한국전산구조공학회
1998년도 봄학술발표회 논문집

PATRAN 데이타베이스를 기반으로 한 스페이스 프레임의 통합설계시스템

Space Frame Integrated Design System based on PATRAN Database

이 재 홍*　　이 주 영**
Lee, Jaehong　　Lee, Joo-young

ABSTRACT

To design a space frame structure by the conventional method is not easy in practical sense since it is generally a three-dimensional complicated form, and stability and nonlinear problems are not easily checked in the design process. This paper describes two modules, the Model Generator which is based on PATRAN user interface that enables users to generate a complicated finite element model; the Optimum Design Module which analyzes output results of analysis program, and designs members of a space frame. The Model Generator is based on PCL while C++ language is used in the Optimum Design Module. Structural analysis is performed by using ABAQUS. All of these modules constitute Space Frame Integrated Design System.

The Core of the system is PATRAN database, in which the Model Generator creates information of a finite element model. Then, PATRAN creates input files needed for the analysis program from the information of the finite element model in the database, and in turn, imports output results of analysis program to the database. Finally, the Optimum Design Module processes member grouping of a space frame based on the output results, and performs optimal member selection of a space frame. This process is repeated until the desired optimum structural members are obtained.

1. INTRODUCTION

This paper presents a hybrid method which is applicable to the integrated design system of a space frame structure. PATRAN(1) and ABAQUS(2) are used as application software in this system. PATRAN creates finite element data of a space frame, transforms the data of PATRAN database into the data of a finite element application, restores the data of the finite element application to the data of PATRAN database, and displays the analysis result.

PATRAN database contains two basic modules; Model Generator which generates all the necessary information including finite element mesh and load data based on FORMEX algebra(3), and Optimum Design Module which optimizes members of a space frame by Korean Standard Steel Design Code. In this study, the

---

* 세종대학교 건축공학과 교수
** 현대건설(주) 기술연구소 연구원

development environment is as follows:

Hardware : SGI Indigo2, SGI Power Challenger
O/S : IRIX 5.2, IRIX 6.2
Language : PCL (PATRAN Command Language), C++
Application : PATRAN, ABAQUS

2. SYSTEM MODELING

2.1 Flowing of Data

A hybrid model is proposed in this paper for an integrated design system. This hybrid model is consisted of linked applications, data files and database. The interrelationship of PATRAN, ABAQUS, Model Generator and Optimum Design Module are displayed in Figure 1. Model Generator and Optimum Design Module are interpreted and executed in the PATRAN environment. As shown in figure 1, PATRAN creates a finite element model and saves data in the database, and then the forward translator of PATRAN generates input files of ABAQUS. Finally, the backward translator of PATRAN is executed to restore output files of ABAQUS to the PATRAN database.

Two individual modules (Model Generator, Optimum Design Module) communicate through this PATRAN database. The Model Generator creates three specific finite element models of space frames such as flat truss, vault truss, and geodesic dome. The Optimum Design Module optimizes member size of a space frame according to allowable stress level.

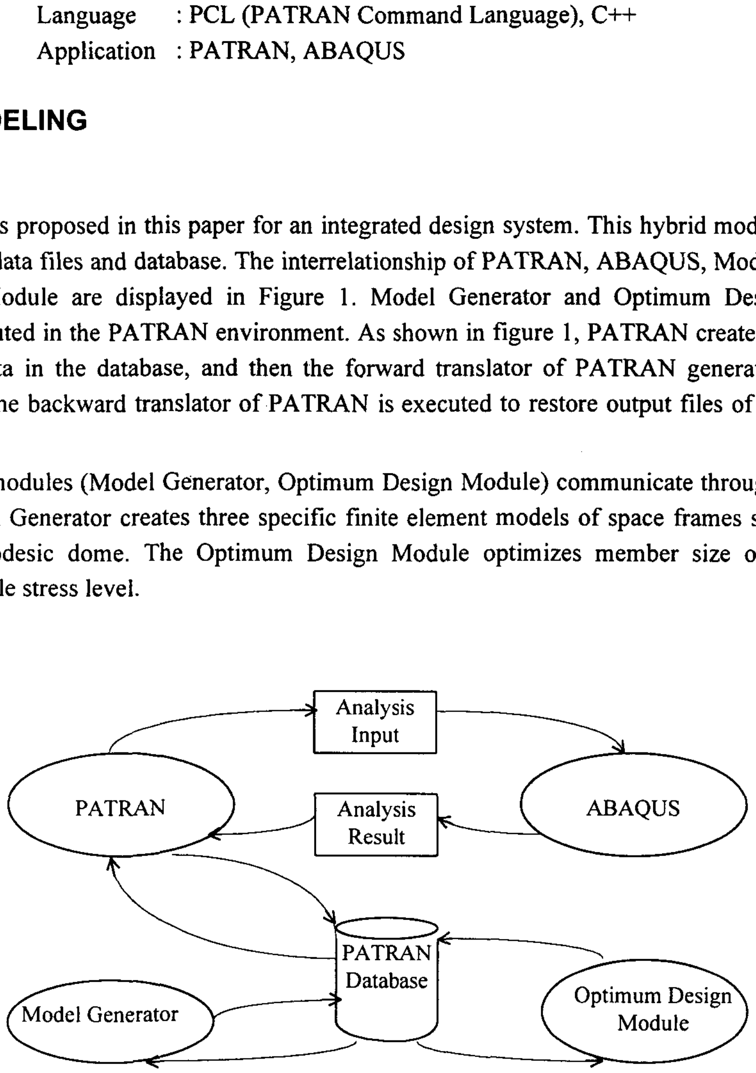

Figure 1. Flowing of data

2.2 Object Model

The object model scheme$^{(4)}$ is emplyed to model this integrated design system. As shown in Figure 2, the system is composed of 'Member', 'M_Group' and 'Space Frame' objects. The 'Member' object describes a member of the space frame. The attributes of the 'Member' object are identifier, cross-section and maximum number of members. The operators of 'Member' object calculate slenderness ratio, allowable stresses and member stresses. The 'M_Group' object consists of the attributes that are number and identifier of members and groups, and its operator is member selection. Association of 'Member' and 'M_Group' objects is 'Include'. An attribute of 'Space Frame' object is number of groups, and its operators are member grouping, member design, and calculation sheet generation, etc. The 'Space Frame' object is related 'M_Group' object with 'include' association. The 'Flat Truss', 'Vault Truss' and 'Geodesic Dome' objects inherit attributes from the 'Space Frame' object, and their operator create finite element model of each object.

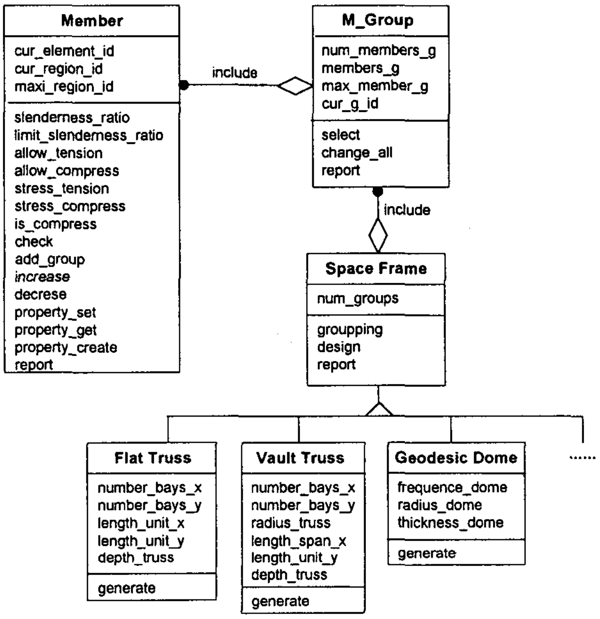

Figure 2. Object model of space frame

3. MODEL GENETATOR

3.1 FORMEX Algebra

As well as the scalar algebra, the FORMEX algebra has an relationship between dependent and independent variables as follows:

$$G = \Phi : E \qquad \cdots\cdots\cdots\cdots \quad (1)$$

where an operator ':' is called "Ralus", E is an independent variable, G is dependent variable, and Φ is a function which defines relationship of these variables. FORMEX algebra functions used in this system are translation, reflection, vertition, projection, and dilatation.

3.2 FORMEX Algebra Form of Space Frame

A space frame is modeled by using FORMEX algebra through array and connection of structural members, and is displayed in geometric coordinates. Nodes and members of a space frame are written in terms of

FORMEX algebra as follows:

Flat truss

Node : $j_1=[1,1]$, $j_2=[3,1]$ ……………………… (2)

Member : $e_1=[1,1;3,1]$,$e_2=[1,1;1,3]$,$a_1=[1,1;2,2]$,$b_2=\mathrm{tran}(1,2):a_1$

$$F=\sum\frac{6}{i=0}\mathrm{tran}(2,2i):\sum\frac{9}{j=0}\mathrm{tran}(1,2j):\sum\frac{3}{k=0}\mathrm{ver}(1,2,2,2)k:[1,1;2,2] \quad \cdots \quad (3)$$

Geodesic dome

External Node: $g1=\sum\frac{\mathrm{nf}}{i=0}\sum\frac{i}{j=0}\mathrm{projid}(3i,\mathrm{nf}-i+2j):[0,0,1]$ ……………… (4)

Internal Node: $g2=\sum\frac{\mathrm{nf}-1}{i=0}\sum\frac{i}{j=0}\mathrm{projid}(3i,\mathrm{nf}-i+2j):[0,0,0]$ ……………… (5)

Member : $F1=\mathrm{vin}(2,\sqrt{10}):g1\#\,\mathrm{vin}(2,10):g2$ ………………… (6)

Diagonal : $F1=\mathrm{vin}(\sqrt{5},\sqrt{5}):(g1\#\,g2)\#\,\mathrm{vin}(3,3):(g1\#\,g2)$ ………………… (7)

Table 1. Section Properties of Circular Tubes

Number	Diameter (mm)	Thickness (mm)	Unit Weight (kg/m)	Sectional Area (cm^2)	Moment of Inertia (cm^4)	Sectional Modulus (cm^3)	Radius of gyration (cm)
1	21.7	2.0	0.972	1.238	0.607	0.560	0.700
2	27.2	2.0	1.24	1.583	1.26	0.930	0.890
3	27.2	2.3	1.41	1.799	1.41	1.03	0.880
4	34.0	2.3	1.80	2.291	2.89	1.70	1.12
5	42.7	2.3	2.29	2.919	5.97	2.80	1.43
6	48.6	2.3	2.63	3.345	8.99	3.70	1.64
7	42.7	2.8	2.76	3.510	7.02	3.29	1.41
…	…	…	…	…	…	…	…
57	558.8	16.0	214.0	272.8	101000.0	3600.0	19.2
58	609.6	16.0	234.0	298.4	132000.0	4320.0	21.0

4. OPTIMUM DESIGN

Since a space frame is consisted of numerous members, numerical optimization can be time consuming and sometimes almost impossible(5)(6). In this regard, semi-automatic optimization process is proposed in this study as following convention.

First, each member of the space frame is idealized as a three-dimensional truss element, and all the members of the space frame are initialized to a uniform material property. After the analysis is done, all the members are grouped in terms of their sectional properties as shown in Table 1 with respect to its stress level. Reanalysis is automatically performed with updated member property information available in the market until all the members of the group satisfy the Design Code with minimum weight. All of the above procedure is written in C++ language.

5. NUMERICAL EXEMPLES

5.1 Flat Truss

A finite element model of a flat truss is considered as shown in figure 3. In this example, each individual member is modeled as a three-dimensional bar element. Three tons are assumed to be loaded vertically at each node of the finite element model, and six nodes are restrained as simple supports. The flat truss is divided into ten groups according to its stress level. The gc1 group, one of the ten groups, is shown in Figure 4. Convergence of optimum design process for the flat truss is shown in Figure 5. In Figure 5, the vertical axis denotes the material number as shown in Table 1. It is noted that it converges after 4 times run.

5.2 Geodesic Dome

As another example, a geodesic dome is considered as shown in figure 6. In this example, all the condition is same as for the flat truss case, and ten nodes are restrained as simple supports. The gc1 group in Figure 7 is one of the ten groups for the geodesic dome. As shown in Figure 8, all the groups converge faster than that of the flat truss. That is because the initial grouping of the geodesic dome is accurate enough to produce uniform stress level all over the member groups.

6. CONCLUDING REMARKS

The space frame integrated design system is developed in this study. This system is consisted of Model Generator and Optimum Design Module. These two modules interactively work within the PATRAN database environment. The FORMEX algebra is employed in Model Generator to create complicated finite element model of a space frame. A semi-automatic optimum design method with appropriate member grouping is proposed in the postprocessing.

It is found that the proposed integrated system significantly reduces time and effort to design space frame members by easily generating finite element model of a space frame and incorporating the semi-automatic optimum design process .

REFERENCES

(1) PDA Engineering (1994), *P3/PATRAN User Manual*, PDA Engineering.

(2) Hibbitt, Karlsson & Sorensen, Inc.(1994), *ABAQUS User's Manual*, Hibbitt, Karlsson & Sorensen, Inc.

(3) Nooshin, H. (1984), *Third International Conference on Space Structures*, Elsevier Applied Science Publisher, London.

(4) Rumbaugh, J., Blaga, M., Premerlani, W., Eddy, F., and Lorensen, W. (1991), *Object-oriented Modeling and Design*, Prentice-Hall.

(5) Wu, S.J., and Chow, P.T. (1995), "Integrated Discrete and Configuration Optimization of Trusses using Genetic Algorithms", Comput.Struct, 55, 695-702.

(6) Lee, C.D. (1996), "Optimization of the Three Dimensional Steel Frame Structures Based on Stochastic Simulated Annealing", Journal of the Architectural institute of Korea, 12(1), 159-167.

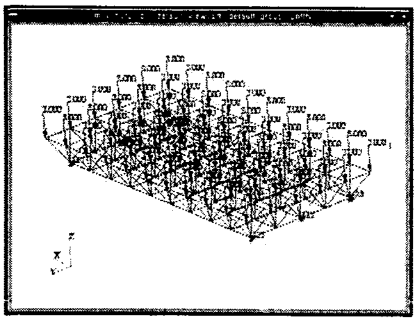

Figure 3. Flat truss

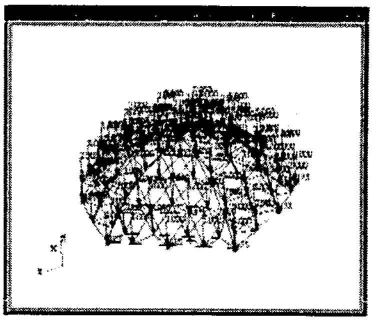

Figure 6. Geodesic dome

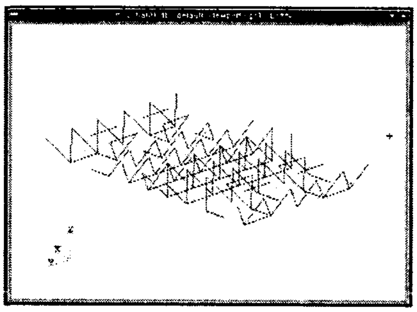

Figure 4. gc1 group

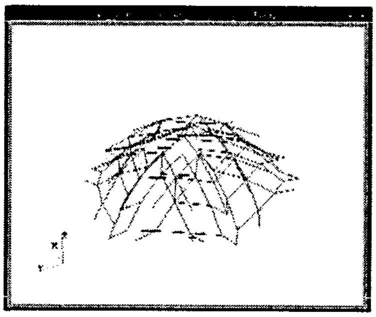

Figure 7. gc1 group

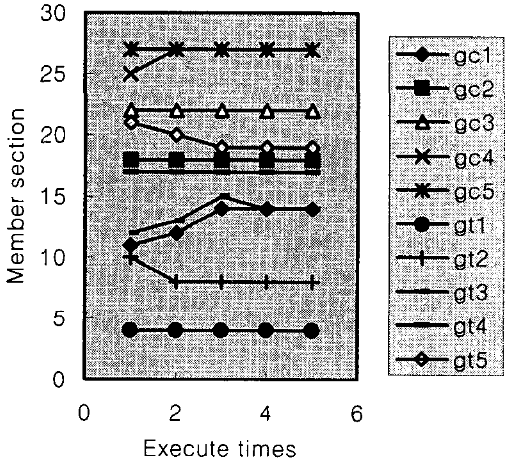

Figure 5. Convergence of flat truss

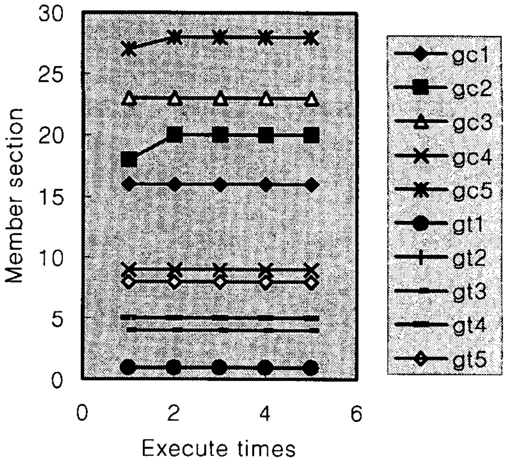

Figure 8. Convergence of geodesic dome