A COMPUTER-AIDED TECHNIQUE FOR GENERATING SIMPLIFIED FRAME-WALL ELASTIC MODELS

Robert A. Bruce (I)

SUMMARY

This study describes a computer-aided technique in which a simplified stick-type model is generated which closely approximates the story translational stiffness of almost any given two-dimensional frame-wall model. The simplified model is made of one-dimensional bending elements and has 4n-4 degrees-of-freedom for n stores. The simplified model can be used to replace a complex two-dimensional frame-wall model in a three-dimensional building analysis program in which the horizontal diaphragms are rigid in plane. Use of the model allows the reduction of computational effort and the transfer of finite-element modeling results to three-dimensional programs using only linear elements.

BACKGROUND

The analysis of multi-story buildings by computer using three-dimensional elastic models is now common in professional practice. Most buildings consist of planar frame-walls cantilevering from a foundation and connected at regular story levels by floor diaphragms. Three-dimensional elastic models are used in order to determine the behavior of these existing or proposed buildings. By the use of computers, elastic building models of great complexity and indeterminacy can be constructed and solved. An analysis using available proven programs readily provides member forces and deformations, story displacements, foundation reactions, and dynamic behavior of structures subjected to vertical or lateral forces.

One type of computer modeling program is the general three-dimensional finite element elastic analysis program, such as SAP or NASTRAN. Analysis of buildings with this type of program involves modeling all the structural elements as assemblies of one, two or three dimensional elements. These programs are capable of quite accurate analysis of elastic behavior due to their completeness. This type of model analysis can be time-consuming to prepare and execute, and is often considered unnecessarily expensive for use in building design.

A second type of elastic modeling program is written especially for building analysis. These programs make simplifying assumptions which greatly reduce input data preparation and computational effort. Program TABS (Ref. 1) is of this type. The assumptions usually made are that horizontal diaphragms are rigid in plane, all lateral loads are applied at floor levels, girders are constrained from axial deformations,

(I) Structural Engineer, H.J. Degenkolb Associates, Engineers, San Francisco, California, USA

intersecting frame-walls are considered to act independently, and members are modeled as either one-dimensional elements or as two-dimensional infill panel elements with restricted degrees-of-freedom.

A complete multi-story building analysis with either type of program would yield the following data for each load case:

1. Base shear, base overturning moment, story shears, and story overturning moment for each frame-wall.

2. Internal forces in members within each frame-wall.

3. Displacement and rotation of the building at each floor, and translational displacement of each frame-wall at each floor.

4. Shear forces in the horizontal diaphragms.

5. Dynamic mode shapes and frequencies (independent of loading).

6. Time-history deflections and frame forces for a given earthquake record.

Although each frame-wall model in the second type of program typically is made up of many elements interconnected to allow numerous degrees-of-freedom, the behavior of each frame-wall model in relation to the total three-dimensional model can be totally described by the flexibility or stiffness matrix of its story translational degrees-of-freedom. Thus, if a frame-wall model having an identical story translational flexibility were substituted in the three-dimensional model for the original, the results in Items 1 and 3 to 6 above would remain unchanged. To obtain the frame-wall member forces (Item 2), one could impose the story translations or forces from the three-dimensional model onto the original frame-wall model.

BASIS FOR SIMPLIFIED MODEL

The story translational flexibility matrix of a frame-wall can be found from the complete stiffness matrix of a frame-wall by a series of matrix operations known as static condensation (Ref. 2). It may also be obtained by imposing unit story translational forces on the frame-wall model and noting the story translations. Most available building analysis programs do not allow a condensed matrix as described above to be directly entered. The programs require that a model of an interconnected assembly of elastic elements be entered.

This study describes a computer-aided technique in which a simplified stick-type model is generated which closely approximates the story translational stiffness of almost any given original frame-wall model. The simplified model is an interconnected assembly of one-dimensional bending elements and has 4n-4 degrees-of-freedom (DOFs) for n stories. The model can be centered in the place of a frame-wall to a three-dimensional program of the second type described above.

There are several advantages to using the simplified model. When the simplified models replace complex frame-wall models with numerous column, spandrel and panel elements, considerably fewer degrees-of-freedom are used in the three-dimensional model, lessening the computational effort to solve that model. In seismic rehabilitation design, existing frame-walls which will remain unchanged maybe entered as simplified models, while the complete models of proposed additional or altered existing frame-walls are successively entered until acceptable forces and displacements in all frame-walls are obtained. Highly irregular existing or proposed frame-walls, known to be difficult to accurately model using one-dimensional and panel elements, may be modeled first using finite elements to more accurately determine their story translational flexibility. A simplified model approximating this flexibility can then be generated for use in the three-dimensional model. Final story forces or displacements to the simplified model, as determined in the three-dimensional analysis, can then be applied to the original frame-wall model to find element forces.

The simplified model used in the program was originally suggested and used by Aoyama (Ref. 3). This model was called the "FB" model, and is shown in Figure 1. A simplified version of the Aoyma "FB" model was conceived which is more suitable for modeling with linear bending elements. This model is shown in Figure 2. In this simplified model, the fixed end outside columns simulate shear deformations, in that their end moments are uncoupled from the columns above and below. The interior hinged end columns act as bending elements, in that their end moments are coupled to their extensions above or below. The forces at any story level in the model are coupled to up to four other story levels. With the DOFs numbered as shown, the stiffness matrix is narrowly banded, and can be solved as a DOF x 6 matrix in an appropriate banded solver.

PROGRAM DESCRIPTION

The purpose of the computer program is to create a simplified model as described above which accurately approximates the story translation stiffness of a given original frame-wall model. This is done by using the program to adjust the moments of inertia of the bending elements of the simplified model until its story translation flexibility matrix converges upon that of the original frame-wall. To accomplish this, the story translation stiffness matrix of the original frame-wall is entered as input to the program. Matching the stiffness matrices is equivalent to matching flexibility matrices because the two are inversions of one another. The element-to-DOF connectivities and initial trial moment of inertia (I) values are successively entered, and the model is solved for a trial flexibility matrix. Numerical methods are used to find the optimum I values needed to make the trial matrix best approximate the given frame-wall matrix. The iterative process is repeated until the matrices converge to the desired accuracy. The two stiffness matrices and the final I values are then printed.

The program was written in BASIC language for use on a 16k RAM micro-computer. Because the simplified model has few degrees-of-freedom and can be assembled so as to be narrow-banded, a 16K computer is large enough to store the program and analyze a ten-story model. A flowchart of the program is shown in Figure 3. A discussion of the functions of the various subprograms is presented below.

SUBPROGRAM FUNCTIONS

Control Subprogram

The control subprogram performs the input and output functions, calls the subprograms, assigns element property values, and evaluates the iteration to convergence of the original and trial stiffness matrices. The two matrices are normalized for comparison, with the largest story translation made equal to one. The final normalized trial matrix is again scaled by comparison with the maximum story deflection of the given flexibility matrix to create the final simplified model matrix which approximates the original matrix. The printed output lists the two matrices and the generated I values for the final simplified model.

Element Matrix Generator, Loader and Global Solver Subprograms

The element matrix generator and global matrix loader subprograms are standard routines which assemble the appropriate slope-deflection equations and enter them to the global matrix at the locations appropriate to the global DOFs. The SYMSOL equation solver was chosen for the global matrix solver to take advantage of the favorable banded nature of the simplified model. The SYMSOL solver was modified to efficiently process multiple unit story load cases.

Element Property Optimization Subprogram

For each element iteration, four trial element I values are computed to bracket the previous I value. The program solves the trial flexibility matrix for each of the four I values. The sum of the squares of the difference between the entries of the original and each of the four trial values. The element property optimization subprogram then uses a Vandermonde matrix to generate a cubic curve which passes through the four trial values. The quadratic expression for the slope of this curve is then solved to yield a minimum value for the sum of the squares, which corresponds to the optimum I value.

USE OF THE PROGRAM

The use of the program is demonstrated in the following example problem, which is typical of the type of problem solved by engineers engaged in seismic design of new structures or of corrective measures for existing structures. In this example, shown in Figure 4, the original frame-wall is difficult to model accurately with linear elements or infill panels due to its geometry. Even if the frame-wall were modeled with these elements, it would have a large number of degrees of freedom.

To obtain a more accurate model, the frame wall as shown was modeled using SUBWAL, a finite element program developed especially for use in building analysis (Ref. 4). The SUBWAL model was analyzed using unit story loads to determine the flexibility matrix for this frame-wall. The shape of this flexibility matrix is shown graphically at the top of Figure 4, with arrows representing the unit loads at each alternate story level. This flexibility matrix was then entered as input to the program for solution. The program was run to converge the matrices and produce a final simplified model. The lower triangles of the symmetrical normalized flexibility matrices for the original frame-wall and the simplified model are shown, as are the final I values for the simplified model. Note that the flexibility of the simplified model closely approximates that of the original.

The above example shear-wall was one of six walls of a hospital wing. A comparative analysis was performed on the hospital wing using program TABS. In the first analysis, all six walls were modeled with linear elements and shear panels, in accordance with the standard procedures of using TABS. In the second analysis, the four most complex walls were first solved separately as modeled for the first analysis to determine their individual flexibility matrices. Each of the four flexibility matrices was then entered as input to the simplified model program for solution. The program produced four simplified models which closely approximated the original models. The two remaining original frame-walls were not complex and were modeled as in the first analysis.

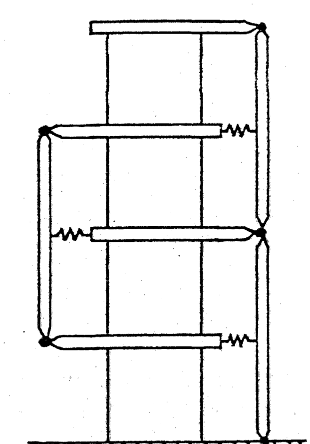


Figure 1 - Aoyama 'FB' Model

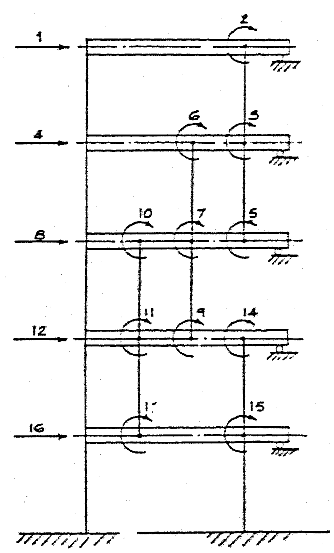


Figure 2 - Simplified 'FB' Model

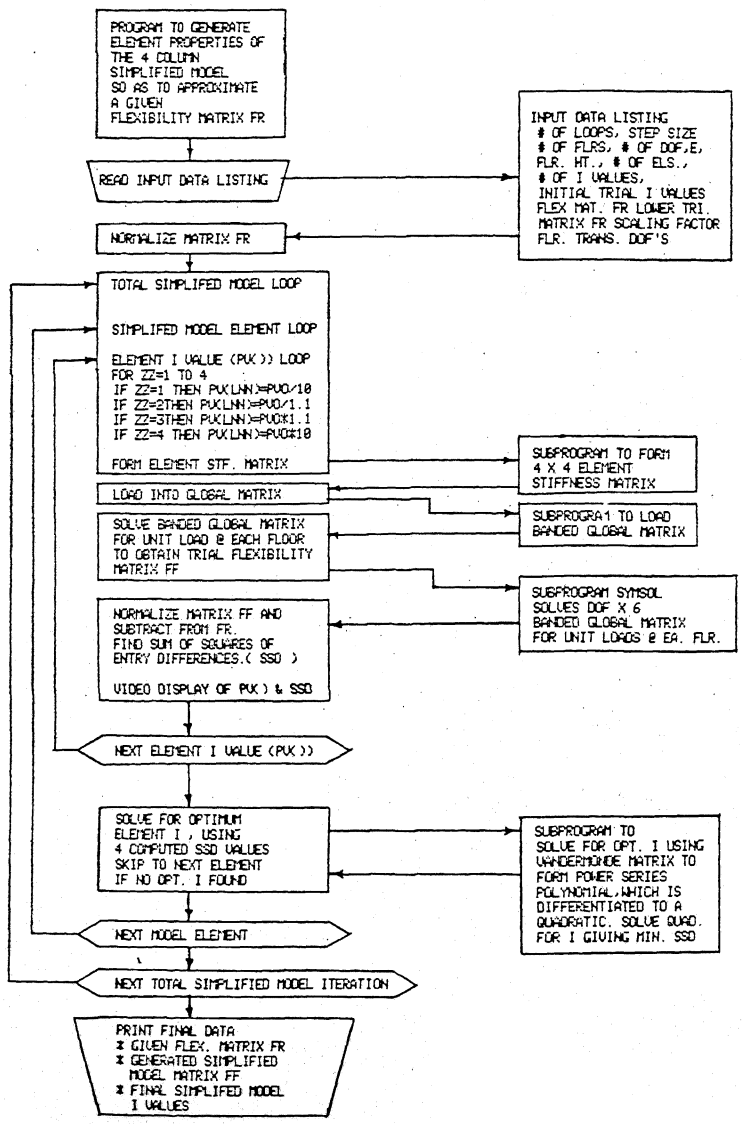
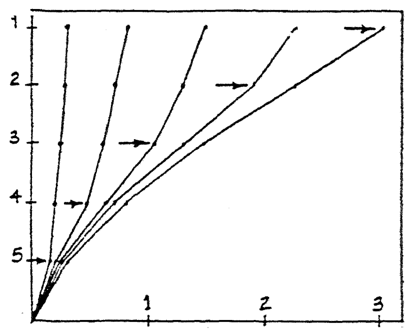


Figure 3 - Program Flowchart



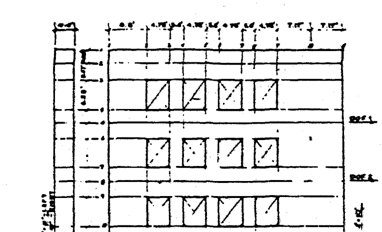
Shape of flexibility matrix for original model.

$$\begin{bmatrix} 3.0409E-05 & & & & \\ 2.28189999E-05 & 1.92859999E-05 & & & \\ 1.49539999E-05 & 1.31439999E-05 & 1.07039999E-05 & & \\ 8.08699999E-06 & 7.22999999E-06 & 6.22399999E-06 & 4.83099999E-06 & \\ 3.00799999E-06 & 2.76399999E-06 & 2.49899999E-06 & 2.19599999E-06 & 1.82419999E-06 \end{bmatrix}$$

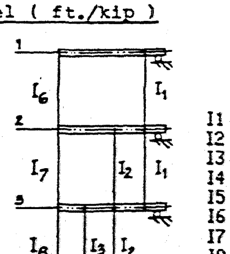
Flexibility matrix of original model (ft./kip)

$$\begin{bmatrix} 3.0409E-05 & & & & \\ 2.30014788E-05 & 1.94918729E-05 & & & \\ 1.44154576E-05 & 1.30823553E-05 & 1.0647739E-05 & & \\ 7.70329263E-06 & 7.31273839E-06 & 6.59347735E-06 & 5.41347208E-06 & \\ 2.45146469E-06 & 2.35340844E-06 & 2.17433032E-06 & 1.87656052E-06 & 9.88549717E-07 \end{bmatrix}$$

Flexibility matrix of simplified model (ft./kip)



Original Frame-wall #1



I1 = 585.
I2 = 592.
I3 = 378.
I4 = 713.
I5 = 1023
I6 = 60.5
I7 = 15.9
I8 = 64.3
I9 = 46.9
I10 = 4.57

Simplified Model

Figure 4 - Example Problem

The four simplified models and the two original models were then used in TABS for the second analysis, and the TABS program was run for the same load cases as used for the first analysis. A dynamic analysis for modal shapes and frequencies was also run for both model assemblies.

The results of the first and second analysis were compared. The modal frequencies and shapes were very close. Static displacements and story shears were closer for the upper floors then for the lower floors. This is due to the comparison routine used in the iterative convergence in the program, which compares the square of the differences between the entries of the original and trial matrices, thus favoring the larger displacements in the upper stories. Overall, the solutions are considered close enough for most practical applications.

The first analysis using the original models required almost seven times the computer time as that required by the second analysis, which used the simplified models. This magnitude of savings could be significant if multiple runs were needed to adjust the stiffness or placement of additional or modified frame-walls in an actual application.

CONCLUSION

The technique offers a simplified frame-wall model which can have significantly fewer degrees-of-freedom than the original model. The effort involved in generating the model can be offset by the effort or expense saved with its repeated use. The accuracy of the model must be seen in comparison with the general practice of multi-story building lateral analysis, in which many simplifying assumptions are considered acceptable.

REFERENCES

1. Wilson, E.L., and Dovey, H.H., Three-Dimensional Analysis of Building Systems, TABS, EERC Report No. 72-8, College of Engineering, University of California, 1972.

2. Clough, R.W., and Penzien, J., Dynamics of Structures, McGraw-Hill, 1975.

3. Aoyama, H., Simple Non-Linear Models for Seismic Response of Reinforced Concrete Buildings, Proceedings of Review Meeting, U.S.-Japan Cooperative Research Program in Earthquake Engineering, with emphasis on Safety of School Buildings, Japan Earthquake Engineering Promotion Society, 1976.

4. Le, D.Q., Pettersson, H., and Popov, E.P., SUBWAL, A... Finite Element Program for ...Structural Walls, EERC Report No. 77/09, College of Engineering, University of California, 1977.