2D & 3D Finite Element Method Packages of CEMTool for Engineering PDE Problems

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

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 2D & 3D finite element method (FEM) packages for CEMTool. We discuss the detailed structures and the important features of pre-processor, solver, and post-processor of CEMTool 2D & 3D FEM packages. In contrast to the existing MATLAB PDE Toolbox, our proposed FEM packages 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 fast solving technique, our FEM packages can guarantee the shorter computational time than MATLAB PDE Toolbox. Consequently, with our new FEM packages, we can overcome some disadvantages or limitations of the existing MATLAB PDE Toolbox.

Keywords—CEMTool, Finite element method (FEM), Numerical analysis, Partial differential equation (PDE)

I. INTRODUCTION

FINITE element method (FEM) provides a greater flexibility to model complex geometries than finite difference and finite volume methods do. 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 convienently. This has also helped the finite element method become a powerful tool. 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 partial differential equation (PDE) in MATLAB, the MATLAB PDE Toolbox [1] contains basic tools for the study and solution of PDE in two space dimensions (2D) 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 or limitations 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

The authors are with the School of Electrical Engineering & Computer Science, Seoul National University, Seoul, 151-742, Korea. (e-mail: hironaka@cisl.snu.ac.kr)

analysis. (5) It can not be applied to three space dimensions (3D) problems.

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]. 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 [4], [5], [6], [7], [8], [9]. In this paper, we present new FEM packages which overcomes the some existing disadvantages or limitations of MATLAB PDE Toolbox. Our 2D & 3D FEM packages for CEMTool contains 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 [8], [9]. Also, it contains mesh refinement and control algorithms by the dense factor in graphic user interface (GUI) FEM packages. Therefore, 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 algorithms and fast solving techniques. Finally, our FEM packages can applied to various kinds of 3D engineering problems.

In Section 2, GUI based 2D CEMTool FEM package is presented. In Section 3, we discuss command-mode FEM package for CEMTool. In Section 4, we introduce some structures and features of 3D GUI CEMTool FEM package. The conclusion is given in Section 5.

II. GUI 2D FEM PACKAGE OF CEMTOOL

A. GUI 2D FEM Pre-processor

CEMTool GUI 2D 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 tools and dialogs. It is possible to make a partial modification using FEM property setup software of CEMTool FEM Toolbox. These features of GUI 2D FEM pre-processor are described in Fig. 1.

The new feature of our 2D FEM packages is that it is possible to refine the generated mesh using some control
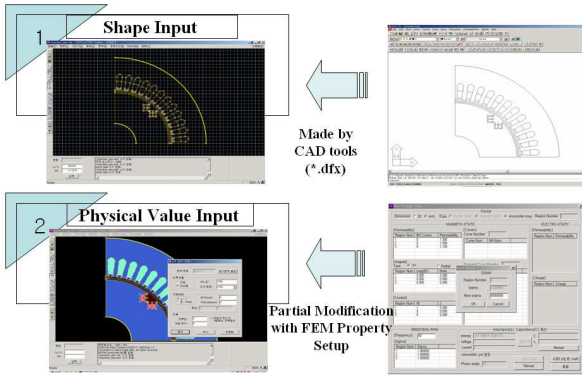
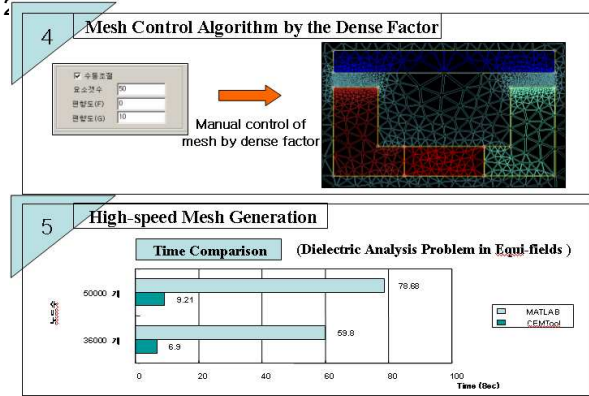Fig. 1.   Some features of GUI FEM pre-processor

Fig. 3.   Mesh control algorithm and high-speed mesh generation

parameters. With new high-performance mesh refinement algorithms, as seen in Fig. 2, we can make an effective mesh refinement by assigning the lines and specifying the regions together with the mesh numbers. Using this technique, more accurate results can be obtained in pre-defined lines and regions.

Another feature of our GUI 2D FEM pre-processor is efficient mesh control algorithm. Consider the field analysis problem for C-shaped permanent magnet. In this problem, because it is important to observe the gap of iron core, we need to divide the mesh of this region more densely. Then, we can apply a manual mesh control algorithm of CEMTool 2D FEM Toolbox by two kinds of dense factor as seen in Fig. 3. For demonstration of high-speed mesh generation, consider the dielectric analysis problem in equi-fields. As seen in Fig. 3, our packages guarantee about 10 times shorter computational time than MATLAB PDE Toolbox for cases of 3600 nodes and 5000 nodes.

B. GUI 2D FEM Solver

GUI 2D FEM solver uses various information produced in GUI FEM pre-processor and performs the finite element analysis. First, it selects the interpolation functions of triangular elements and, using this interpolation functions, determine the relationships between triangular elements. With obtained

shape functions, in order to make the coefficient matrix of the overall system, it assembles the relationship equations between triangular elements. It solves the assembled linear matrix equation and obtains the solutions at every node in a pre-defined region. Our GUI 2D FEM solver employs nested dissection method (NDM) which is one of efficient direct solving techniques. This method requires the minimum memory for fill-in operation. Therefore, it requires the shorter computational time than MATLAB PDE Toolbox. Also, because it contains various kinds of the well-known optimization libraries such as LAPACK library and Intel library, fast performance and accurate solutions are guaranteed. To demonstrate the fast performance of our packages, we consider the dielectric analysis problem in equi-fields. As seen in Fig. 4, it is much faster than MATLAB PDE Toolbox for cases of 3600 nodes and 5000 nodes.
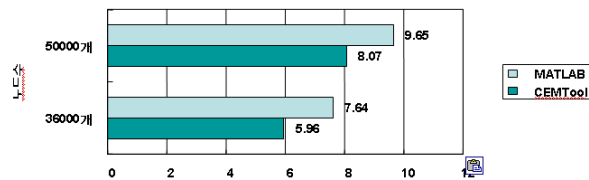

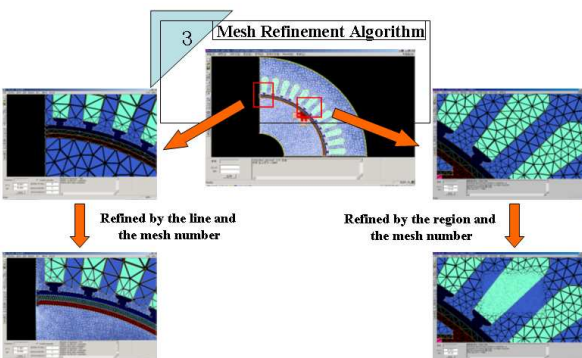Fig. 4.   Time comparisons for assembling and solving

C. GUI 2D FEM Post-processor

CEMTool GUI 2D FEM post-processor represents various information in a pre-defined region after the finite element analysis is completed. To visualize the results of the finite element analysis, it uses information produced in GUI 2D FEM pre-processor and solutions obtained from GUI 2D FEM solver. Fig. 5 shows that CEMTool GUI 2D FEM post-processor can visualize vector potential, flux density, mesh shape, and material distribution 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 many external softwares such as Origin and Excel. If we assign a section in post-processor, the detailed data in the assigned

Fig. 2.   Mesh refinement algorithm

Vector Potential   **Flux Density**

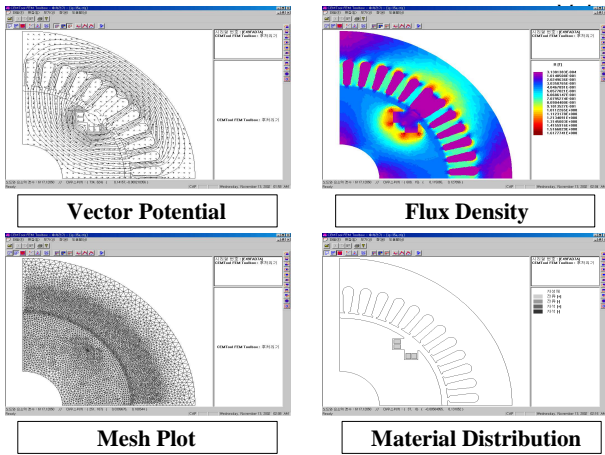**Mesh Plot**   **Material Distribution**

Fig. 5.   Various plots for FEM post-processing

section can be saved in a text file. Therefore, we can use this data in the external softwares as seen in Fig. 6.

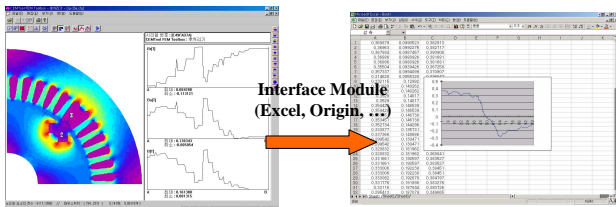Interface Module (Excel, Origin, ...)

Fig. 6.   Interface with the external softwares

III. COMMMAND-MODE FEM PACKAGE FOR CEMTOOL

A. Basic Command-mode FEM Functions

When we need the solutions of PDE represented by equation form in a pre-defined region, it is not appropriate to use the GUI based FEM package. The new FEM package which can solve the combination of the reserved words directly is a good choice for this situation. 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 PDE problems represented by the combination of some reserved words and mathematical operators. Basically, the border function of CEMTool FEM package 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 package by using the information on a maximum number of vertices as the only parameter of the makemesh function. For an assignment of 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 package. In order to construct the solutions of PDE we want to solve, we need the FEM solver

funtion of CEMTool FEM package with the information of the PDE which is represented by the combination of some basic mathematical operators and some reserved words such as "laplace" and "sin". Because we employ the compiler based technique in the command-mode FEM solve function, compared with MATLAB PDE Toolbox, we need not classify PDEs we want to solve as elliptic equations, parabolic ones, and hyperbolic ones. Finally, the post-processing functions such as "meshplot", "equi-potential plot", and "3D plot" can be used to visualize the solutions of the PDE generated by CEMTool FEM package. The basic structure of CEMTool command-mode FEM package can be described in Fig. 7. In addition, it includes the dynamic FEM codes for real-time analysis of time-varying PDE problems.
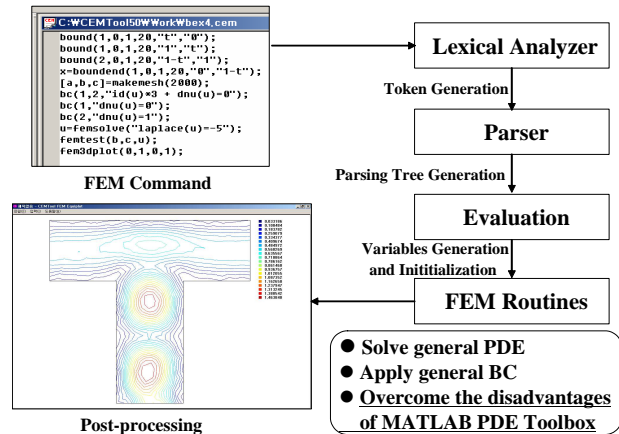

Fig. 7.   CEMTool command-mode FEM package

B. 3D Post-processor of Commmand-mode FEM Package

In order to visualize the solutions of PDEs in a user-defined region, CEMTool command-mode FEM package has the powerful plot functions. The meshplot function shows the triangular mesh generated by the makemesh function without the information on height of the input shape. Also, the equiplot function of CEMTool FEM package can visualize the equi-potential lines with the information on height of the input shape in 2D coordinates. The new feature of the post-processing functions for CEMTool command-mode FEM package is 3D plot. This 3D plot uses OpenGL library for an enhanced 3D visualization. It shows the triangular meshes and the equi-potential lines in 3D coordinates with the information on height of the input shape. In Fig. 8, the 3D plot for Poisson equation in a user-defined rectangular region is given. In addition, it has the various functions such as rotation, transition, zoom-in, zoom-out, painting the faces, smooth view, and light function.

IV. GUI 3D FEM PACKAGE OF CEMTOOL

A. GUI 3D FEM Pre-processor

CEMTool 3D FEM pre-processor requires the 3D modeling information for a real shape that we want to analyze. Also, it
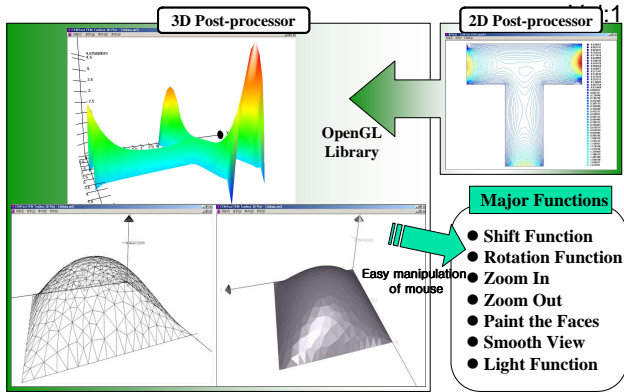
Fig. 8. 3D Post-processor for command-mode FEM package

Fig. 10. 3D mesh generation

needs the information on the characteristic of material, and the boundary condition of the region, the division of elements as inputs. Then, by using input data, it generates the information on elements, nodes, regions, and materials for finite element analysis as output. In CEMTool 3D FEM pre-processor, it is easy to enter simple 3D model shapes. It is possible to make six 3D standard models (hexahedron, sphere, cylinder, prism, cone, and torus) by clicking some icons. In addition, if the interface module of CEMTool 3D FEM pre-processor with CAD softwares is used, we can enter the complex 3D shapes designed by various existing CAD tools such as SolidWorks as seen in Fig. 9. Then, we can generate 3D tetrahedral elements by using the new 3D mesh generation algorithm as seen in Fig. 10. If we choose appropriate options for mesh generation (radius-edge ratio or volume) according to problem features, we can obtain more accurate results. Also, we can input the material values and the boundary conditions with efficient user interfaces.
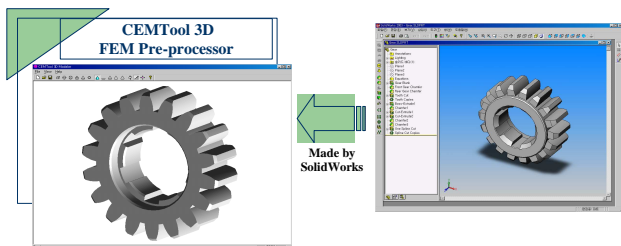
C. GUI 3D FEM Post-processor

CEMTool 3D FEM post-processor visualizes the information on mesh and flux density of 3D shape model as seen in Fig. 11. The deformed position of 3D shape model and the stress in every node also can be represented in 3D mesh plot and 3D flux density plot, respectively. Fig. 12 shows that it is possible to interface with the external softwares such as Origin and Excel.

V. CONCLUSION

In this paper, we present new GUI based and command-mode based FEM packages for CEMTool. Our FEM packages can overcome existing disadvantages and limitations of MATLAB PDE Toolbox with the introduction of the compiler based technique, the new mesh refinement algorithm, the fast solving techniques. Due to these methods, in contrast to MATLAB PDE Toolbox, we can deal with equation based PDEs and can guarantee the shorter computational time for the finite element

Fig. 9. Interface with SolidWorks

Fig. 11. 3D mesh plot and 3D flux density plot

B. GUI 3D FEM Solver

CEMTool GUI 3D FEM solver has almost the same structure as CEMTool GUI 2D FEM solver. In particular, it employs band solving method and skyline solving method for solving the linear algebraic matrix equation efficiently. Skyline solving method is widely used as an optimized method for solving the large scaled linear algebraic matrix equation. Also, because CEMTool GUI 3D FEM solver contains various kinds of the well-known optimization libraries such as LAPACK, we can obtain the solutions very efficiently.
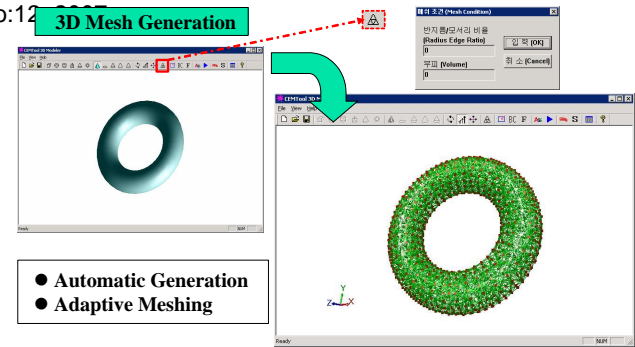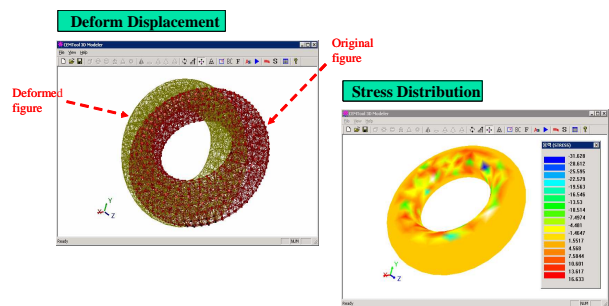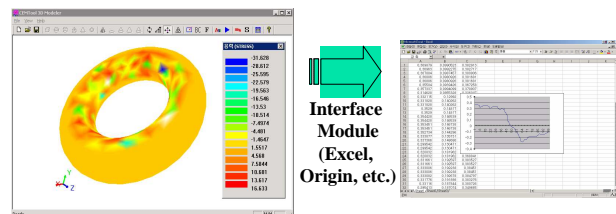

Fig. 12. Interface with external softwares

analysis. In addition, we can refine mesh very effectively and can deal with 3D FEM problems. Because new CEMTool 2D & 3D FEM packages have the advanced electromagnetics & structural analysis libraries, they can be applied to practical analysis & design problems such as motor field analysis, scattering wave analysis, and microstrip patch antenna analysis [10], [11].

REFERENCES

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

[2] 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.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] 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.

[8] 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.

[9] 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.

[10] 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.

[11] J. H. Park, C. K. Ahn, and W. H. Kwon, "On the implementation of microstrip patch antenna analysis package in CEMTool," *In the proceedings of KIEE Annual Conference*, 2004.