

Aspects on input and output for finite element programs

Autor(en): **Tagnfors, H. / Wiberg, N.-E.**

Objekttyp: **Article**

Zeitschrift: **IABSE reports of the working commissions = Rapports des commissions de travail AIPC = IVBH Berichte der Arbeitskommissionen**

Band (Jahr): **31 (1978)**

PDF erstellt am: **09.08.2024**

Persistenter Link: <https://doi.org/10.5169/seals-24897>

Nutzungsbedingungen

Die ETH-Bibliothek ist Anbieterin der digitalisierten Zeitschriften. Sie besitzt keine Urheberrechte an den Inhalten der Zeitschriften. Die Rechte liegen in der Regel bei den Herausgebern.

Die auf der Plattform e-periodica veröffentlichten Dokumente stehen für nicht-kommerzielle Zwecke in Lehre und Forschung sowie für die private Nutzung frei zur Verfügung. Einzelne Dateien oder Ausdrucke aus diesem Angebot können zusammen mit diesen Nutzungsbedingungen und den korrekten Herkunftsbezeichnungen weitergegeben werden.

Das Veröffentlichen von Bildern in Print- und Online-Publikationen ist nur mit vorheriger Genehmigung der Rechteinhaber erlaubt. Die systematische Speicherung von Teilen des elektronischen Angebots auf anderen Servern bedarf ebenfalls des schriftlichen Einverständnisses der Rechteinhaber.

Haftungsausschluss

Alle Angaben erfolgen ohne Gewähr für Vollständigkeit oder Richtigkeit. Es wird keine Haftung übernommen für Schäden durch die Verwendung von Informationen aus diesem Online-Angebot oder durch das Fehlen von Informationen. Dies gilt auch für Inhalte Dritter, die über dieses Angebot zugänglich sind.

**IABSE
AIPC
IVBH**

**COLLOQUIUM on:
"INTERFACE BETWEEN COMPUTING AND DESIGN IN STRUCTURAL ENGINEERING"**
August 30, 31 - September 1, 1978 - ISMES - BERGAMO (ITALY)

Aspects on Input and Output for Finite Element Programs

Considérations sur l'input et l'output de programmes avec des éléments finis

Aspekte des In-und Output von Programmen mit finiten Elementen

H. TAGNFORS

Res. Eng., Cent. for Comp. Calc.
Chalmers Univ. of Technology
Gothenburg, Sweden

N.-E. WIBERG

Professor, Dept. of Struc. Mech.
Chalmers Univ. of Technology
Gothenburg, Sweden

Summary

For an engineer in a structural design office one of the main requirements for a finite element program is that it must be simple to use in order to minimize the total work in the design process. In order to diminish the gap between man and machine, a general finite element program SITU has been developed. The program, suited for the small and medium sized office has a simple and logical input, logical program structure and an output with high readability.

Résumé

Le projeteur d'un bureau d'ingénieurs a besoin d'un programme avec des éléments finis qui soit simple à l'emploi afin de réduire au minimum le travail total dans l'étude du projet. Le programme SITU a été développé dans ce sens et particulièrement pour les bureaux d'ingénieurs moyen et petits: l'input est simple et logique, la structure du programme est logique et l'output est facilement lisible.

Zusammenfassung

Der Entwerfer eines Ingenieurbüros braucht ein Computerprogramm mit finiten Elementen, welches so einfach sein soll, dass die ganze Arbeit in der Entwurfsphase am geringsten ist.

Das Programm SITU wurde zu diesem Zweck besonders für kleinere und mittlere Ingenieurbüros entwickelt; der Input ist einfach und logisch, die Programmstruktur ist logisch und der Output ist lesbar und verständlich.

1. INTRODUCTION

The finite element method has become a powerful tool for the analysis of complex structural engineering problems. The input and output parts of the computer programs are often not as developed as the calculation part. For an engineer in a structural design office one of the main requirements is that the program must be simple to use, in order to minimize the total work in the design process. This is of utmost importance for all the engineers who do not use the computer every day. The discussion in this paper concerns mainly a small office, with a small equipment: a non-graphic terminal connected to a large computer.

The paper will discuss the use of finite element programs in the design process and different requirements including aspects on safety for the preparation of input data. The need for effective solution techniques, for equation systems and non-linear time dependent problems, is stated. Different ways of result presentation are compared and aspects on readability are laid.

In order to diminish the gap between man and machine a general finite element program, SITU, which is Simple To Use is under development. The program, suited for the small and medium sized office, has a simple and logical input, logical program structure, and an output with high readability. As a demonstration problem the analysis of a wood diaphragm, where the cover is attached to the frame by nonlinear nails, is presented.

2. GENERAL ASPECTS ON USE OF FEM-PROGRAMS

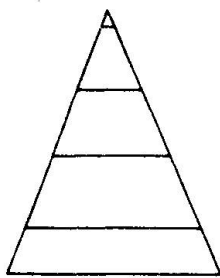
2.1 Computational cost

The aim of an industrial process is to minimize the total cost for a product of a required quality. In structural analysis, extensive use is made of the finite element method (FEM), [1], especially if complicated shape, properties and loading are to be simulated realistically. The finite element analysis may be made in three main

steps:

- . Setting up the calculation model and the relevant data
- . Numerical calculation
- . Presentation and evaluation of results

In practice, the first and third steps usually needs much more effort than the second step. Fig. 1 shows a general estimation of different costs in connection with computer calculations.



computer cost $\sim 10 = 10^1$ \$
 input and output $\sim 100 = 10^2$ \$
 reading the manual
 for the first time
 and making some
 mistakes $\sim 1000 = 10^3$ \$
 program writing $\sim 10^4 - 10^6$ \$

Fig. 1 Cost triangle for FEM-calculations

As seen in Fig. 1 the cost for input and output may be 10 times the direct computer cost. Errors in input data gives large total costs. Reading the manual for the first time may cost another ten times, depending of the type of program. The numerical calculations need very little action by the user. The input and output steps rely heavily on the effort of the analyst to set up, handle and evaluate large data sets. A logical structure and a logical input and readable output and well written manuals may diminish the user cost to a great extent.

In addition we have costs for the equipment at the office. In the discussions below the cost for investments are very limited, in order to suit small construction units. The equipment is a non graphic terminal. Large lists of results are supposed to be mailed.

2.2 The engineer and the computer

The engineers main concern is to solve his particular problems in the easiest and quickest manner, preferably at as low cost as possible. In order to simplify the use of a computer some basic

requirements must be fulfilled:

- . Physically easy to use (terminal on the desk)
- . Simple instructions to the computer
- . Special and general programs, easy to use

A very important matter is to make the contact with the computer very easy by having terminals on the desk of each engineer. At the terminal the engineer can create a job and submit it to the computer. Limited amount of result can be studied. Large output can either be taken to a minicomputer-terminal at the office or can be mailed from the computer centre.

The necessary instructions to the computer for running a program must be extremely simple with for example direct questions and answers which need a minimum of knowledge of the job control language of the computer. More people in practice would use computers in practice to a greater extent if the instructions to be given were more user oriented, to suit people who do not use the computer every day.

A program library ought to cover two main types of programs, special programs and general programs. A special program, which may calculate a frame, a plate or a 3D-body, must be so simple to run, that any engineer directly can run the program from a terminal and get the result almost immediately with minimal possibility of all types of errors. A general program is supposed for complicated problems e.g. coupled structures, (different element types mixed together), non-linear problems and time dependent problems.

The program package CHALMFEM, [2] can to a certain extent meet the requirements above, Fig. 2. After identification (/ID) conversive instructions are given to the operative system concerning used FEM-program, files on secondary memory etc. An input file for the program is generated. The chosen FEM-program reads data from the input file and produces results on a result file. By inspection of the result file, changes of the structure can be decided, and performed by changing the input file.

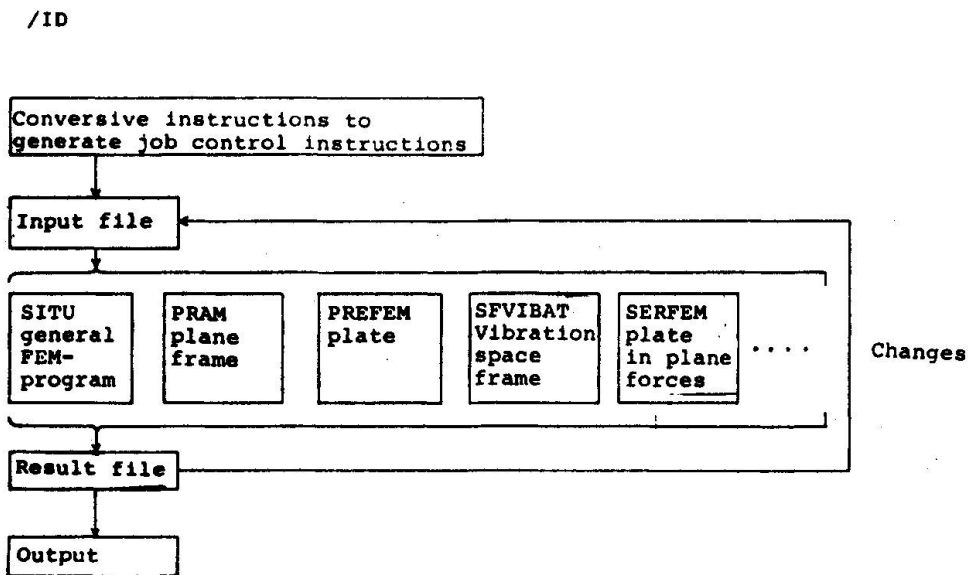


Fig. 2 Calculation with programs in CHALMFEM

2.3 Demands on a general program

From the engineers point of view he want to describe the problem in a simple way and obtain a correct result which contain only what he is interested of. To meet these demands some requirements must be fullfilled:

- . A language for giving input data, close to the engineers
- . Safe algorithms for the calculations
- . A variety of output selections.

The structure of a general FEM-program must be logical and divided into a number of well defined modules.

In general; input data, calculation and output must be made so that a "safe" result is obtained. The old manual way of writing input data which for example use fixed format, node- and element numbering as input, steering of the calculation by a number in a special input position, is time consuming, tedious and error-prone. However, it is possible to construct a simple language for giving input data even for the complicated problems.

In nonlinear and time dependent problems there are a need for effective routines in order to minimize the computer cost and to obtain

a numerically safe solution. Such routines are time-stepping routines which gives an automatic selection of time steps based on a local error estimation [3]. Other routines are direct and iterative solution routines for equation systems [4]. Use of substructuring will simplify input, calculation and output.

A simple and readable output with possibilities of output selection is also a very important part in a FEM-program. A list of results, referred to the structure by node numbers and variable numbers are difficult to read and errors might be undetected. Plotted diagrams are of course very useful, but has to be complemented by the results presented at the terminal or the line printer.

A general program SITU [5], under development, has a very simple and logical input in the language of an engineer.

3. SITU, A FEM-PROGRAM SIMPLE TO USE

3.1 The program structure in SITU

The total structure is built by plane substructures S, see Fig. 3.

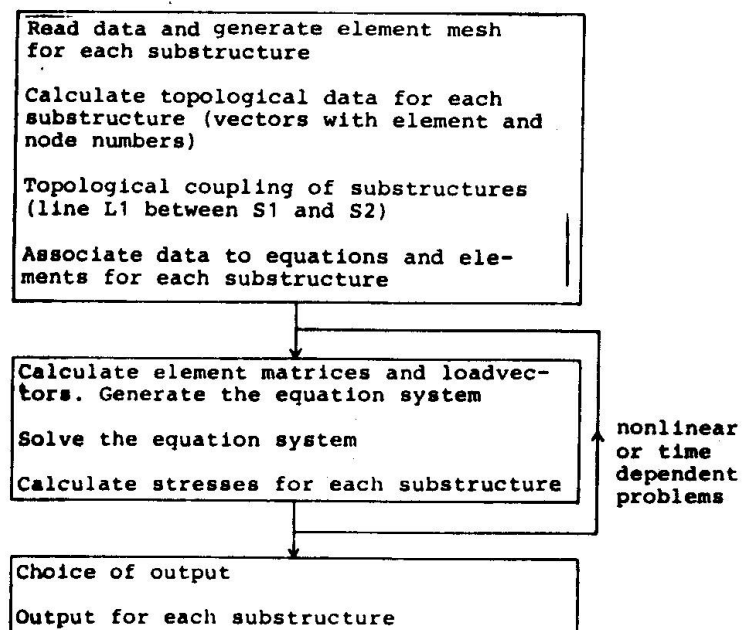
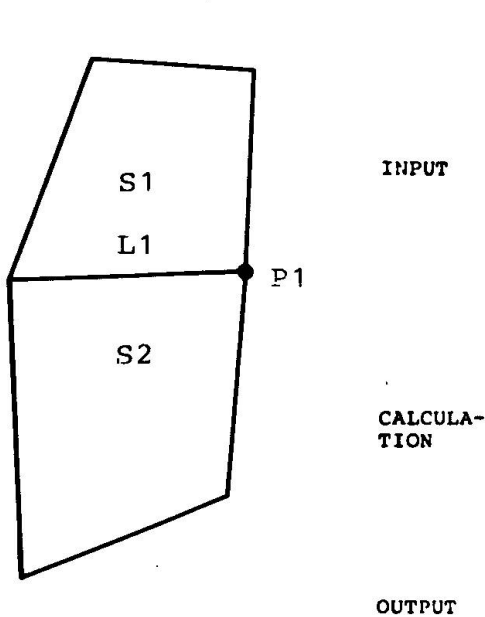


Fig. 3 Structure built by substructures

Fig. 4 Program structure in SITU

Each substructure is geometrically defined by points (P), lines (L) and areas (A). Loads, boundary conditions and material properties etc are defined and assigned to the structure by given geometrical properties. Positions where results are wanted are referred to by given names of geometrical properties. By this method the user does not have to concern with node numbers, variable numbers etc, because these numberings are only used in the program and does not appear in input and output. The main program structure in SITU is given in Fig. 4. Input and output are described below in an example.

4. NUMERICAL CALCULATION OF A WOOD-DIAPHRAGM

4.1 Problem description

The stresses and displacements in the wall in Fig. 5a are to be calculated by FEM. The wall is a wood-diaphragm consisting of a frame with a nailed sheet, see [6]. From the calculation point of view, the wall is separated into three substructures, frame, sheet and nails, Fig 5b.

The frame is divided into beam elements, the sheet into plane elements (8 node element based on a biquadratic displacement field). The frame and the sheet are coupled in a number of nodes by nail elements. The nails are concentrated to these points. The material in the frame and sheets is linear elastic. The nail has a nonlinear force-displacement (T, t) relationship, given by a polygon, see input data. This structure gives a very complex description

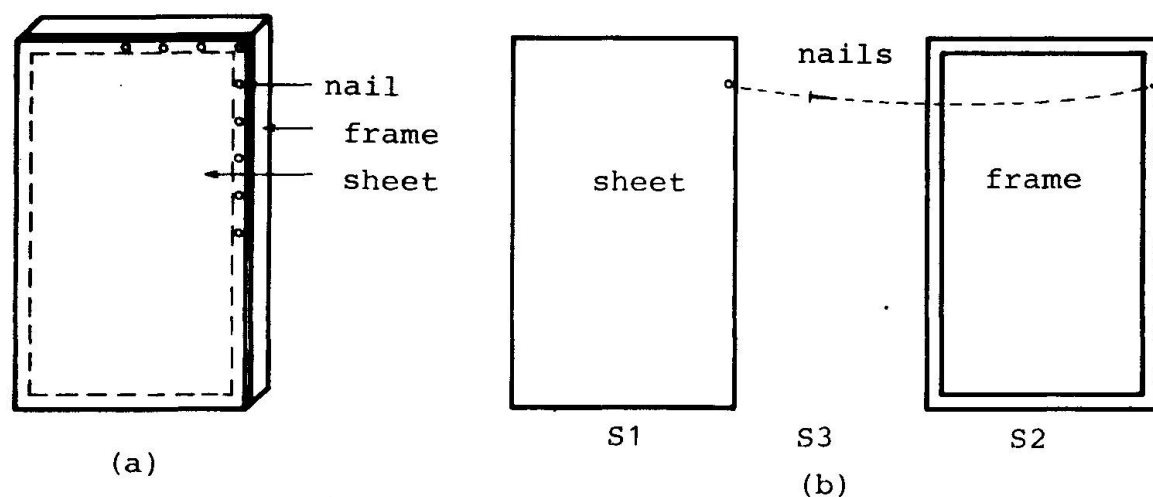


Fig. 5 Wood-diaphragm

4.2 Input data

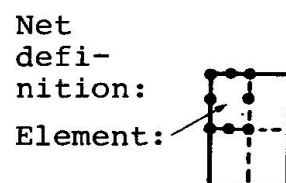
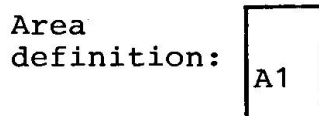
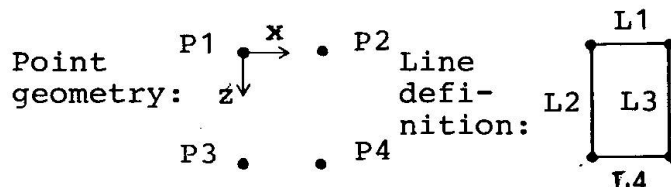
The input data for the example in Fig. 5 will be described below. The element mesh is here chosen very coarse in order to make the description small in size. The needed input is given below together with some explaining figures.

```
SHEET + NAILS + FRAME
SUBSTRUCTURE S1 SHEET
POINTS
P1 0. 0.
P2 2000. 0.
P3 0 3000
P4 2000 3000
LINES L1 P1 P2
L2 P1 P3
L3 P2 P4
L4 P3 P4
AREA A1 L2 L4 L3 L1
NET TYPE 21 2 2
ELEMENT 21
DEFINITION MATERIAL AREA MA1 A
2000 0 13
DEFINITION LOAD POINT PL1 A
1000
ASSIGN MA1 A1
ASSIGN PL1 P1 X
PRINT DISPLACEMENTS
```

```
SUBSTRUCTURE S2 NAILS
POINTS ACCORDING S1
LINES ACCORDING S1
NET TYPE 1 5 ELEMENTS ON LINE
ELEMENT 1 X Z X1 Z1
DEFINITION MATERIAL POINT MA1 B
0. 0.
0.1 100
0.8 200
2 300
3.8 400
7 475
14 500
14 0
20 0
ASSIGN MA1 L1 L2 L3 L4
PRINT NAIL FORCES
```

```
SUBSTRUCTURE S3 FRAME
POINTS ACCORDING S1
LINES ACCORDING S1
NET TYPE 11 4 ELEMENTS ON LINE
ELEMENT 11 X1 Z1 Y1 X1 Z1 Y1
DEFINITION MATERIAL LINE MA1 A
10000 4275 721400
DEFINITION MATERIAL LINE MA2 A
10000 2025 341720
DEFINITION BOUNDARY POINT BO1 A
0
ASSIGN MA1 L1 L4
ASSIGN MA2 L2 L3
ASSIGN BO1 P3 P4 X1 Z1
PRINT INPUT DATA
```

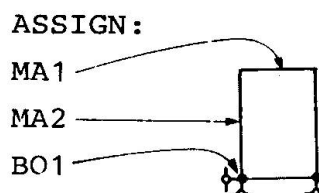
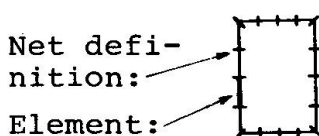
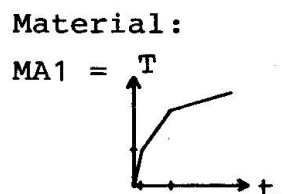
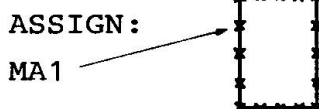
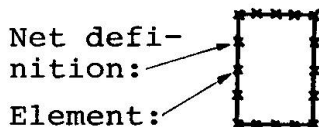
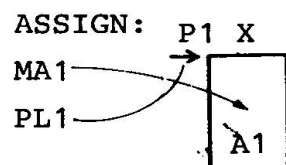
```
STRUCTURE S4
ADD S1 S2 S3
EQUAL BY NAME OF LINES
SOLVE
LOAD 1. INCREMENT 1.
END OF DATA
```



Material:

MA1 = (E, ν , t)
(2000, 0, 13)

PL1 = (P)
(1000)



Material:

MA1 = (E, A, I)
(1000, 4275, 721400)

MA2 = (E, A, I)
(1000, 2025, 341720)

BO1 = (p)
(0)

COUPLING OF SUBSTRUCTURES
NUMERICAL CALCULATION

4.3 Input data check

Results from the input analysis are shown in matrix as a receipt. It is given for each substructure when requested. Fig. 8 shows the material properties and boundary conditions for the frame (substructure 3).

| MATERIAL PROPERTIES FOR ELEMENTS | | | | | | | | |
|----------------------------------|-----|-----|-----|-----|----------|--------|-------|---------|
| ----- | | | | | MATERIAL | E | A | I |
| P1 | MA1 | MA1 | MA1 | MA1 | MA1 | 10000. | 4275. | 721400. |
| MA2 | | | | | MA2 | 10000. | 2025. | 341720. |
| MA2 | | | | | MA2 | | | |
| MA2 | | | | | MA2 | | | |
| MA2 | | | | | MA2 | | | |
| MA2 | | | | | MA2 | | | |
| P3 | MA1 | MA1 | MA1 | MA1 | P4 | | | |

(a)

PRESCRIBED DISPLACEMENTS AT THE NODES

PRESCRIBED DISPLACEMENTS IN X-DIRECTION

| | | | | | | | | | |
|----|---|---|---|---|----|---|---|---|---|
| P1 | . | . | . | . | P2 | . | . | . | . |
| . | . | . | . | . | . | . | . | . | . |
| . | . | . | . | . | . | . | . | . | . |
| P3 | . | . | . | . | P4 | . | . | . | . |
| 0 | . | . | . | . | 0 | . | . | . | . |

PRESCRIBED DISPLACEMENTS IN Z-DIRECTION

| | | | | | | | | | |
|----|---|---|---|---|----|---|---|---|---|
| P1 | . | . | . | . | P2 | . | . | . | . |
| . | . | . | . | . | . | . | . | . | . |
| . | . | . | . | . | . | . | . | . | . |
| P3 | . | . | . | . | P4 | . | . | . | . |
| 0 | . | . | . | . | 0 | . | . | . | . |

(b)

Fig. 8 Input data receipt for the frame (substructure 3)

(a) Material properties

(b) Boundary conditions

4.4 Output from the numerical calculation

As an example of the output from the numerical calculations the displacements at the nodes of the sheet (substructure 1) are given in Fig. 9a, and the nail forces (substructure 2) are shown in Fig. 9b.

DISPLACEMENTS AT THE NODES

DISPLACEMENTS IN X-DIRECTION MULTIPLY PRINTED VALUES BY 10 RAISED TO -3

| | | | | | | |
|------|------|------|------|------|------|------|
| P1 | 2475 | 2398 | 2332 | 2321 | P2 | 2319 |
| 1852 | | | 1867 | | 1859 | |
| 1364 | 1359 | 1353 | 1359 | 1363 | | |
| 863 | | 863 | | 859 | | |
| P3 | 374 | 377 | 378 | 379 | P4 | 377 |

NAIL FORCES

MULTIPLY PRINTED VALUES BY 10 RAISED TO -3

| | | | | | | |
|---------|---------|--------|--------|--------|--------|--------|
| P1 | -305015 | -65986 | -15757 | 72125 | P2 | 323505 |
| -409697 | | | | | 350136 | |
| -387989 | | | | | 389872 | |
| -344986 | | | | | 470977 | |
| P3 | -155991 | 135957 | 263397 | 397993 | P4 | 695672 |

Fig. 9 (a) Displacements at the nodes of the sheet

(b) Nail forces

5 REFERENCES

- [1] O.C. Zienkiewicz: "The finite element method in engineering science", Mc Grow-Hill, London 1971
- [2] "CHALMFEM: Computer program for FEM-analysis of static, dynamic and transient problems, developed at Chalmers University of Technology", Computer Centre in Gothenburg, Gothenburg 1977
- [3] K Runesson, N-E Wiberg and A Wolfbrant: "A new Rosenbrock-type method applied to a finite element-discretized nonlinear diffusion-connection equation", MAFELAP 1978, The mathematics of finite elements and applications, 18-21 April 1978, Brunel 1978.
- [4] R Glemberg and N-E Wiberg: "Iterative solution of finite element equation", Chalmers University of Technology, Department of Structural Mechanics, to be published, Gothenburg 1978
- [5] H Tågnfors and N-E Wiberg: "SITU, A general finite element program, Simple To Use". Chalmers University of Technology, Department of Structural Mechanics, to be published, Gothenburg 1978
- [6] H Tågnfors and N-E Wiberg: "Wood-diaphragms analysed by the finite element method", (In swedish), Chalmers University of Technology, Department of Structural Mechanics, to be published, Gothenburg 1978.