pre-processor and solution obtained from GUI FEM solver. Fig 3 shows that CEMTool GUI FEM post-processor can represent vector potential plot, flux density plot, mesh plot, and material distribution plot after the finite element field analysis for motor is completed. 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. If we assign a section in post-processor, the detailed data in the assigned section can be saved as a text file. So, we can use this data in the external software.

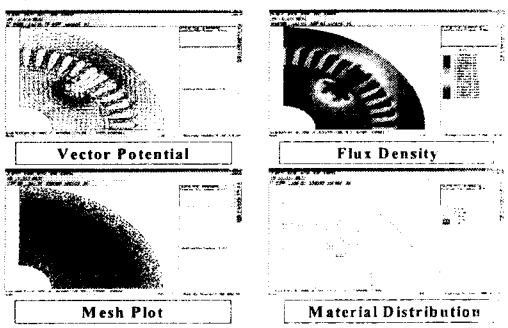

Fig 3. Various plots for FEM post processing

3. Commmand-mode FEM Package of CEMTool

3.1 Basic Command-mode FEM Functions

In order to obtain the solution of PDE represented by a equation form in a pre-defined region, it is not appropriate to use GUI based FEM package. In this situation, we need a new FEM package which can solve the combination of the reserved words directly. CEMTool command-mode FEM package has the lexical analyzer and the parser to deal with the general combination of reserved words such as "sin", "cos", and "convect". That is why we can solve the PDE represented by the combination of some reserved words and mathematical operators. Basically, the border function of CEMTool FEM Toolbox requires some information on the boundary which is represented by the combination of the reserved words in x and y coordinates. After defining the boundary region, in order to generate the triangular mesh based on a Delaunay-Voronoi algorithm, we can employ the makemesh function of CEMTool FEM Toolbox by using the information on the maximum number of vertices as the only parameter of the makemesh function. For the assignment of the boundary conditions, the boundary condition function can be used with the information of the identification numbers and the general boundary conditions represented by some reserved words of CEMTool FEM Toolbox. In order to construct the solution of the PDE we want to solve, we need the FEM solver funtion of CEMTool FEM Toolbox with the information of the PDE which is represented by the combination of the basic mathematical operator and the reserved words. Since we employ the compiler based technique in the command-mode FEM solve function, compared to MATLAB PDE Toolbox, we need not classify the PDE we want to solve as elliptic equations, parabolic equations, and hyperbolic equations. Finally, the post-processing functions such as "meshplot", "equi-potential plot", and "3D plot" can be used to visualize the solution of the PDE generated by CEMTool FEM Toolbox. The basic structure of CEMTool command-mode FEM Toolbox can be described in Fig 4. In addition, it includes the

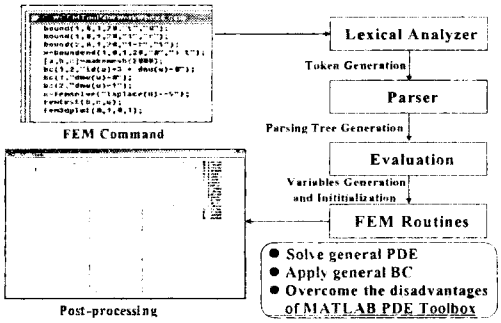

Fig 4. CEMTool command mode FEM package

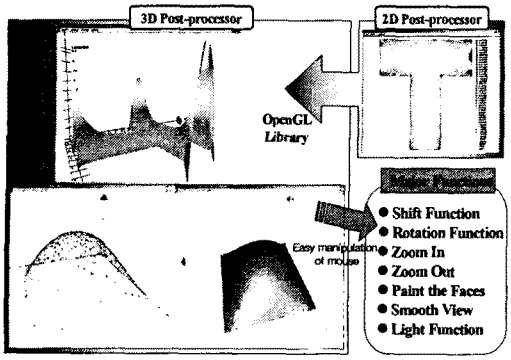

Fig 5. CEMTool 3D FEM Post-processor

dynamic FEM codes for real-time solution of time-varying PDE.

3.2 3D FEM Post-processor

In order to visualize the solutions of PDE in a user-defined region, CEMTool command-mode FEM Toolbox has the powerful plot functions. The meshplot function shows the triangular mesh generated by the makemesh function without the information on height. Also, the equiplot function of CEMTool FEM Toolbox can visualize the equi-potential lines with the information on height in 2-dimensional coordinates. The new feature of the post-processing functions of CEMTool FEM Toolbox is the FEM 3D plot. The FEM 3D plot uses OpenGL library for the enhanced 3D graphic. This function shows the triangular meshes and the equi-potential lines in 3-dimensional coordinates with the information on height. In Fig 5, the 3D plot for Poisson equation in the rectangular region is given. In addition, The FEM 3D plot has the various functions such as rotation, transition, zoom-in, zoom-out, painting the faces, smooth view, and light function.

4. FEM Library for Electromagnetics Analysis

In this section, we present the advanced CEMTool FEM library for practical electromagnetics problems. The detailed FEM electromagnetics library can be summarized as follows:

- If there exists the magnetic flux more than the critical value in magnetic substance, the magnetic permeability is changed and we have to consider the nonlinearity of the magnetic permeability. In CEMTool FEM library, using Newton-Rapson method, we have developed the nonlinear analysis module to guarantee the stability of analysis for magnetic problems.
- We have developed the axis-symmetric analysis module for electric, magnetic dipole problems. Also, this module can consider the nonlinearity of the magnetic permeability when there exists the magnetic flux more than the critical value in magnetic substance.
- We have developed the AC analysis library and the eddy current analysis module. This was implemented by the help of FEM Property Setup program. CEMTool eddy current analysis module can simulate the distortion made by the eddy current in conductor when there exists the change of magnetic fields.

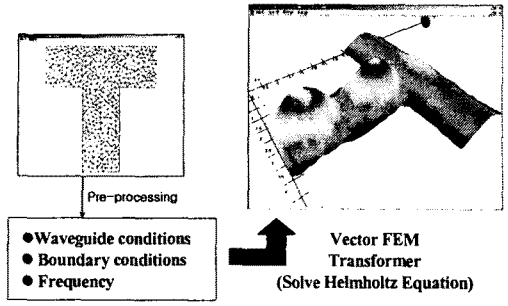

Fig 6. Microwave analysis module

- For easy modification and identification of FEM analysis data, FEM Property Setup was constructed.
- If we use the node based FEM for Helmholtz Equation, we may obtain the spurious solution. In order to remove this spurious solution, using the line based FEM, we have developed the microwave analysis module and it is possible to obtain the analysis result for electronic wave in waveguide as seen in Fig 6. After the mesh generation in waveguide, if we apply the waveguide conditions, the boundary conditions, and the operating frequency, the accurate analysis result can be obtained from CEMTool Vector FEM Transformer.

5. Conclusion

In this paper, we present new GUI based and command-mode based FEM package in CEMTool. Our FEM package can overcome the existing disadvantages of MATLAB PDE Toolbox with the introduction of the compiler based technique, the new mesh refinement algorithm, and the nested dissection method. Due to the these techniques, in contrast to MATLAB PDE Toolbox, we can deal with an equation based PDE and can guarantee the shorter computational time for the finite element analysis, and can refine mesh very effectively. Since CEMTool FEM package has the advanced electromagnetics library, it can be applied to practical analysis and design problems such as motor field analysis and scattering wave analysis.

[Reference]

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

[2] 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.

[3] 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.

[4] 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.

[5] 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.