UDC 004.9

O. Halushko, I. Farmaga, P. Shmigelskyi
Lviv Polytechnic National University,
CAD Department

THERMAL ANALYSIS OF COMPOSITES IN THE TERMET SYSTEM / ТЕПЛОВЕ МОДЕЛЮВАННЯ КОМПОЗИТІВ У СИСТЕМІ TERMET

The paper describes main approach*es to* thermal analysis of composites in Termet CAE system. Also it describes some principles of FEM implementation in the system.

Key words: thermal analysis, FEM, composite material, CAE.

Описано основні підходи теплового аналізу композитних матеріалів, які реалізовані у системі Термет. Наведено деякі принципи реалізації МСЕ у системі.

Ключові слова: теплове моделювання, МСЕ, композитний матеріал, САПР.

Introduction

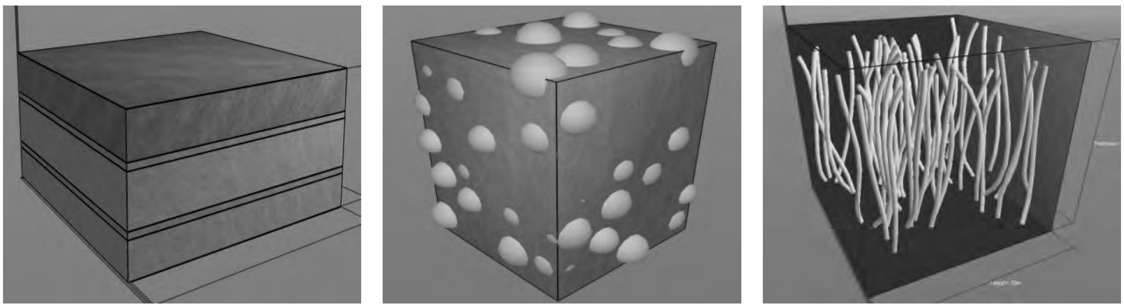
The modern design process requires detailed and precise*d* modeling and analysis tools. It greatly increases quality of design and efficiency of engineers work. Such problems appear in the mechanical construction design, electronics and material design. A real problem of modeling is too complicated to solve them analytically. Finite element analysis is very flexible to solve different physical problems. It allows solving problems of structural mechanics, thermal analysis, fluid flow, electromagnetic fields. This *is* why the method become very popular and is used in most modern CAD/CAE systems in mentioned engineering fields.

The Termet CAE system uses FEM for thermal analysis of composite materials. Some composite materials have a complicated heterogeneous structure, consists of various materials and particles of different shapes. FEM allows *to* solv*e* such kind of the problem. Also the method can take into account mixed boundary conditions and materials with nonlinear properties. It allows *to* solv*e* steady-state and time dependent problems.

Composite materials types

The Termet system allows *to* analyze composite materials of three common types [fig. 1]:

– laminar
– particle-reinforced composites
– fiber-reinforced composites



a *b* *c*

Fig. 1. Composites types: a – laminar; b – particle-reinforced; c– fiber-reinforced

Laminar composite consists of parallel layers of different materials [fig. 1a], cemented together. Particle-reinforced composites [fig. 1b], consist of matrix materials with inclusions of other material particles. Particles may have various and irregular shapes. The Termet system models them by spherical shapes of different sizes. Fiber-reinforced composites [fig. 1c] consist of matrix material with inclusions of fibers. Fibers may be oriented and randomly oriented, continuous or discontinuous. Positions of fibers provide different properties of composites.

FEM Analysis

FEM analysis consists of three main stages: preprocessing, solution and postprocessing.

Preprocessing phase includes such stages as: preparation of the composites geometrical model, boundary conditions settings and generation of the finite element mesh. It is a complex task for an engineer to create a composites model with droplets, because he/she has to set a lot of droplets manually. In order to accomplish this kind of task automatically, in the Termet system we introduced special generators of complex structure [4]. With their help, an engineer is able to build a model of composite with predefined parameters. It makes engineers work much easier and reduces time for geometry development.

In the Termet system iterative and non-iterative methods of final element mesh generation were developed and implemented. Non-iterative methods have a great speed and are more suitable for laminar composites, because laminar composites have a regular structure. On the other hand, iterative methods for mesh generation are more effective for composites with droplets. They have longer work time in comparison to non-iterative methods, but they provide more precise meshes for this type of composites.

A finite element mesh divides geometry on elements with nodes in each of them. Nodes, which are only a point in a space, in general are located at corners of the element or, probably, on edges. In case of 2D analysis or 3D analysis of thin shells, elements, generally are two dimensional, but they can be bent on at small angle in order to reproduce a 3D surface.

In case of solid objects 3D analysis, elements have thickness in all three dimensions. In addition, when geometry, material and boundary conditions are symmetric over an axis, some special types of elements can be used, such as symmetric elements.

For an each node degree of freedom is defined. Solid elements generally have three degrees of freedom for an each node. Rotation is defined as a shifting of group of nodes with respect to other nodes. On the other hand, thin elements have six degrees of freedom for an each node: three for a shifting and three rotations. An addition of rotation degrees allows an evaluation process through a shell, for example, stresses can be evaluated as a rate of rotation for a one element with respect to another. For thermal analysis the only one degree of freedom per node is used.

Mesh development is a complex task in FEA. In the past, in order to achieve an acceptable approximation, nodes should be positioned manually. Modern approach laying on mesh development based on a geometrical model, which can be a wireframe (1) as nodes and curves representing edge, (2) surfaces which bounds the geometry or (3) a solid, which defines material placement. A solid geometry is better, but often it is too complex. From a geometrical point of view, the main rule in FEM is used of simplification every time where it is possible, in order to avoid complex model, which requires highly experienced analysts.

Material properties should be changed with respect to a type of analysis. A linear static analysis, for example, needs a modulus of stiffness, a Poisson coefficient and, probably, a material density. For a thermal analysis temperature properties of materials are obligatory. A definition of boundary conditions for a model with its translation to FEM gives the best results, because the main part of errors is introduced by incorrect defined boundary conditions.

In comparison to preprocessing and post-processing phases, which are generally interactive and needs high level of assistants from analyst, solution phase often is a bash process, but it needs a big amount of computational resources. Main equations are assembled into a matrix form and then they are solved by the application of a numerical method. An assembling process depends not only on the analysis type (for example, static or dynamic) but also on a type and properties of model element, a material and boundary conditions.

In case of linear static analysis, assembled equations are represented in a form of Kd = r, where K is a stiffness matrix, d is a shift vector of node degrees of freedom, and r – a vector of nodes load.

A system of linear algebraic equation is characterized by a sparse banded structure and a big size. It forces to use numerical methods which are applicable in this case.

A postprocessing usually starts from an analysis of previous modeling phases. Most of operations print their results to a log file, which should be analyzed in order to fix probable errors or improve accuracy of an obtained FEA solution.

Obtained analysis results can be displayed in a top of geometrical model, visualized as an animation of numerical series. An important aspect is a way of representation and selecting key elements of a solution, such as a level of mechanical stiffness or a high temperature of a part of the object.

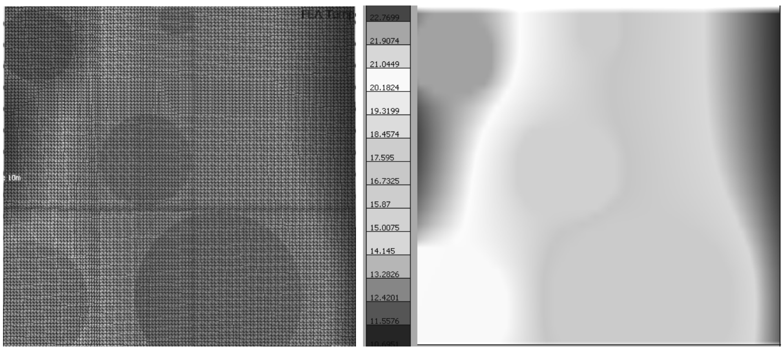
Results also can be introduced in a form of report which includes a description of a modeling process, saved results in a database for future analysis by means of methods of intellectual data analysis.

A finite element mesh can also be displayed for visual representation of geometry decomposition and in order to provide information about a necessity of mesh generation configuration. Also it can be stored in a database for a future usage in another type of analysis, this operation can save time.

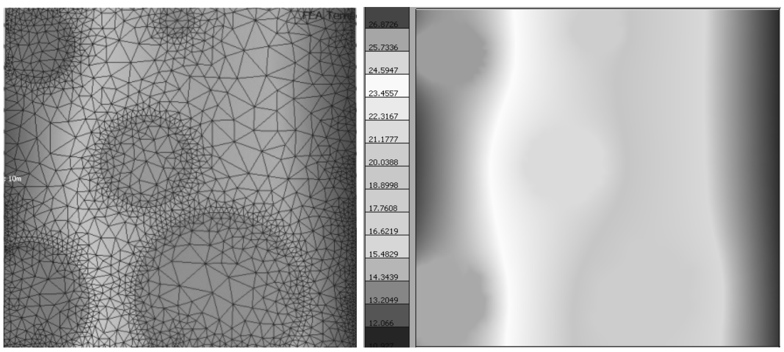
In the Termet systems special thermal field visualizations are implemented. It provides a simple but powerful way to display temperature distribution in an object. Also they can show zones of equal temperature and make a plot of thermal distribution in a user selected point. At the end of the analysis, a user can automatically create a report which includes composite properties, boundary conditions, and a graphical representation of modeling results with a temperature in defined points.

Finite Element Mesh Generators

There are two types of finite element mesh generators: direct and iterative. Direct methods perform only one algorithm iteration. Solution time can be determined by parameters of input data. Iterative methods at first build approximate mesh and then perform improvement and reconstruction until required condition will be satisfied.



a



b

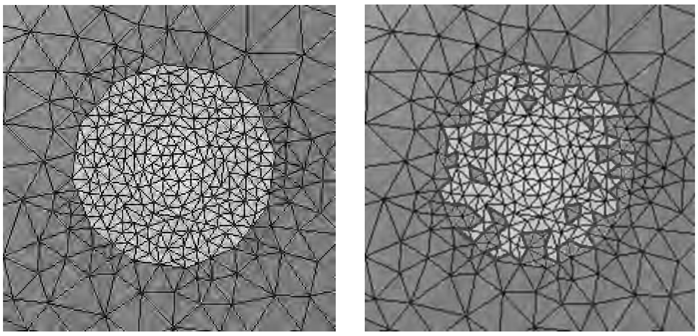
*Fig. 2. Meshes generated by direct (a) and iterative methods (b)
and results of theirs analysis*

The main advantage of direct methods is their performance. But mostly such algorithms developed for the specific type task. This why those methods are not universal and different task required different meshing algorithms.

Iterative methods allow *to* build meshes for various geometric forms and tasks. Those algorithms mostly are versatile and they can be used for various types of tasks. The mesh generation process may take a much more time in comparison with direct methods. But such methods can be applied to various types of geometric, where direct methods are not applicable.

Termet system implements both methods [1, 2]. Both methods are important, because system analyzes different types of composites with completely different geometries. Laminar composites have simple regular structure. For this type of composite the direct method is preferred. Composites with inclusions have complicated heterogeneous structure. The iterative methods are applicable there. Direct methods don't provide good accuracy for those geometric, but they can be used to achieve a fast approximate solution of the task. Fig. 2 shows meshes generated by direct and iterative methods and results of their analyses.

Between matrix material and inclusion may not exists a condition of perfect thermal contact. Due *to* technological processes in contact area may occur voids or mutual penetration of materials. To simulate such phenomena generators are able to build special transition layer around inclusions [3, 4] (fig. 3). Generators have settings of properties of transition layers for each type of inclusions. This is set of materials and theirs concentrations, the transition layer consists of. Size of transition layer may be constant of depends on inclusion size. If the transition layer consists of matrix material and material for inclusion, this can simulate shape warping of inclusion or mutual penetration of materials (fig. 2b).



a *b*

Fig. 3. Translation layer a – one material; b – mix of materials

Methods for solving systems of linear algebraic equations

The system of linear algebraic equations which appears as a result of FEA is characterized by sparse a banded structure of a big size. This constrains should be taken into consideration in a solver development process in order to obtain a precise solution *at* a reasonable time.

Also, storing the coefficient matrix is an independent problem. Its size makes almost impossible keeping this matrix in a simple way, even on modern systems. So, it is important to use special data structures which combine small size and provides a reasonable access time for a solution algorithm.

A decision about the solution method is based on achievements in numerical methods and on the experience from usage of such methods for similar tasks. In this case we can include the Gaussian elimination method with nonzero element access modifications, the successive over relaxation method and the conjugate gradient method.

The Gaussian elimination method is a well-known stationary non iterative method which ends after a well-defined number of steps and its accuracy depends only on a precision of floating point operations. A defined number of steps and simplicity of this method ensure some advantages of this method in comparison with others. But in case of large size systems which are ill-conditioned, introduced errors corrupt the solution and reduce an application sphere for this method.

The successive over relaxation method is an iterative and stationary method. It makes a successive approximation of the solution in the iterative way [5, 6]. This method is a modification of the Gauss-Seidel method. A modification is an introduction of the relaxation coefficient, which optimal value ensures faster approximation. Determination of an optimal value for this coefficient is not a trivial task and requires complex computation or usage of heuristic techniques [7].

The Conjugate Gradient method is non-stationary iterative method. It imposes constraints on the matrix, such as positive-defines and symmetrical. In this method, a solution is approximated by a combination of conjugate vectors, where each next is based on a previous and on an error on a previous approximation [8]. Caused by a step error correction, this method shows high start approximation.

It was accomplished a research which compares a solution time for the Gauss elimination as a representative for a non-iterative, and the conjugate gradient as a representative for iterative methods. The main target of this research was a discovering if more complex conjugate gradient method is able to achieve a smaller solution time in comparison with a simple Gaussian elimination for a systems, which are obtained as a result of FEA. Solutions were obtained for different composites with different mesh parameters and research results are presented in a table 1, where GE means Gauss elimination and CG means conjugate gradient..

Table 1

№#	Project	Precision	Min angle	Min. edge	Max. edge	Elements	Band	Nodes	Time, s	Method
11	Droplet_3	3.65874e-19	25	0.0008	0.01	48882	1667	9544	400.976	GE
22	Droplet_3	3.65874e-19	25	0.0008	0.01	48882	1667	9544	125.045	CG
33	Fiber_1, B_1	1e-17	20	0.004	0.01	54885	2496	11128	1052.33	GE
14	Fiber_1, B_1	1e-17	20	0.004	0.01	54885	2496	11128	158.778	CG
15	Fiber_2	1e-11	20	0.001	0.01	51108	1071	8755	251.935	GE
16	Fiber_2	1e-11	20	0.001	0.01	51108	1071	8755	126.117	CG
17	Fiber_2, B_2	1e-11	20	0.001	0.01	51108	1071	8755	256.469	GE
18	Fiber_2, B_2	1e-11	20	0.001	0.01	51108	1071	8755	135.876	CG
19	Fiber_2, B_2	1e-15	25	0.0008	0.01	29292	839	5537	103.86	GE
110	Fiber_2, B_2	1e-15	25	0.0008	0.01	29292	839	5537	69.745	CG

Conclusion

In this work are presented different types of composite materials implemented in the Termet system, provided the characteristics of their structure and assumption which was used to build models of such composites.

Provided existing approaches to a generation of a complex geometry and their implementation and it was demonstrated an approach to modeling of a particular aspects of technology.

Described by us main stages of a FEA and implementations of the in the Termet system, provided examples usage of the different mesh generator in the dependence from the geometry, their advantages and disadvantages.

A decision process for a solver of a system of linear algebraic equations from which is obtained as a result of FEM is explained and showed research results of their efficiency for the FEA tasks in thermal analysis.

1. Farmaga I. Regular and Adaptive Meshing Algorithms for Modeling of Spherical Inclusions by Finite Element Method / I. Farmaga, M. Lobur, P. Shmigelskyi [et al.] // Proc. of the XI-th Intern. Conf. Modern Problems of Radio Engineering, Telecommunications and Computer Science (TCSET'2012). – Lviv-Slavske, 2012. – P. 505–507. 2. Farmaga I. Finite Element Meshing of Composites with Spherical Inclusions / I. Farmaga, M. Lobur, P. Shmigelskyi [et al.] // Machine Dynamics Research. – Poland, Warsaw University of Technology, 2011. – Vol. 35, No. 3 – P. 13–21. – ISSN: 2080-9948. 3. Farmaga I. Thermal Analysis Methods for Automated Design of Composite Materials in TERMET Software / Ihor Farmaga, Oleh Matviykiv, Mykhaylo Lobur, [et al.] // Micromechanics and Microsystems Europe Workshop 2012 (MME 2012), Ilmenau, Germany, 2012. – Session D – D 07. 4. Farmaga I. Generation of Particle Reinforced Composites Models in TERMET Software / Ihor Farmaga, Krzysztof Kurzydlowski, Marian Lobur [et al.] // Proceedings of the XX Ukrainian-Polish Conference CAD in Machinery Design - Implementation and Educational Issues. – Lviv, 2012. – P. 42–43. 5. Y. Saad, "Iterative Methods For Srapse Linear Systems," Society for Industrial and Applyed Mathematics, pp.92-95, 2003. 6. D. M. Young, "Iterative Methods For Solving Partial Difference Equations Of Elliptic Type," Harvard University, pp.66–69, 1950. 7. S. Yang, K. Gobbert, "The Optimal Relaxation Parameter for the SOR Method Applied to a Classical Model Problem," Department of Mathematics and Statistics, University of Maryland, pp.15–18, 1950. 8. Saad, Yousef. "Iterative methods for sparse linear systems", Philadelphia, Pa.: Society for Industrial and Applied Mathematics. p. 195. 2003