ELSEVIER

Finite Elements in Analysis and Design 38 (2002) 179–192

FINITE ELEMENTS IN ANALYSIS AND DESIGN

www.elsevier.com/locate/finel

Integration of finite element analysis and optimum design on gear systems

Chien-Hsing Li[a], Hong-Shun Chiou[a], Chinghua Hung[a,*], Yun-Yuan Chang[b], Cheng-Chung Yen[b]

[a]Department of Mechanical Engineering, National Chiao Tung University, 100, Ta Hsuch Road, Hsinchu, Taiwan
[b]Industrial Technology Research Institute, Hsinchu, Taiwan

Abstract

This work establishes a batch module called "integration of finite element analysis and optimum design" by taking gear systems as testing examples. This batch module consists of I-DEAS, ABAQUS/Standard and MOST software, which serve as the preprocessor, the numerical solver and the optimizer, respectively. A simple and practical method was developed, by which this module was enabled to search for contact nodes and elements and to automatically define the contact surfaces for contact analysis. A simple gear-pair system and a complete planetary gear system were successively used as testing examples for this integrated module. The module will automatically construct the geometrical model, analyze contact stress and solve for the optimal solutions when gearing parameters are input. The results are expected to enhance the technology of gear system design. © 2002 Elsevier Science B.V. All rights reserved.

Keywords: Finite element analysis; Optimum design; Planetary gear system; Contact stress analysis

1. Introduction

Most engineering problems are too complicated to be solved using simple formulae. Modern engineers usually use all kinds of software packages to facilitate various phases of research and development. Unfortunately, there exist no or few interfaces between these packages, which fact makes the work of integration very difficult. For instance, finite element software is extensively used in engineering problems, such as structural analysis, thermal conductivity and

* Corresponding author. Tel.: + 886-3-5712121 ext 55160; fax: + 886-3-5720634.
E-mail address: chhung@cc.nctu.edu.tw (C. Hung).

PII: S0168-874X(01)00057-9

electro-magnetic problems. Optimum design packages aid solving for the optimal design variables. Some commercial programs have managed to combine these two systems into one batch module, but their applications are very limited. Many practical design problems, especially those which take the geometrical parameters as design variables, still require engineers to handle repeatedly different CAD, CAE, and optimization software.

Consider the design of gear systems. When gear profile is initially chosen, CAD software is used to construct a geometrical model. This model is then converted into a finite element mesh for stress analysis. If the results of the analysis are not satisfactory, due to a strength failure sub-optimal design, the profile of the gears must be modified and the geometrical model and the finite element mesh must be rebuilt. These tedious procedures are then repeated until optimal solutions are obtained. In the present study, a batch module called "integration of finite element analysis and optimum design" was developed to shorten the above procedure. The shape of the gears can be changed using this batch module, and the optimal solution can be obtained automatically. Accordingly, engineering designers no longer need to repeatedly rebuild the model when the geometrical shape must be changed. Design is thus more efficient and prospects for research into gear systems for academics and industry are enhanced.

Three software packages, I-DEAS, ABAQUS/Standard and MOST were used in this module for preprocessing, finite element analysis and optimization, respectively. Interfacial programs were constructed to connect these three packages. In addition, searching algorithms for contact nodes and elements were also developed and integrated into this module to address the contact problems that are essential in the analysis of gear system.

Two gear systems were taken as testing examples to validate the use of this integrated module. The first was a simple gear system consisting of one pinion and one gear, in which the pressure angle was chosen as the design variable and minimum contact stress as the objective function. The other was a complete planetary gear system including one sun gear, three planetary gears and one stationary ring gear. The optimal gap tolerance between the planetary gear and the pins was obtained using the developed module.

2. Components of the integrated module

Fig. 1 shows contents of the integrated module with the tasks of all components. Details of these components are described in the following paragraphs.

Integrated design engineering analysis software (I-DEAS) is an integrated package of mechanical software tools. This software has been designed to facilitate different concurrent engineering designs and analyses. I-DEAS was adopted because of its capability to create a complex geometrical model with a free-mesh function and good compatibility with other components of the module. The export function of I-DEAS was used for exporting the information of the finite element model into the type of input file required by the ABAQUS/Standard [1].

Although I-DEAS is also equipped with an FE solver and optimizer, it does not support sliding contact analysis. Therefore, I-DEAS was involved only as a preprocessor in the developed module.

ABAQUS/Standard is a general finite element software developed by Hibbitt, Karlsson & Sorensen, Inc. The package has great potential in many applications, including both linear and nonlinear problems. In this work, its powerful capability was used to address sliding contact

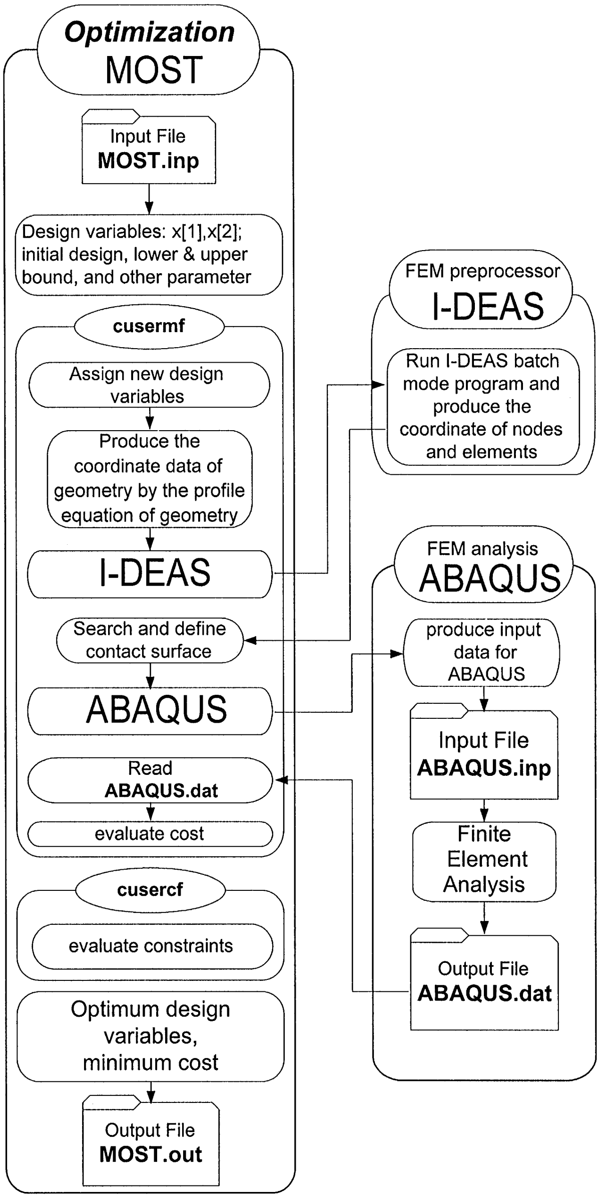


Fig. 1. Components of the integrated module and its interface.

problems without point-to-point contact constraint [2]. Briefly, the package was used to solve the finite element model and generate analyzed results such as stress and strain distributions that are consequently used in the optimization process.

Multifunctional optimization system tool (MOST), developed in C language, is used to solve multi-objective optimization problems with both continuous and discrete design variables [3]. The

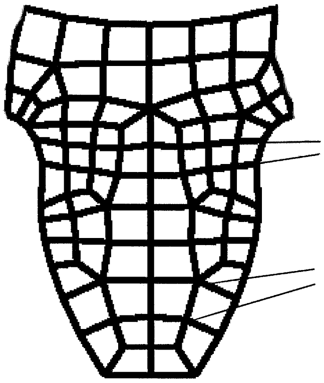


Fig. 2. Rule to judge whether the nodes lay on contact surface.

MOST code contains three main modules for solving problems with continuous variables, non-continuous variables, and multi-objective.

In this work, only the module for solving continuous variables problem was used and the sequential quadratic programming (SQP) method was selected for its accuracy as a single objective optimizer. This primary module has been run through many test problems, with those results indicating that the module handles large-scale engineering optimization problems with excellent convergent characteristics [4].

A further program was written to search automatically nodes and elements related to contact surfaces. This program was used to provide necessary contact information for the finite element computation when the slide line contact option in the ABAQUS/Standard was activated. The basic algorithm and procedures of this program are explained as follows.

(1) Searching contact nodes and elements. Contact nodes and their corresponding elements can be found by checking and comparing all pairs of consecutive nodes in the element connectivity data, giving that two neighboring nodes along the contact surface can only belong to one element, while two neighboring nodes inside the mesh can belong to two or more elements as shown in Fig. 2.
(2) Sorting orders of contact nodes and elements. Two neighboring elements on the contact surface share only one contact node. If a specific contact node derived in the first procedure was specified, it must belong to two neighboring contact elements. The next contact node number can then be determined when the sequential direction was specified. Follow above rule, all of the neighboring contact nodes and contact elements can be obtained and sorted according to the order of neighboring sequence.

3. The flow of the integrated batch module

The components described above were integrated into a batch module as shown in the flow chart (Fig. 3). Detailed execution procedures follow:

Step 1. Input parameters into the profile equation and generate the coordinate data of the profile nodes. The free mesh function of I-DEAS is used to generate the finite element model with these

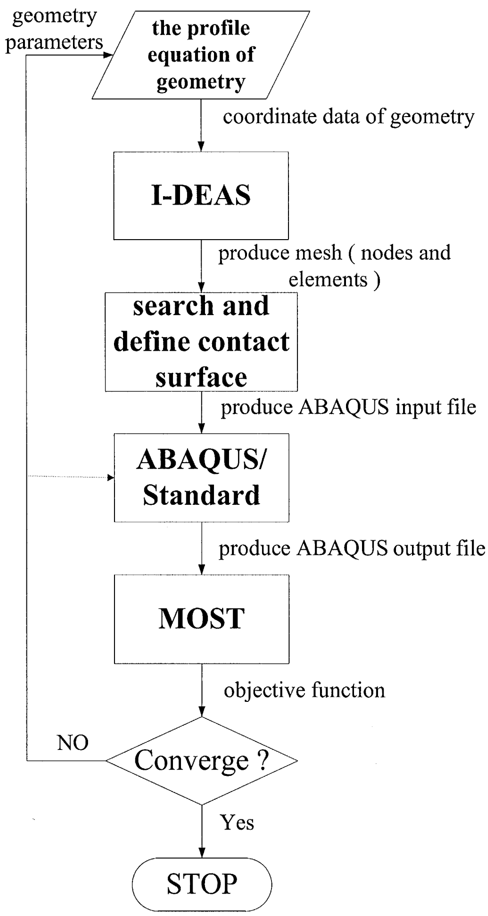


Fig. 3. Flow chart of batch execution module on the integration of finite element analysis and optimum system.

coordinate data, and the data of the nodes and elements are output into a file in I-DEAS command format. Execute the program file and export the finite element model into an ABAQUS/Standard input file.

Step 2. Search contact nodes, elements and define contact surfaces according to the requirement of ABAQUS/Standard. Write these contact data together with material properties and boundary conditions into the complete ABAQUS/Standard input file.

Step 3. Execute ABAQUS/Standard with the input file and solve finite element model to produce stress, strain and other analysis data in an output data file.

Step 4. Execute MOST to evaluate the objective function according to the finite element output with certain specified constraints. If the objective function converges, stop the batch program and output optimal design values, otherwise, modify the geometric parameter and execute another loop of the entire procedure.

Interfacial programs written in C were used to integrate all components according to the entire flow into one batch module program. The program source code was compiled and run on

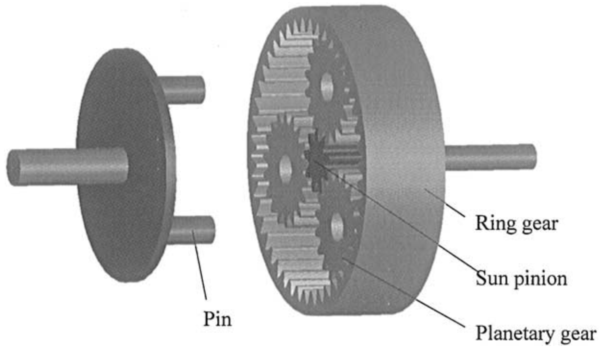


Fig. 4. Diagram of planetary gear system.

a personal computer with the Microsoft NT operating system. Two different gear systems were then taken as testing examples to verify the usefulness of this module.

4. Testing examples

First, a simple gear-pair including one pinion and one gear was set up to show that the integrated batch module work well. In this case, the pressure angle was taken as the design variable and the contact stress was taken as the objective function. The range of the pressure angle was limited between 18° and 24° to satisfy industrial specifications. The batch module was executed to seek an optimal pressure angle at which the equivalent contact stress is minimal.

The second testing case was extended from the above system by adding one stationary ring gear to form a simplified analysis model of a planetary gear system. This model represents a complete planetary gear system that consists of one sun gear, three evenly spaced planetary gears and one ring gear as shown in Fig. 4. The optimal gap tolerance between the pin and the interior diameter of planetary gear is investigated in this simulation case. That is to say, an optimal gap size was sought to prolong the designed fatigue life of the planetary gear system.

4.1. The profile equation of gears

An cross section of a 3D helical gear in the normal direction was used to represent the 2D spur gear used in this analysis [5]. The profile equation of a helical gear with involute shape teeth was generated by a rack cutter as follows.

$$A_1 = l_p \cos\psi_n - A_0 + r_1, \tag{1}$$

$$B_1 = (l_p \cos\psi_n - A_0)\cot\psi_n \sin\lambda_f, \tag{2}$$

$$x = A_1 \cos\phi_1 + B_1 \sin\phi_1, \tag{3}$$

$$y = A_1 \sin\phi_1 - B_1 \cos\phi_1 \tag{4}$$

Table 1
Parameters of rack cutter and constants on equation of gear tooth surfaces

Rack cutter on sun gear	$l_p = 0.5 \sim 2.3$, $\Delta l_p = 0.3$
Rack cutter on planetary gear	$l_p = 0.4 \sim 2.5$, $\Delta l_p = 0.3$
Rack cutter on ring gear	$l_p = 0.0 \sim 2.4$, $\Delta l_p = 0.6$
Module (m)	1.25
Diametrical pitch (P_n)	0.8
Pressure angle (Ψ_n)	24.5°
Addendum	1.25
Dedendum	1.25

Table 2
The specification on sun gear, planetary gear and ring gear

	Sun gear	Planetary gear	Ring gear
Number of teeth	12	18	12
Pitch diameter (mm)	15	22.5	15
Backlash (mm)	0.025	0.0	0.025
B_0 ($\pi/4P_{n)}$ (mm)	0.95675	0.3125π	0.3125π

$$z = (A_0 \tan\psi_n - l_p \sin\psi_n)\cos\lambda_f + \left(\frac{A_0}{\cos\psi_n \sin\psi_n} - \frac{l_p}{\sin\psi_n}\right)\tan\lambda_f \sin\lambda_f + \frac{B_0}{\cos\lambda_f} + r_1\phi_1 \tan\lambda_f, \tag{5}$$

where l_p and A_0 represent the parameters of rack cutter, and parameters r_1, ψ_n, λ_f and ϕ_1 represent the pitch circle radius of the gear, the normal pressure angle, the lead angle and the rotational angle of the gear during the cutting process, respectively.

When the z-axis coordinate is set to zero and the lead angle of helical gear λ_f is set to 75°, the resulting cross section of the helical gear presents the profile of a spur gear, according to the above formulae. The equation of meshing is expressed as follows.

$$\phi_1 = \left\{(A_0 \tan\psi_n - l_p \sin\psi_n)\cos^2\lambda_f/\sin\lambda_f + \left(\frac{A_0}{\cos\psi_n \sin\psi_n} - \frac{l_p}{\sin\psi_n}\right)\sin\lambda_f + \frac{B_0}{\sin\lambda_f}\right\}\Big/ r_1. \tag{6}$$

The formulae (1), (2), (3), (4) and (6) completely define a pair of spur gears with involute teeth.

4.2. Parameters of the planetary gear system and material properties

Tables 1 and 2 list the data concerning the rack cutter and the gears in a planetary system, which are needed in the profile equation to generate the gear tooth surface for a typical planetary gearbox with reduction ratio one to five.

Table 3
Material properties for gears

Material	S45C
Elastic modules (MPa)	202.0E3
Poisson ratio	0.2875
Density (kg/mm^3)	74.0E-7
Yield strength (MPa)	490

Carbon steel S45C is usually used for the gears in a typical gearbox with a planetary gear system. The material properties of S45C are listed in Table 3.

4.3. Finite element models, load controls and constraints

Creating an FE model requires first selecting a suitable element type and then generating the corresponding mesh system. In the present study, two-dimensional 4-node bilinear elements were used for all gears. A few rigid beam elements (BEAM element in ABAQUS/Standard) were connected between nodes on the interior circle of the pinion and the center of the interior circle. The multi-point-constraints (MPC function of ABAQUS/Standard) were used to specify the translation and rotation of the rigid beam elements [2].

A linear torsion spring was defined on the center of the pinion to make the force boundary conditions more realistic. A rotational angle could be input on the torsion spring and the required input torque in the simulation could thus be specified. In this study, the input torque was set to 1234 N mm in both test cases. Different load conditions and constraints are detailed below.

(i) The simple gear-pair system. For the simple gear-pair system, the constraint on the gear was defined in the same way as it was set for the pinion. Both the pinion and the gear were constrained to rotate about their axes only and there were no translations. During the first step of the simulation, a counterclockwise torque was specified on pinion and the reference node of the gear was fixed. In the second step, a counterclockwise rotation of 0.25 radians was defined on the rotational axis of the gear, and the gear-pair system exhibited realistic behavior during the simulation.
(ii) The planetary gear system. More complicated and realistic conditions were applied in the second case of the planetary gear system. An additional pin within the planetary gear was defined as a rigid body and was referenced to its own center (Fig. 5). The sun gear was constrained to rotate about the axis through its center and the ring gear was stationary. When the reference node of the pin was specified to rotate about the center of sun gear, the relative motion of the planetary gear and the sun gear could be achieved by the contact conditions set between pin and gear and between gear and gear.

4.4. Optimization

In the case of the simple gear-pair system, MOST was used to solve for the optimal pressure angle associated with the minimum von Mises contact stress that had significant effect on the

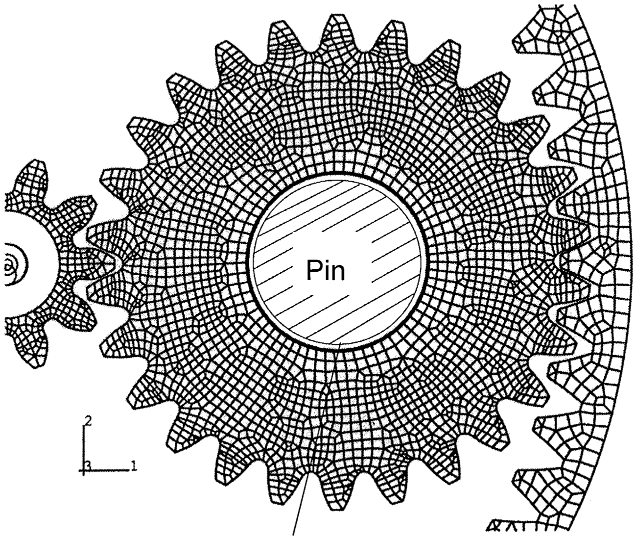


Fig. 5. Relationship between the pin and the planetary gear.

fatigue of the gear tooth surface. Let $F(\boldsymbol{\Psi}_n)$ be a function of the pressure angle corresponding to the contact stress of the gear-pair. The optimization problem in the usual terms is then stated as follows.

$$\text{Minimize}\, F(\psi_n) = \text{Min}\{\text{Max}(\sigma_f)\} \tag{7}$$

$$\text{subject to} \quad 18^\circ \leqslant \psi_n \leqslant 24^\circ, \tag{8}$$

where σ_f stands for the von Mises contact stress.

In the second case of the planetary gear system, optimal gap tolerance between the planetary gear and the pin was chosen as the object. But during analysis, the inner radius of the planetary gear was taken as the design variable during optimization because the diameter of the pin was kept to a constant value (5 mm). Let $F(r)$ be the function of the radius representing the contact stress in the planetary gear system. The optimization problem is then,

$$\text{Minimize}\, F(r) = \text{Min}\{\text{Max}(\text{maximum of}\, \sigma_f - \text{minimum of}\, \sigma_f)\} \tag{9}$$

$$\text{subject to} \quad 5\,\text{mm} \leqslant r \leqslant 5.4\,\text{mm} \tag{10}$$

where r represents the radius of the interior circle of the planetary gear and ó$_f$ represents the von Mises equivalent stress. Considering the above objective function allows the fatigue life of the planetary gearbox to be prolonged. Distortion energy theory can predict failure accurately from experimental result [6]. According to this theory, the fatigue life is extended when the variation between maximal and minimal von Mises stresses for a meshing gear system is smaller.

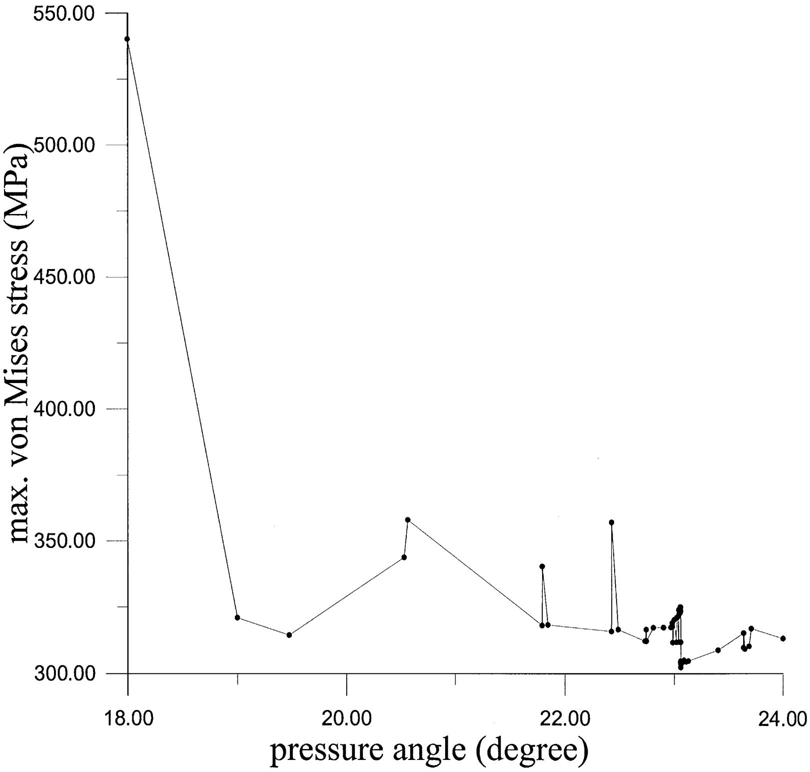
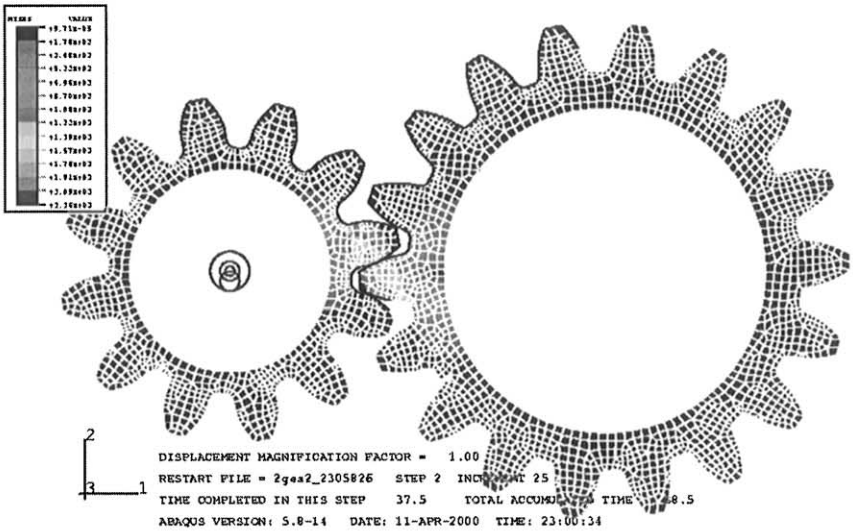


Fig. 6. Relationship between the pressure angle and the contact stress in the gear pair system.

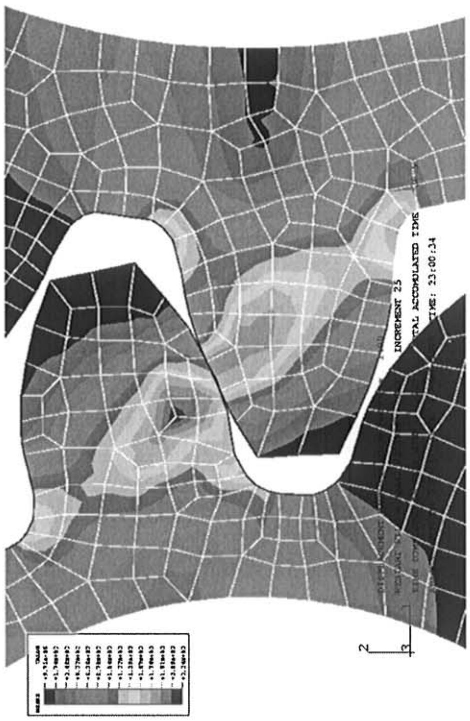
4.5. Results

Good results are obtained for either test examples when the flow procedures of the integrated batch module are followed, starting with the coordinate data calculation, through mesh generation and contact stress analysis, to the optimal solution. In the simple gear-pair system, the optimal solution was obtained after 85 iterations. The minimized contact stress was 302.5 MPa when the optimized pressure angle equaled 23.6°. The relationship between contact stress and pressure angle is shown in Fig. 6. Fig. 7 shows the distribution of von Mises stress in the optimized gear-pair system.

In the planetary gear system, the results from the batch module showed that when the inner radius of the planetary gear is 5.1 mm (gap tolerance equals to 0.1 mm), the smallest stress variation of 330.24 MPa was obtained. The relationship between the stress variation and the radius of the interior circle of the planetary gear is shown in Fig 8. The stress contours of the optimal solution for the planetary gear system are shown in Fig. 9. The maximum von Mises stress is 331.40 MPa and the minimum von Mises stress is 1.16 MPa.



(a) The global distribution of contact stress



(b) Local distribution of contact stress

Fig. 7. Distributions of stress on optimized gear pair system.

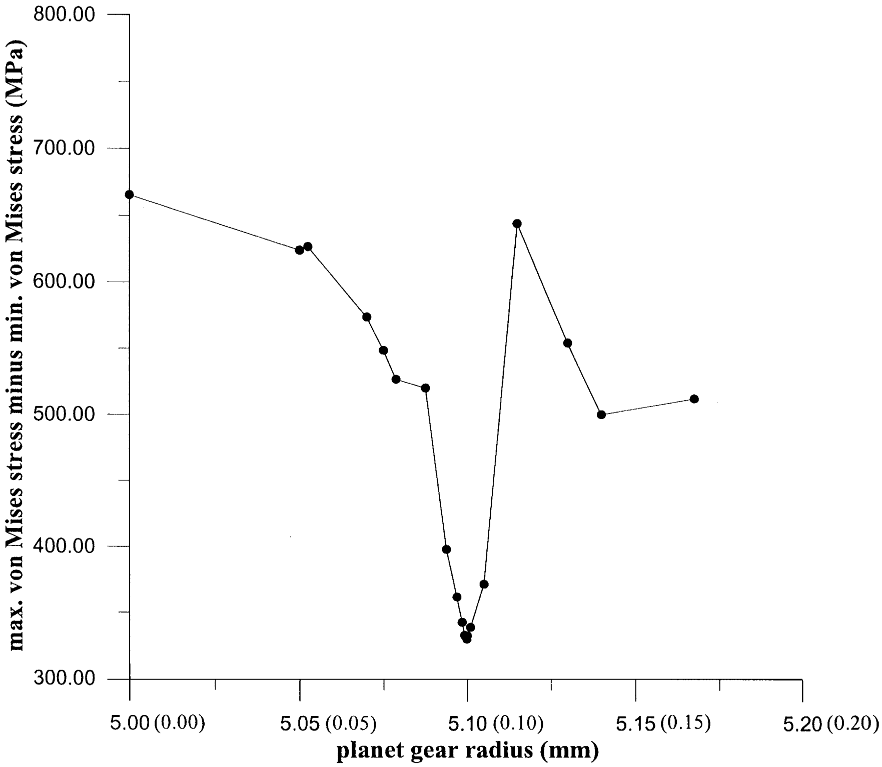
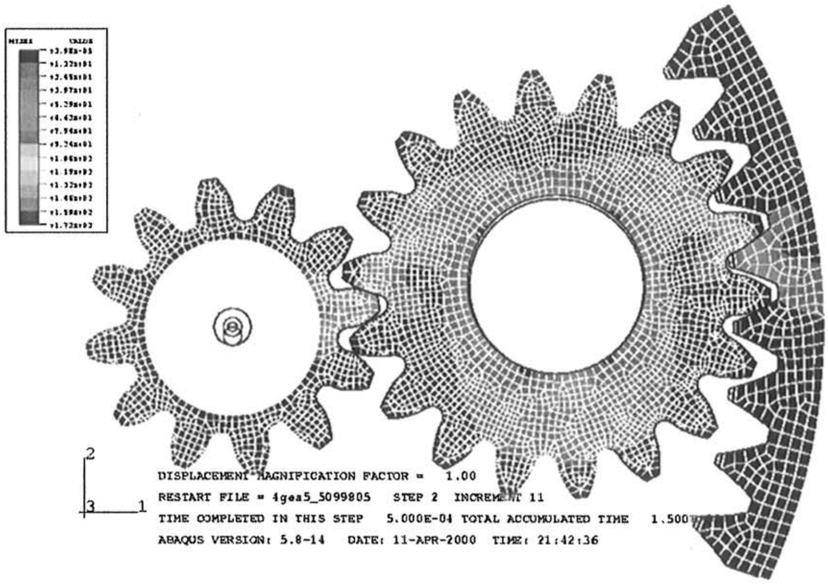


Fig. 8. Relationship between the interior circle radius of planetary gear (gap tolerance in parenthesis) and the variations of contact stress.

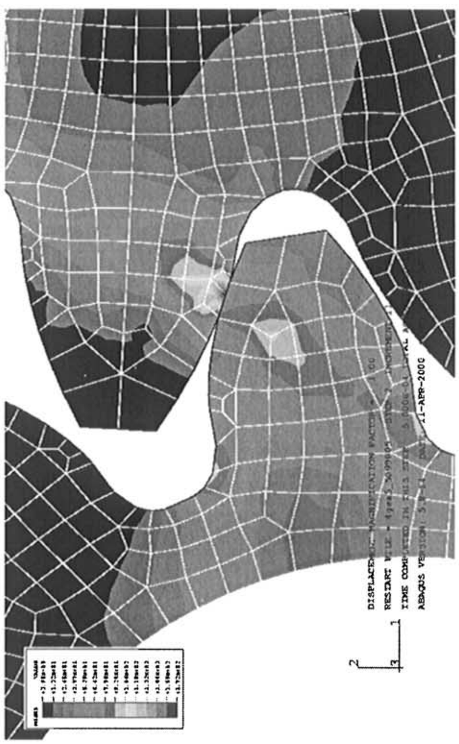
5. Conclusions

The major advantage of applying the "integration of finite element and optimum system" module to the gear systems is the high efficiency of the design procedure. In the present study, geometrical parameters were selected as design variables, and the gear tooth profiles can be modified by computer with this module rather than by manual operations. The test examples described above verified that this module saves time and resources. A summary follows.

1. It is hard work to generate manually the mesh system in a special geometric pattern because of the complexity of the gear shape. In this paper, the function of free mesh in I-DEAS was adopted by specifying the characteristic length of a typical element. A suitable mesh density was thus achieved and preprocessing these two dimensional problems took 3–5 min.
2. This integrated module was applied to two-dimensional problems only. It will be possible to extend the module to solve three-dimensional problems such as helical gear systems.
3. The above tests took pressure angle and the gap tolerance between gears and pins as design variables. Only contact stress was considered although these variables may also affect the



(a) The global distribution of contact stress



(b) Local distribution of contact stress

Fig. 9. Distributions of stress on optimized planetary gear system.

transmission function or produce additional noise and vibrations. Further investigations should be conducted and the algorithm for multi-objective function in MOST should be included for more realistic and accurate analyses.

4. Continuous design was applied to the gear-pair system. However, the pressure angle of 23.6° may not be practical in industry. Discrete design might better be used for the future research.

Acknowledgements

The authors would like to thank the Industrial Technology Research Institute (ITRI) for financially supporting this research under Contract No. 893K42AD2.

References

[1] M.H. Lawry, I-DEAS Master Series STUDENT GUIDE, Structural Dynamics Research Corporation, 1994.
[2] Hibbitt, Karlsson & Sorensen, ABAQUS Manual, Hibbitt, Karlsson & Sorensen, Inc., 1992.
[3] C.H. Tseng, W.C. Liao, Yang, T.C., MOST User's Manual, Technical Report No. AODL-91-1-01, Department of Mechanical Engineering, National Chiao Tung University, Version 1.0, 1991.
[4] C.H. Tseng, J.S. Arora, On implementation of computational algorithms of optimal design 1: preliminary investigation, Int. J. Numer. Methods Eng. 26 (1988) 1365–1384.
[5] C.B. Tsay, Helical Gears with Involute Shaped Teeth: Geometry, Computer Simulation, Tooth Contact Analysis, and Stress Analysis Trans, ASME, J. Mech. Transm. Autom. Des. 110 (4) (1988) 482–491.
[6] J.E. Shigley, C.R. Mischke, Mechanical Engineering Design, 5th Edition, McGraw-Hill, New York, 1989.