

Zeitschrift: IABSE congress report = Rapport du congrès AIPC = IVBH
Kongressbericht

Band: 12 (1984)

Rubrik: II. Computer aided structural engineering

Nutzungsbedingungen

Die ETH-Bibliothek ist die Anbieterin der digitalisierten Zeitschriften auf E-Periodica. Sie besitzt keine Urheberrechte an den Zeitschriften und ist nicht verantwortlich für deren Inhalte. Die Rechte liegen in der Regel bei den Herausgebern beziehungsweise den externen Rechteinhabern. Das Veröffentlichen von Bildern in Print- und Online-Publikationen sowie auf Social Media-Kanälen oder Webseiten ist nur mit vorheriger Genehmigung der Rechteinhaber erlaubt. [Mehr erfahren](#)

Conditions d'utilisation

L'ETH Library est le fournisseur des revues numérisées. Elle ne détient aucun droit d'auteur sur les revues et n'est pas responsable de leur contenu. En règle générale, les droits sont détenus par les éditeurs ou les détenteurs de droits externes. La reproduction d'images dans des publications imprimées ou en ligne ainsi que sur des canaux de médias sociaux ou des sites web n'est autorisée qu'avec l'accord préalable des détenteurs des droits. [En savoir plus](#)

Terms of use

The ETH Library is the provider of the digitised journals. It does not own any copyrights to the journals and is not responsible for their content. The rights usually lie with the publishers or the external rights holders. Publishing images in print and online publications, as well as on social media channels or websites, is only permitted with the prior consent of the rights holders. [Find out more](#)

Download PDF: 22.02.2026

ETH-Bibliothek Zürich, E-Periodica, <https://www.e-periodica.ch>

SEMINAR

II

Computer-Aided Structural Engineering

Génie des structures assisté par ordinateur

Computergestützter konstruktiver Ingenieurbau

Chairman: J.B. Menzies, United Kingdom

Coordinator: M. Fanelli, Italy

General Reporter: S. Fenves, USA

Leere Seite
Blank page
Page vide

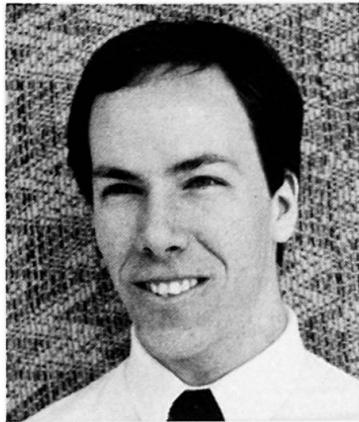
Computer-Aided Design Evaluation of Steel Structures

Evaluation de projets de construction métallique assistés par ordinateur

Computerunterstützte Projektierung von Stahlbauten

Bruce FORDE

Res. Assist.
Univ. of British Columbia
Vancouver, BC, Canada



Bruce Forde, born in 1961, got his civil engineering degree at the University of British Columbia. He has been employed by various consultants and contractors in engineering and research capacities. Presently, he is a research assistant at UBC while completing a MSc degree.

Yee Chiu LEUNG

President
LSCI Ltd.
Vancouver, BC, Canada



Yee Chiu Leung, born in 1943, got his civil engineering degree at the University of Hongkong. For 15 years, he was involved in various engineering fields such as water supply, roads and bridges, structural and geotechnical, before he came to the University of British Columbia to complete a MSc degree.

Siegfried F. STIEMER

Assoc. Prof.
Univ. of British Columbia
Vancouver, BC, Canada



Siegfried F. Stiemer, born 1947, got his degree as Diplom-Engineer at the University of Stuttgart, W.-Germany, in Aeronautical Engineering. Here he finished his Ph.D. and then spent two post-doctoral years at the University of California, Berkeley. His present activities in Canada include various research topics in structural steel design.

SUMMARY

This paper presents the results of an extensive research and development project which was aimed at improving design/fabrication interaction by implementing an information system which utilizes a microcomputer for monitoring the fabrication process. The information collected by this system is analyzed and converted to provide the engineer with data which will be used in the evaluation of designs and in the assessment of the fabrication plant.

RESUME

Ce rapport présente les résultats d'un vaste projet de recherche et de développement visant l'amélioration de l'interaction conception/fabrication. Cette amélioration se réalise par l'implantation d'un système d'information utilisant un micro-ordinateur pour la gestion du procédé de fabrication. Les informations recueillies par le système sont analysées et converties pour fournir à l'ingénieur des informations utiles à l'évaluation des différents projets ainsi que du procédé de fabrication.

ZUSAMMENFASSUNG

Ergebnisse eines ausgedehnten Forschungs- und Entwicklungsprojektes werden dargestellt, das die Konstruktion und Herstellung und deren gegenseitige Beeinflussung verbessert durch die Einführung eines Informationssystems, das den Herstellungsprozess mit Hilfe von Microcomputern beschreibt. Die durch dieses System gesammelten Informationen werden analysiert und aufbereitet, um dem Ingenieur Daten bereitzustellen, die entweder zur Konstruktionsbewertung oder zur Betriebsaufnahme dienen können.



1. INTRODUCTION

In the past, "experience" has been the key to efficient structural steel design. Successful engineers have established design guidelines which take advantage of information that has been acquired through interaction with fabricators. However, in an effort to maintain confidentiality of their direct costs, fabricators are likely to provide the designer with information of only a qualitative nature. The fabricators have a more quantitative look at the information since they are able to directly view the operation and can account for the costs as they are incurred. Nevertheless, even the fabricators are somewhat unsure of the exact costs associated with various structural forms, due to the variation of the type and quantity of work done for any one project.

The size and complexity of the projects that are undertaken by most structural steel fabricators has a significant variation over time. It is this inherent variability of the fabrication process which poses the largest problem to the designer. In order to assign a cost to an arbitrary structural form, costs for each component can be calculated from a standard set of fabrication variables, which are in turn determined and updated through analysis of past projects. The fabrication process, although composed of physically simple operations, as a whole is extremely complicated; hence, associating costs with the fabrication of specific structural components is difficult. In fact the volume of work involved in the collection, organization, and analysis of data required to identify where costs are incurred has effectively eliminated this approach from conventional practice.

Recent developments in microcomputer technology have opened a new area of research in structural engineering. Addressable memory sizes are now adequate and affordable for even medium or small fabricators. Through cooperation with local steel fabricators a tool for evaluating structural steel designs has been developed at the University of British Columbia. Information can now be gathered at the activity level, and stored in digital form, permitting the designer to evaluate the efficiency of an arbitrary design by means of an estimation program.

This paper describes a systematic procedure for determining structural steel fabrication cost data, based on an information system which utilizes a microcomputer for monitoring the fabrication process. The information collected by this system is analyzed and converted to provide the engineer with data which will be used in the evaluation of designs. The information system will also provide a means of assessing the fabrication plant, so that areas of weakness may be identified.

2. STRUCTURAL STEEL FABRICATION

2.1 Design Efficiency

The efficiency of a structural design can be defined as the extent to which it satisfies structural, architectural, and cost constraints. The role of the structural engineer in the design process has traditionally been to choose and design the most efficient structural system subject to the given constraints. This has been no easy task, since the constraints are linked so that there must be some tradeoff made in reaching the optimal design. Architectural requirements are generally defined by the owner as a given requirement of the project, so structural form is to be varied to provide the most economical design.

Another important fact complicates the situation: structural steel fabrication is usually not a mass production, owing to the nature of the

construction process which makes each project unique. However, elements of jobs may be similar and can be used as a basis for comparison. Assuming that the scope of the project has been set and that the design criteria has been reduced to cost, the designer is posed with the problem of associating costs with various structural forms. In most situations, the designer will not have access to the necessary cost data; hence, the formulation of the design objective can not be in terms of a cost function, and the designer is unable to perform formal structural cost optimization. To surpass this barrier, the designer must investigate the nature of the costs associated with the fabrication process.

2.2 Fabrication Costs

The cost of fabricating steel structures may be broken down into four major categories:

1. Material ;
2. Labour ;
3. Equipment;
4. Overhead .

Steel is purchased from a mill at a unit cost per weight, plus some additional charges also based on weight; hence, the cost of material is approximately proportional to the weight of the structure. This has been the major driving force for minimum weight design, since minimizing the weight also minimizes the material cost.

Labour and equipment costs are not so easily related to the form of the structure. Obviously some designs are more efficient than others in terms of ease of fabrication, but it is difficult to determine why. A key factor in these costs is material handling, which appears to be one area in the fabrication process which has large potential for improvement. Handling costs are presently assumed to be some percentage of the costs directly related to the fabrication operation. The reduction in costs associated with repetitive structural forms may be attributable to improvement in productivity for many operations in the fabrication process, such as material handling, set-up time, and labour efficiency. Designers try to account for this by using as many similar components as possible in a structure. The rising labour and machinery costs are becoming an increasingly important part of the overall cost, so a more deterministic approach to their evaluation is needed.

The fabrication cost of the i th structural member type can be written as:

$$c_i = N \left[\sum_{j=1}^{n_i} c_m j w_j + \sum_{k=1}^{o_i} c_\ell k t_k \right] \quad (1)$$

where c_m , c_ℓ are unit costs for material and labour.

w_j is the weight of material j .

t_k is the time required to perform operation k .

n_i , o_i are the number of components and operations for member i .

N is the number of identical members.

The quantity of material fabricated has an unknown influence on the productivity; hence, an interaction diagram must be developed from experience relating N to T . One method of doing this is to monitor the daily operations of the fabrication plant so that a time-history plot of the work being done is available. This information can be used to identify the rate at which work is proceeding on various components of the structure as functions of time, which can be translated into curves which represent productivity improvement due to repetition of similar acts.



The total cost of a structure can be written as the sum of the component costs:

$$\text{Total Cost} = \sum_j (C_j) + O \quad (2)$$

C_j = cost of component j (as given in equation 1)
 O = overhead costs

Examining these equations reveals that: c_m and c_ℓ , are known constants; W and T are known functions of N ; and N is the variable that we can play with to optimize the costs.

2.3 Erection Costs

The current procedure for estimation of erection costs has a large uncertainty associated with its accuracy. The estimated cost of erecting an arbitrary structure has commonly been based on the material weight, without provision for the complexity of the erection procedure. Obviously, some structures are more erection friendly than others; hence, some fabricators will alter their costs by a factor from experience. Decisions of this nature have been in error by as much as 100% and more due to the large variations in productivity associated with the erection activities. These variations may be attributed to the same kind of improvement trends which result from repetitive activities as encountered in the fabrication process.

The proposed procedure for evaluating erection costs follows a similar outline as for fabrication costs. Costs associated with an arbitrary structure will be calculated from a set of standard activities performed during erection (transportation, sorting, lifting, bolting, etc.). A large portion of equipment used during erection is rented; hence, equipment cost is proportional to the erection time. This is quite different from the fabrication process where most equipment is owned by the fabricator and can be essentially considered as overhead.

3. DESIGN/FABRICATION INTERACTION

3.1 Current Practice

Most steel structures that are constructed in Canada follow a process which has been established for historical, political, and economical reasons. The three parties involved in the construction process are: the owner, a consultant, and a contractor. The owner employs the consultant to design the structure, and to supervise the contractor who performs the construction of the structure. Since the design is partially or fully complete at the time when the contract is put up for tender, the design usually will have been done without any interaction between the designer and the contractor. This kind of practice is only acceptable if the designer can foresee the costs associated with various structural forms, so that the design with minimum cost can be chosen. For steel structures, the majority of the cost is incurred in a fabrication plant, of which the designer has only a little knowledge. In this case it is apparent that the consultant is unable to design the most cost efficient building, unless there is some interaction with the fabricator.

Presently, several structural steel fabricators receive design drawings from the consultant with the invitation to tender. A detailed fabrication cost estimate is made from these drawings and submitted back to the consultant in the form of a bid. At this time the fabricator has identified some areas where savings can be made by suggesting some structurally equivalent, but more cost efficient designs for certain components of the structure.

Quite often the designer may receive notice from the fabricator at the time of the bid that there may be some means of cutting costs, since this is an effective means of making his bid more attractive. But after this time, these changes are usually not suggested by the fabricator due to the inability of the design process to easily accept changes.

The characteristics of each fabrication plant depends on the size, and type of projects that the fabricator is involved with. This means that every fabrication plant will have its own set of cost variables which make it more suitable for the fabrication of certain types of structures than for others. Thus, interaction between the designer and the fabricator is essential for all projects, and must become an integral part of the construction process.

3.2 The Proposed Information System

An information system is a planning and control process which is based on the collection, organization, and analysis of data utilizing a computer. The implementation of an information system in a non-computerized environment is likely to be done over some period of time; hence, it must be compatible with the present manual system so that regular operation can be maintained during the transition period. The requirement of compatibility is to have the ability to accept and transmit information that is normally handled by the present system without requiring extensive changes at the base level. This can be achieved by doing the replacement in a modular nature. Care must be taken to avoid simply duplicating the present system in a computerized form, since this may accentuate the limitations and inefficiencies inherent in the manual system while ignoring the newly available computer graphic aids.

The operation and efficiency of an information system is dependent on the "language" of the data communication. A work breakdown structure, which represents the fabrication process in code form, must be implemented to facilitate effective communication between users of the system. The design of a work breakdown structure must be independent of the organization-responsibility structure, so that information pertaining to one item has the same meaning at various levels of production. Information which has been translated into code should be meaningful to the users, so that the work breakdown structure can be associated with physical operations in the fabrication process.

The data collected using the work breakdown structure can be organized to provide two types of information: project data, and process data. Project data pertains to a particular project or structure, and will be organized in a structure database. Each item in the structure database will be tracked through the plant by monitoring item progress at key operation checkpoints. A time/progress record for each item will be available to the fabricator, so that a deterministic analysis of the costs incurred during the fabrication of various structural components may be performed.

Process data pertains to all plant processes (cleaning, cutting, drilling/punching, welding, material handling, etc.). This data is assembled from the same monitoring operation as project data, but is organized to provide information about the efficiency of the plant itself. Simulation of the fabrication plant operation is required in order to evaluate the efficiency of the plant configuration, and with the use of the process data examination of expansion or replacement investments are made possible. Thus, an interactive plant optimization can be done by investigating each activity in the fabrication process. Monitoring the job progress from the component level provides information which would be used by engineering, drafting, purchasing, shop, shipping, and managerial operations. This information could then be used to interactively optimize the processing of material through the plant.

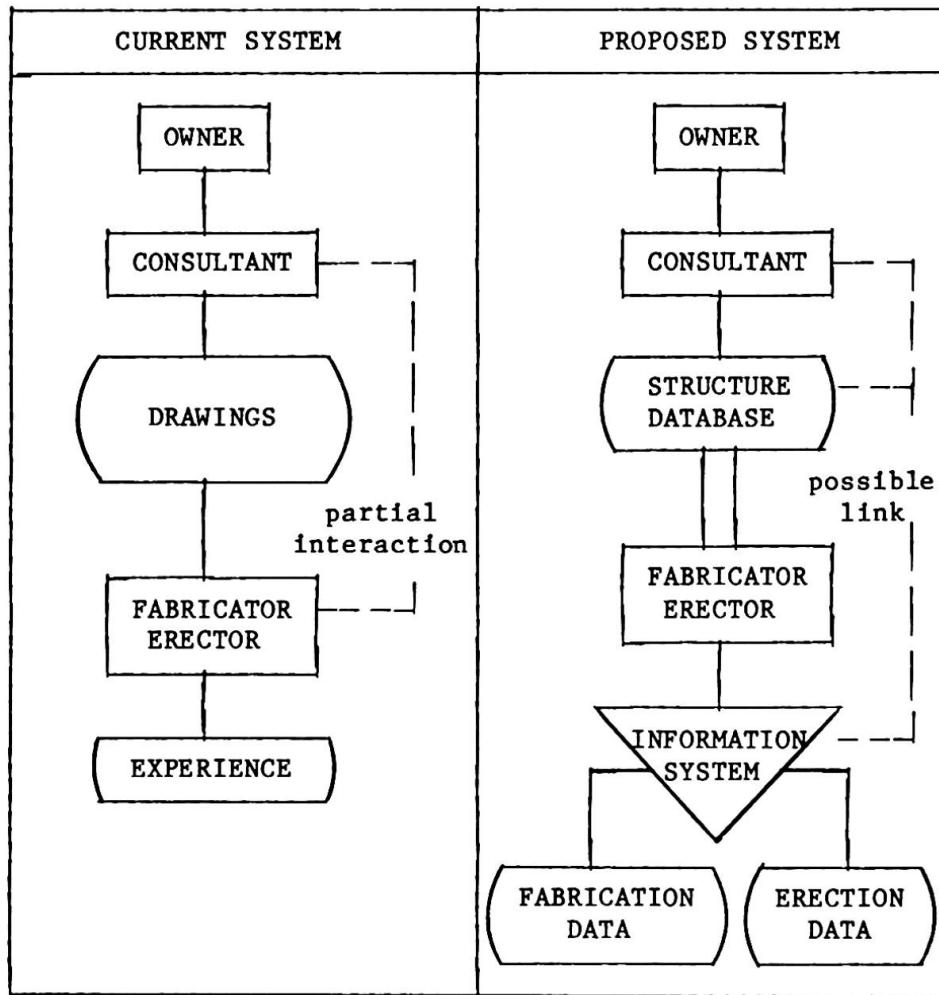


Figure 1. Current and Proposed Systems.

3.3 Optimization

As previously explained, structural cost optimization has not been performed in the past due to a lack of cost information. During this time, the design profession has used some other design tools, combined with some good common sense, to attain near optimum cost designs. One not very successful approach has been to calculate a minimum weight design in order to minimize the material costs. Experienced designers recognize that cost savings due to the peculiarities of the fabrication process could be attained through design simplification and repetition of elements. The problem facing designers, is still to determine the tradeoff point between these philosophies which produces the minimum cost design.

The proposed new design process takes the above procedure one step further. The consultant can directly compare the costs associated with alternative designs thereby identifying the components with minimum cost. The combination of components which produces an overall minimum can be selected from the available alternatives by using a dynamic program. This program can identify the optimal structural design by examination of alternative structural components and fabrication productivity curves.

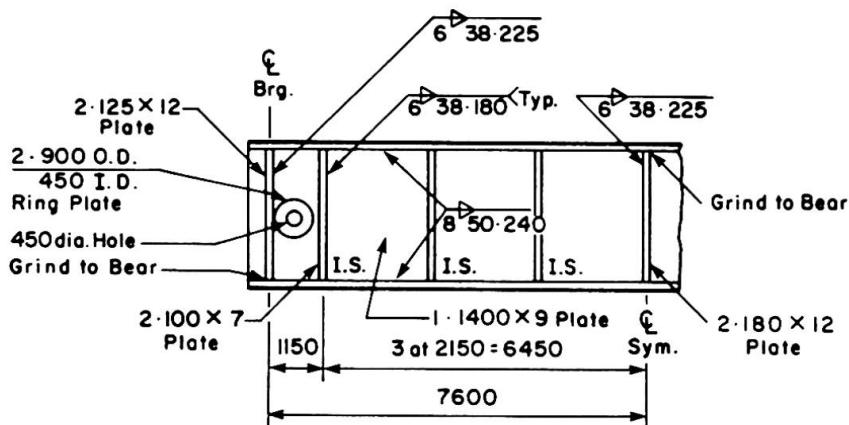
Since the number of components is proportional to the number of alternative structural forms the memory requirements of the program expand dramatically as the structure size increases. Fortunately the productivity

curves associated with the fabrication process level off to approximately constant values once the number of components is sufficiently high. This may allow us to separate the structure into modules which can be individually optimized while ensuring that an overall optimum is reached.

4. APPLICATION OF THE DESIGN EVALUATION PROGRAM

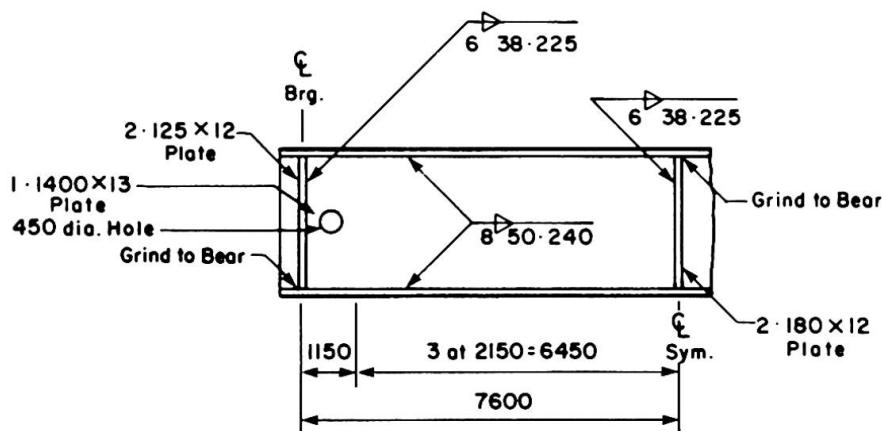
The following example illustrates the use of a routine which utilizes part of the design evaluation program. Using the most recently available fabrication data, and a cost estimation program based on equation 1, two designs for a girder are compared. The results do not intend to show that one scheme is necessarily better in all cases than the other, rather the purpose is to demonstrate how a consultant could use this tool to achieve more economical designs.

4.1 Girder With Stiffeners and Doubler Plate



The total cost of material and fabrication excluding flanges is \$2411.

4.2 Girder Without Stiffeners and Doubler Plate, Web Thickened to Achieve Same Capacity as 4.1



The total cost of material and fabrication excluding flanges is \$2049.



5. SUMMARY

The benefit provided by this procedure is a direct means for evaluating the efficiency of the current design, fabrication and erection processes. Since the designer will be able to identify the fabrication and erection costs associated with a given design, this tool can be used to interactively or automatically optimize the design of a structure. By providing the contractor with a device for monitoring and controlling the fabrication and erection activities, areas of weakness can be identified and modified appropriately.

The final aim of this research is to provide better access to "experience", so that more cost efficient designs can be obtained with ease. The hope is that once the designer is free from the conventional tedious tasks associated with evaluating the efficiency of a design, innovation and productivity will improve.

6. REFERENCES

1. BERRY, G.L., Shop Floor Information System: Design and Implementation, Engineering Digest, (1984).
2. CISC, A Project Analysis Approach to Building, 1980.
3. DONNELLY, J.A., Determining the Cost of Welded Joints, Engineering Journal, AISC, (October 1968).
4. DE GARMO, E.P., Materials and Processes in Manufacturing, 5th Ed., New York: MacMillan Publishing Co., Inc., (1979).
5. MOSES, F., and GOBLE, G., Minimum Cost Structures by Dynamic Programming, Engineering Journal, AISC (July 1970).
6. NIXON, D., Estimating the Cost of Small Steel Buildings, C.J.C.E., Vol. 1, No. 2, (December 1974).
7. PEURIFOY, R.L., Estimating Construction Costs, 3rd Ed., New York: McGraw-Hill Book Company, 1975.
8. RUSSELL, A.D., Cost Optimization of a Structural Roof System, M.A.Sc. Thesis, University of British Columbia, 1970.
9. SHOEMAKER, M.M., The Building Estimator's Reference Book, The Frank R. Walker Company, 1977.

Use of CAD in Finite Element Analysis

Utilisation de la CAO pour le calcul avec des éléments finis

Einsatz von CAD bei Berechnungen mit finiten Elementen

Dieter D. PFAFFINGER

Dr. sc. techn.

P + W Engineering
Zurich, Switzerland



Dieter D. Pfaffinger studied Civil Engineering at the ETH, Zurich, and received his Ph.D. degree there. He then went to Brown University, USA. Later he joined Fides in Zurich to head the structural-engineering department. He is now partner of P+W Engineering and also lecturer at the ETH.

Ueli C. WALDER

Dr. sc. techn.

Dr. Walder and Partners
Bern, Switzerland



After studying Civil Engineering and obtaining his doctorate at the ETH, Zurich, Ueli C. Walder worked for two years in an engineering company in Austria. He then founded Dr. Walder and Partners. He developed the internationally known FE-program FLASH and is now active in the field of CAD.

SUMMARY

The increasing use of CAD-systems makes it feasible to incorporate in a much better way the FE-analysis in the design process. In addition to the graphic-display facilities of modern FE-programs, the geometric data base of CAD-systems offers many new additional possibilities, some of which are discussed in the paper.

RESUME

L'utilisation croissante des systèmes CAO rend plus facile l'intégration de l'analyse avec les éléments finis dans le projet. Outre les représentations graphiques du programme des éléments finis, la banque de données géométriques des systèmes CAO offre beaucoup de possibilités nouvelles, qui sont présentées dans l'article.

ZUSAMMENFASSUNG

Der zunehmende Einsatz von CAD-Systemen ermöglicht es, FE-Berechnungen viel besser in den gesamten Entwurfsprozess zu integrieren. Neben den graphischen Darstellungen der FE-Programme bietet vor allem die geometrische Datenbank von CAD-Systemen viele neue Möglichkeiten, von denen einige diskutiert werden.



1. INTRODUCTION

Modern finite-element programs offer numerous facilities for graphical displays. By means of these, the analysis model can be completely verified before large and costly calculations are performed. This verification is of great importance, because errors in the model can make comprehensive calculations useless or, if they are not detected, can have consequences on the safety of the structure. For the evaluation of the results, graphic displays are also indispensable for furnishing a quick overview of the structural quantities. Thus the engineer can judge the behavior of the structure immediately and may modify his construction accordingly. Through close interaction between man and computer by means of graphics, the most suitable structure for the required specifications can usually be found rather quickly.

During recent years, the use of CAD-systems has become increasingly popular in the construction industry. In such systems, all geometric data of the structure to be designed are stored in a geometric database. It is feasible to use these data for the structural analysis as well. By this approach, errors in the idealization can be drastically reduced, and the efficiency in preparing the FE-mesh can be greatly improved. In addition, selected results of the subsequent structural analysis can be interfaced with the CAD-system for further processing. As an example, the use of the calculated reinforcements of a reinforced-concrete structure in the CAD-system permits the interactive definition of the reinforcing steel. Many CAD-systems also contain information on prices of materials and parts of the structure. By using this information, it is also possible to determine the economic consequences of structural changes readily. On this background the FE-analysis appears as a subtask within the integrated computer-aided design of structures.

In the following, some of the possibilities of graphical representations of FE-programs will be shown which can support the user very effectively. In addition, it will be demonstrated on the basis of the CAD-system CAD-B "Computer-Aided Design of Buildings" of IEZ and the program FLASH, how the structural analysis with finite elements can be integrated into the design process.

2. THE USE OF GRAPHICS FOR THE VERIFICATION OF THE MODEL

One of the standard graphical means for verifying a model is the plot of the mesh. Of particular use are options to shrink the elements around their centers of gravity. By this it can be verified that all elements in the model exist. For larger structures, it is advisable to remove the hidden lines. The resulting plots generally give a very clear picture of the structure. For shell structures, it is useful to verify the angle between elements by plotting the edges of element junctures which exceed a certain given angle. By means of such plots, the user can make sure that the curvatures of the shell are sufficiently well idealized. As an illustration, Figure 1 shows the mesh of an arch dam with selected angles for the elements. This structure is presently under study at the Institute of Structural Engineering of the Swiss Federal Institute of Technology.

In some cases, it is useful to plot the local as well as the global coordinate systems of a structure. This is especially useful for the beam elements because the orientation of the local axes is usually not immediately clear from the plot of the mesh. Figure 2 shows as an example the plot of the element coordinate systems as well as of the system used for the modeling of a part of a stell construction. By means of such plots, errors in the orientation of element loads can easily be avoided.

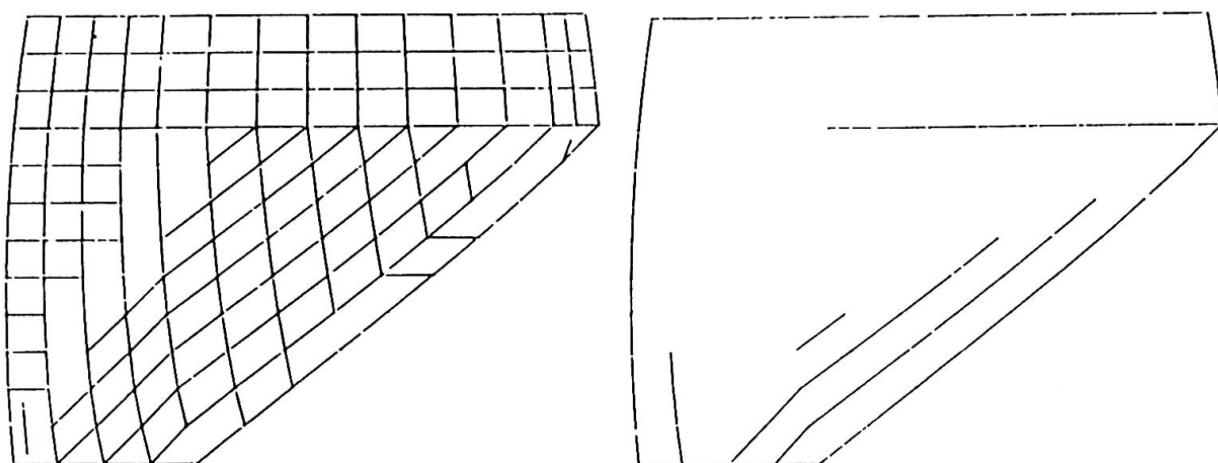


Figure 1: Meshes of the model of an arch dam showing the edges of element junctures exceeding 2° and 5° , respectively

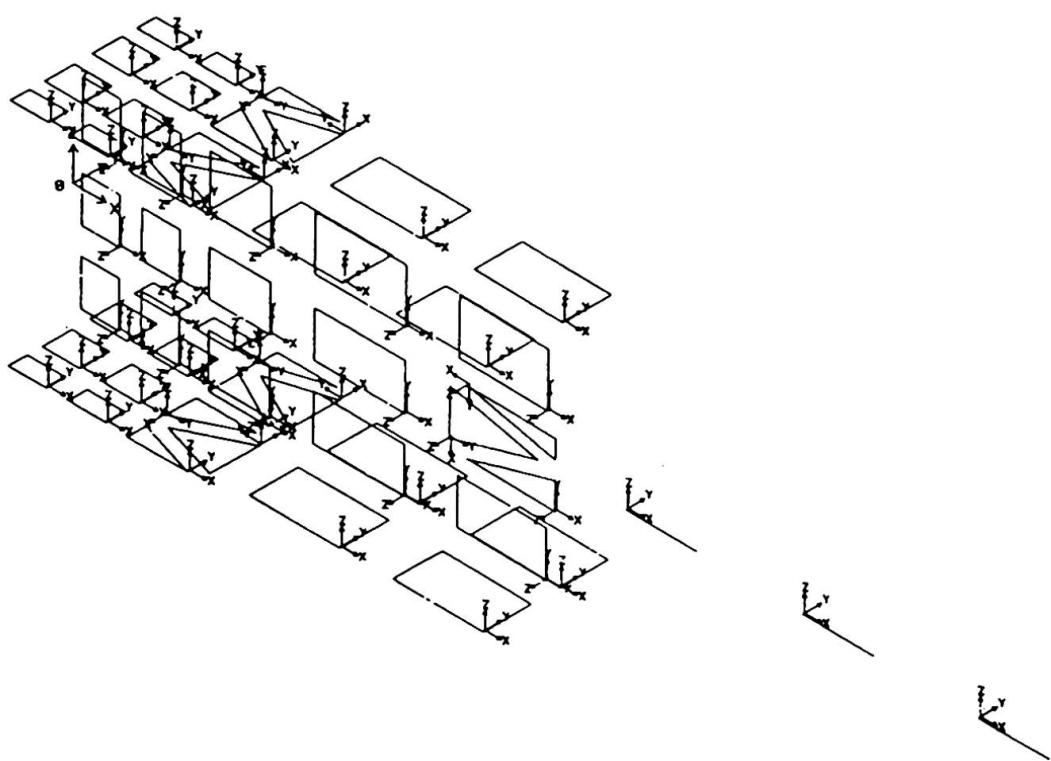


Figure 2: Local coordinate systems



A chronic source of errors is the formulation of the boundary conditions. Therefore the graphic representation of the boundary conditions with respect to the degrees of freedom and of the coordinate system are of great importance. The same is true for linear constraint equations.

Figure 3 shows a shell structure for which the outline and the boundary conditions have been plotted. At each boundary node, the node number and the boundary condition with respect to the constrained degrees of freedom and also the coordinate systems used are displayed.

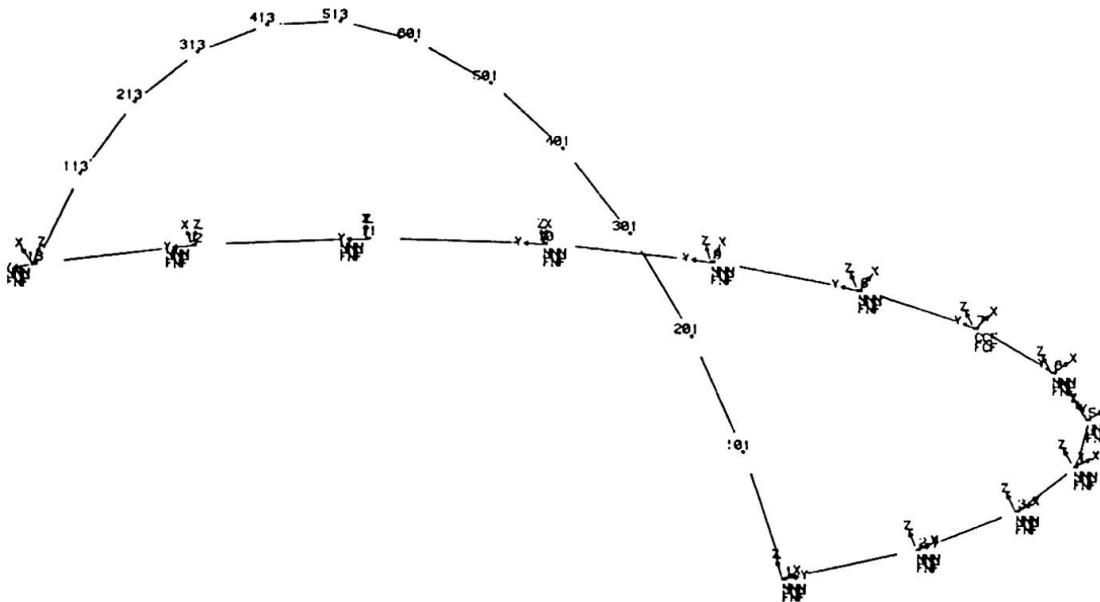


Figure 3: Outline of shell with boundary conditions

3. DISPLAY OF RESULTS

After the analysis, the deformed shape of the structure under the action of the static or dynamic loads is plotted first of all. If the deformed shape is properly scaled, errors in the stiffness properties, the boundary conditions or the loads can frequently be detected right away. Of special interest is the representation of the forces and stresses. For this, either plots along sections or contour-plots can be used. The deformed shape is a means for verifying the kinematic boundary conditions. As a corollary, the stress field permits checking the quality of approximation of the static boundary conditions.

As static boundary conditions are automatically satisfied in pure-displacement models, the quality of the stress field along the boundaries is a measure for the quality of the discretization (Figure 4).

In addition to the above-mentioned structural quantities, the support reactions, strain energies, contents of reinforcement etc. for specific load cases as well as their envelopes can also be plotted. To illustrate this point, Figure 5 shows the extreme values (envelopes) of the support reactions in a flat slab.

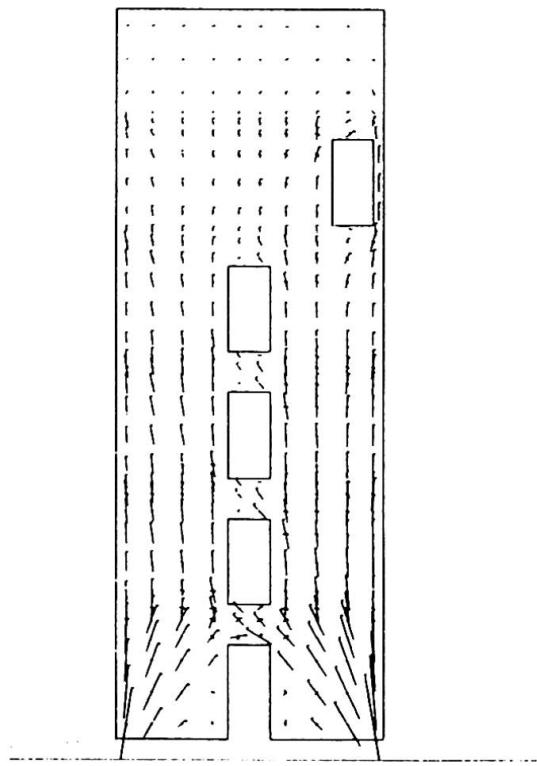


Figure 4: Principal stresses in a shear wall

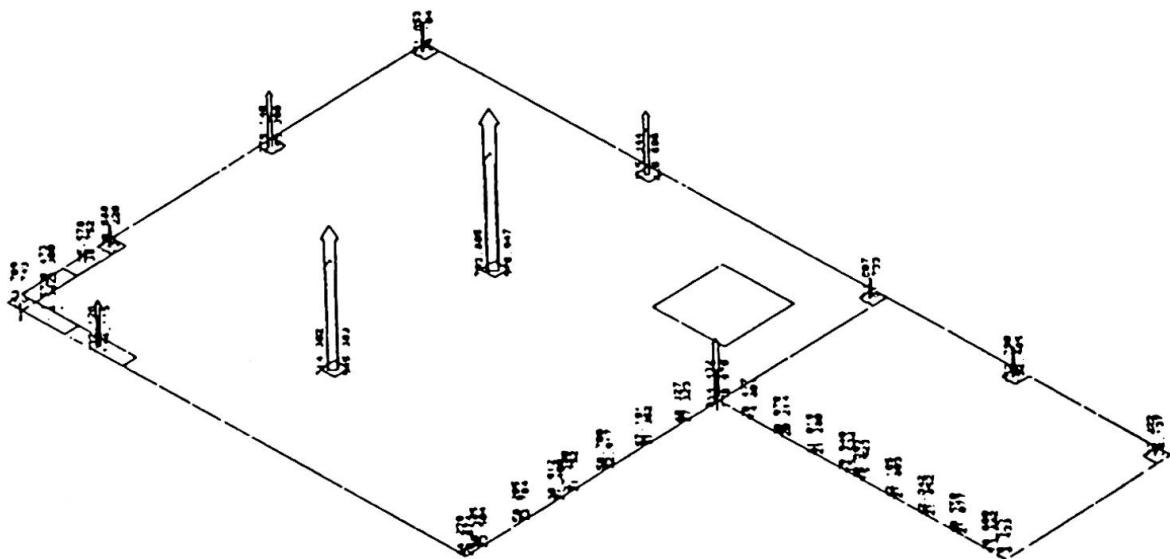


Figure 5: Extreme values of support reactions in a flat slab

4. COMPUTER-AIDED DESIGN AND FE-ANALYSIS

The support given to the engineer by interactive mesh-generators as well as by the graphic display facilities has contributed much to the wide distribution of FE-methods. In recent years, however, the use of CAD-systems in the design process has become more and more popular. Once the structure is designed by such a system, all geometric data are defined. By using the geometric data base, it is possible to support the engineer very effectively in his work in setting up the FE-model. To do so, it is necessary that interfaces be developed between the CAD-system and the FE-system, and that the FE-system be capable of analyzing the structures encountered. In the building-construction industry, for instance, the analysis of slabs, flat slabs, and of slabs with stiffeners and holes has to be dealt with.

Modern CAD-systems also provide the user with a method data base. This method data base has been used in the CAD-system CAD-B to interface the system with FLASH. By means of simple user-oriented commands, the engineer can extract the geometric data and request the automated mesh generation. The system supports the user in many respects. For instance, supporting walls are introduced automatically as support lines. He can also select the appropriate code requirements. The results of the automated mesh generation immediately appear on the screen and can be modified interactively. The data are completed by describing the material properties, the load cases and the output requests.

After the analysis, the results will be displayed on the screen. These results can now be interfaced with the CAD-system. For example by using for the contour lines of the reinforcements, the reinforcement bars or meshes of reinforcement can be designed interactively. As a by-product, the lists for the reinforcement steel and the cutting drawings for the steel meshes are automatically generated.

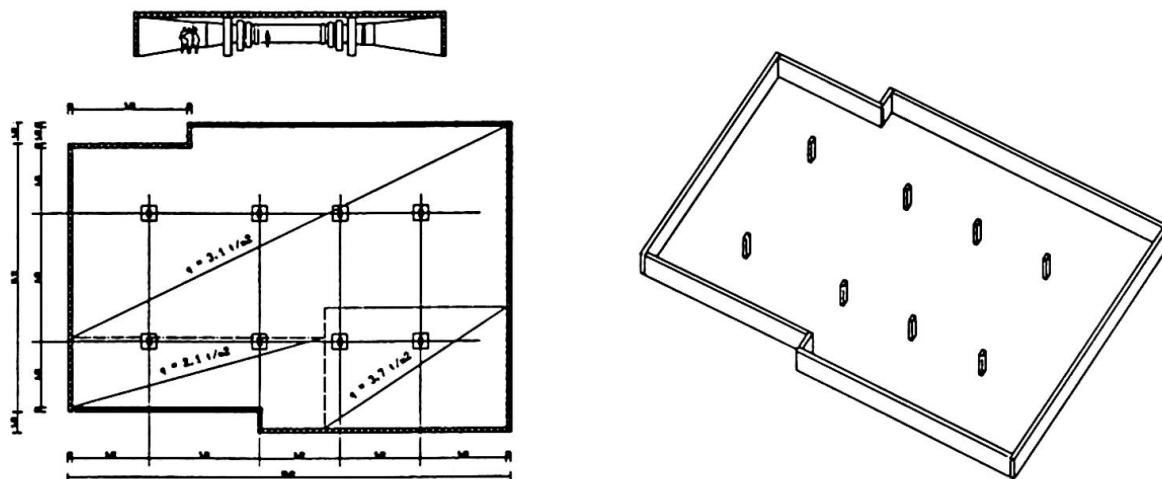


Figure 6: Pedestrian underpass designed with the CAD-system CAD-B

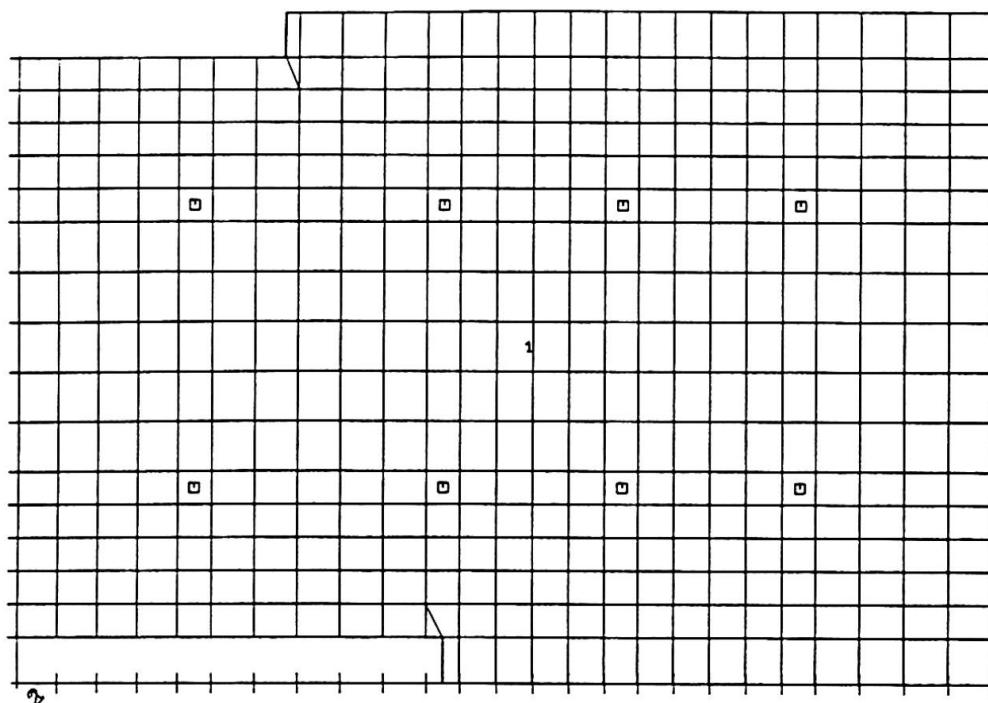


Figure 7: Automatically generated finite-element mesh on the basis of given boundary modes

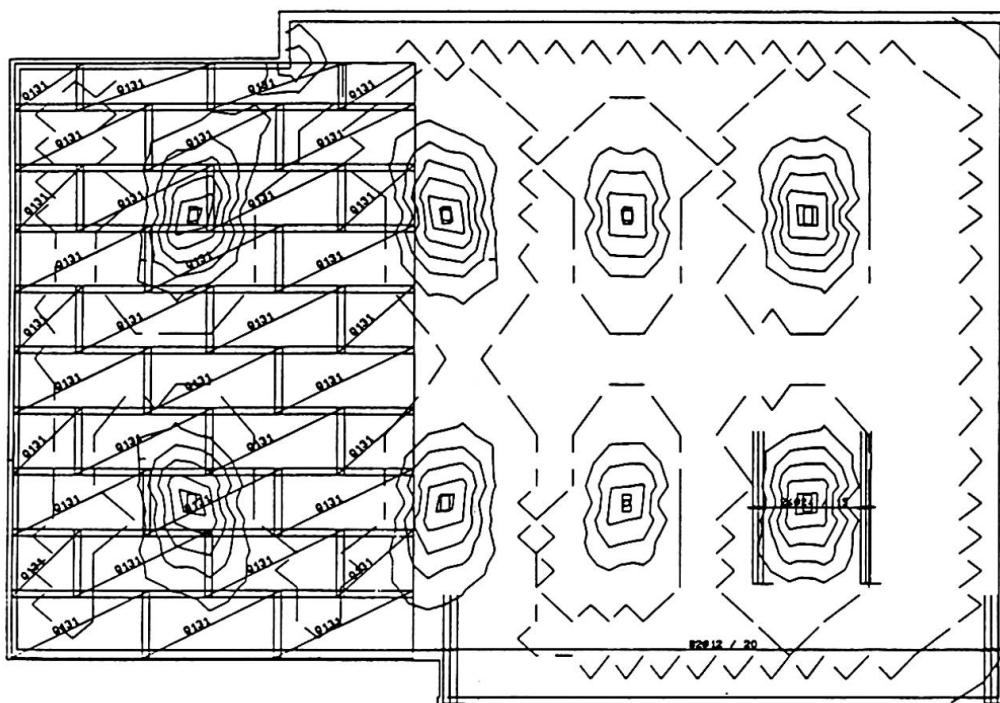


Figure 8: Interactive design of the reinforcement based on the contour lines of required steel as calculated by FLASH

Figures 6 - 8 show the different phases of an integrated CAD-design and the analysis process for a slab.



REFERENCES

1. MERKEL P., HIEGELE E., Gebäudeentwurf, Systembeschreibung. IEZ, Wiesenstr. 4, 6148 Bensheim, Switzerland, January 1984.
2. ODEN J.T., BATHE K.J., A Commentary on Computational Mechanics. Applied Mechanics, Vol. 31, 1978, pp. 1053-1058.
3. PFAFFINGER D., Effective Use of Structural Computer Programs. Proc. IABSE Colloquium "Interface between Computing and Design in Structural Engineering," Bergamo, 1978. Reports of the Working Commissions, Vol. 31, IABSE, Zurich, Switzerland, 1978.
4. PFAFFINGER D., Man-Computer Communication in Structural Analysis. Proceedings, 11th IABSE Congress, Vienna, 1980.
5. PFAFFINGER D., WALDER, U., Einsatz graphischer Hilfsmittel bei der FEM. Tagung "Finite Elemente - Anwendungen in der Baupraxis." TU Munich, March 1984.
6. PIAN T.H.H., Element Stiffness-Matrices for Boundary Compatibility and for Prescribed Boundary Stresses. Proceedings First Conference on Matrix Methods in Structural Mechanics, Dayton, Ohio, October 1965, AFFDL-TR-66-80 (Nov. 1966).
7. POLONY S., REYER E., Zuverlässigkeitsschätzungen und Kontrollmöglichkeiten (Prüfung) zu praktischen Berechnungen mit der Finite-Element-Methode. Die Bau-technik 52, 1975, pp. 374-384.
8. WALDER U., ANDERHEGGEN E., GREEN D., Das Computerprogramm FLASH. IABSE Publications, Vol. 76-II, September 1976.
9. WALDER U., GREEN D., The Finite Element Programs FLASH2 and STATIK. A Handbook of Finite Element Systems. CML Publications, Southampton, 1981.
10. WALDER U., GREEN D., PFAFFINGER D., FLASH Benutzerhandbuch, 7. Auflage. FLASH User Manual, 5th Edition RZW, Tannackerstr. 2, 3073 Gümligen and Forchstr. 21, 8032 Zurich, Switzerland, April 1984.

Computer-Aided Design of Box Girders Using a Simple Non-Linear Technique

Projet de poutres-caissons par l'emploi d'une simple technique non-linéaire

Entwurf von Kastenträgern mit einem einfachen nicht-linearen Verfahren

A.R. CUSENS

Head and Prof. of Civil Eng.
University of Leeds
Leeds, England



Before going to Leeds in 1979, Tony Cusens was Professor at the University of Dundee for 13 years. His main research field is concrete structures, especially bridges and he is the co-author of two books on bridge deck analysis. He is Vice-chairman of the British Group of IABSE and President of the Concrete Society 1983-84.

P. LENGYEL

Chief Engineer
Videoton Fejlesztési Intézet
Budapest, Hungary



Peter Lengyel, born 1953, is a structural engineer and applied mathematician, who received his degrees at universities in Budapest. He is a candidate of technical sciences in bridge engineering and doctor of numerical analysis. He manages a department of CAD programming and has been involved in FEM research.

SUMMARY

The paper describes the application of a geometrical, non-linear, finite strip method to the computer-aided design of box girders. It includes two examples of application of the method. A square isotropic plate is analysed, showing excellent correlation with a finite element solution. Then, using this method, the Danube bridge failure of 1969 is re-examined, confirming the mode of failure established by previous authors, but resulting in different numerical values.

RESUME

Cette étude décrit l'application d'une méthode non-linéaire géométrique de bande finie au projet par ordinateur de poutres-caissons. Elle comprend deux exemples d'application de la méthode. Une plaque carrée isotrope est analysée et montre une corrélation excellente avec une solution par éléments finis. L'analyse de la rupture, en 1969, du pont sur la Danube confirme le mode de rupture établi antérieurement par d'autres auteurs, mais aboutit à des valeurs numériques différentes.

ZUSAMMENFASSUNG

Diese Arbeit präsentiert die Anwendung einer geometrisch nicht-linearen Methode der finiten Streifen für die rechnerunterstützte Projektierung von Kastenträgern. Sie beinhaltet zwei Anwendungsbeispiele der Methode. Eine Rechteckplatte wird analysiert. Eine vollkommene Korrelation mit der Lösung der Methode der finiten Elemente zeigt sich. Danach wird mit Hilfe dieses Verfahrens der 1969 erfolgte Einsturz der Donaubrücke nochmals analysiert. Die von früheren Autoren angegebenen Bruchursachen werden bestätigt. Es ergeben sich jedoch verschiedene numerische Werte.



1. INTRODUCTION

The finite strip method has been applied widely since the early seventies and has been shown to be an efficient tool for the analysis of box girders. The first versions of the method were based on a linear displacement formulation of the finite element procedure using a combination of polynomial and harmonic functions for the solution of simply supported folded plate structures [1]; this was later extended to multi-span structures [6].

The development of a geometrical non-linear finite strip technique has been the subject of research in recent years [3, 4, 8]. However, solutions have been based primarily on special displacement functions and this results in a restricted characterisation of geometrical non-linearity.

The idea of closely following the finite element procedure was put forward by the authors in a previous publication [5]. Since then this technique has been further tested and applied with the development of Fortran programs for different mini- and micro-computers.

This paper is concerned primarily with the presentation of two representative applications of the method. However, a short summary of the technique is given in the Appendix.

The examples in this paper demonstrate the analysis of steel structures with longitudinal stiffeners, simply supported at two opposite ends (Fig. 1). For simple supports, the following conditions are assumed: the vertical displacement (w) of the plate is perpendicular to its plane, the transverse rotation (θ) and the transverse displacement (u) are zero, while the longitudinal displacement (v) is non-zero.

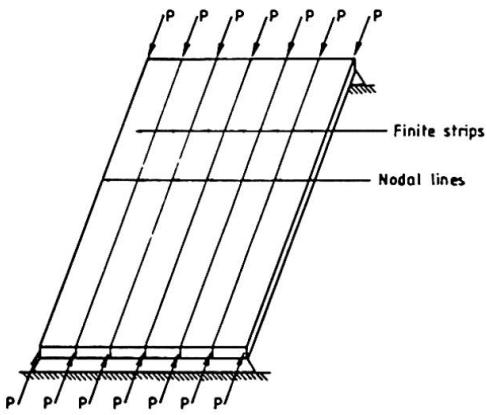


Fig. 1

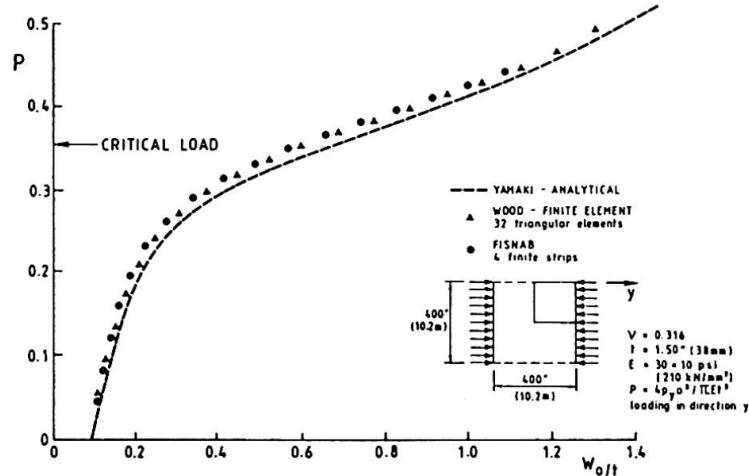


Fig. 2

The aim of the analysis is primarily to investigate the greatest lateral displacement and the corresponding stress distributions of the plate under increasing load in the longitudinal plane.

The examples in this paper were solved by applying only one harmonic; however the effect of more than one harmonic can also be found. In both examples an initial central displacement was assumed by applying a vertical concentrated load P_0 at the mid-point of the plate.

The results were obtained by running the Fortran program FISNAB which has been implemented on Leeds University's PRIME 750 and VIDEOTON's VT 600 computers.

2. ANALYSIS OF A SIMPLY SUPPORTED RECTANGULAR PLATE

Figure 2 shows the case of a rectangular plate simply supported at four edges. The support conditions of the horizontally loaded edges have already been described. The unloaded edge conditions were obtained by prescribing the vertical displacement of the plate in these nodal lines as zero, while the other three displacements - θ , u and v - were not prescribed.

This is a problem for which several solutions are available in the literature, thus providing useful comparisons. The square plate analysed by Yamaki [12] and Wood [11] is used here. Its dimensions are 400 x 400 in by 1.5 in thick (10 m x 10 m x 38 mm) and the initial central displacement is 0.15 in (3.8 mm). Figure 2 shows the relation between this ratio of the central displacement and plate thickness and the quantity P characteristic of the compressive loading. This quantity is approximated by Wood [11] and Yamaki [12] as:

$$P = \frac{4p_y a^2}{E t^2}$$

where p_y is the average compressive stress in direction y . The dashed line shows the analytical result of Yamaki, triangles denote the result of the finite element analysis of Wood, who investigated one quarter of the plate using 32 triangular elements. The results from the FISNAB program are shown by circles and are based upon the application of four finite strips and one harmonic. Advantage was taken of the symmetrical nature of the problem and one half of the plate only was analysed.

3. ANALYSIS OF THE FAILURE OF THE VIENNA DANUBE BRIDGE

3.1 In the evening of 6th November 1969 the Vienna Danube bridge, while under construction, failed at three sections. The bridge had two intermediate supports forming spans of 82, 210 and 120 metres. The structure of the bridge was formed of two torsionally stiff box girders of 7.6 m width and 5.2-7.5 m height, at 15.6 m spacing. The box girders supported a 32 m wide orthotropic plate deck (Fig. 3).

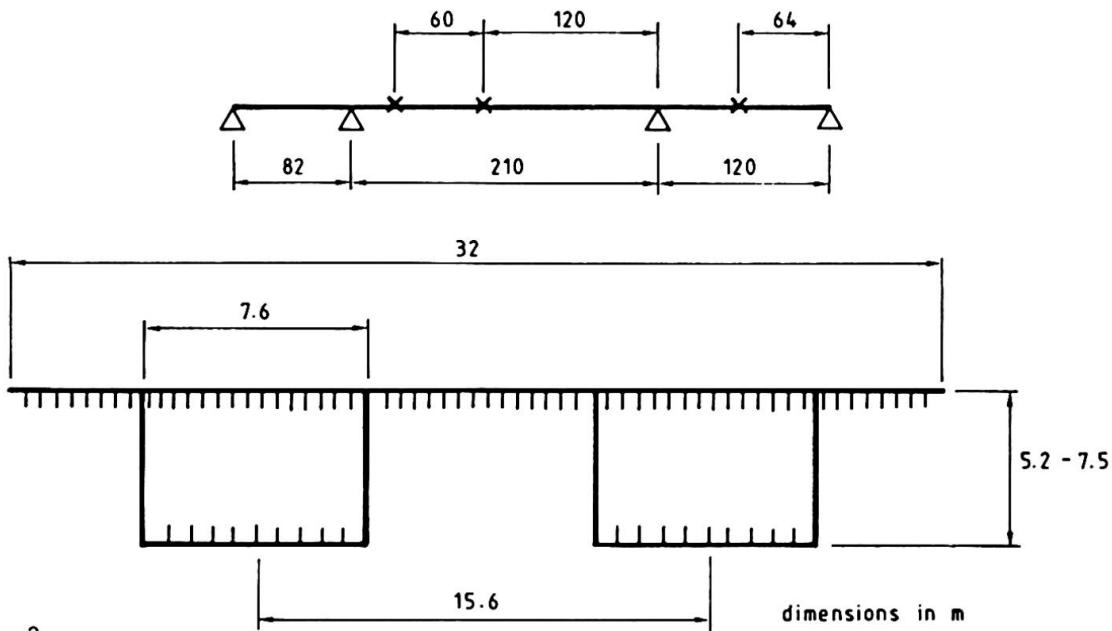


Fig. 3

The collapse of the bridge launched a fierce technical debate especially because the dimensioning and construction were carried out following the current Austrian specifications.



Prof. P. Cicin of Vienna, Prof. K. Sattler of Graz and Prof. P. Roik of Berlin, published different explanations in the journal "Tiefbau" in 1970 [2, 9, 10]. Further research on this problem at the University of Liege was later published by Maquoi and Massonnet [7]. Based primarily on this paper the change of displacement of the lower flange mid-point with increasing longitudinal horizontal loading was analysed together with the stress distribution in the cross-section corresponding to each of the loading values. These results are presented in the following sections.

3.2 According to a nearby observatory, the fracture occurred at three cross-sections, at five second intervals, at 8.44 p.m. The middle closing member had been put in at 8 a.m. when the observatory temperature was 4.2°C . The temperature had risen by 2.2°C by 2 p.m., and then decreased by 4.9°C . It is reasonable to assume that at the bridge site a somewhat greater increase of $8-10^{\circ}\text{C}$ occurred with a later decrease of $12-15^{\circ}\text{C}$.

The most important factor in considering the cause of the failure is the greater increase of loading of the bridges compared with the expected load because of:

- a) the significant change of ambient temperature;
- b) the unexpectedly non-uniform distributed dead load.

It is apparent that initially the entire lower flange of the box girder failed and the full collapse of the cross section followed afterwards.

3.3 Based on these considerations, Maquoi and Massonnet have analysed the yielding of the lower flange.

The most important aim of the analytical method devised was to take into consideration the essential deviation of the stress distribution from a uniform one in the lower flange. This deviation is caused by three factors:

- a) The shear lag effect, i.e. consideration of the deformations due to the shearing forces;
- b) The curvature of the entire flange, increasing under the compressive forces;
- c) Curvature of the plate between individual longitudinal stiffeners.

The combination of these factors results in the theoretical distribution of the longitudinal stresses of the stiffened plate shown as curve II of Fig. 4. If the effect of c) is omitted, then curve I is obtained, while line A denotes an approximately uniform distribution.

Maquoi and Massonnet took the factors a) and b) into consideration by transforming the lower flange into an orthotropic plate and investigating this with one harmonic according to non-linear analysis. In this way a non-uniform longitudinal stress distribution was obtained. Since this method could not follow exactly the effect c), the authors investigated a segment of plate between two longitudinal stiffeners, which had an initial deformation.

They took into consideration the approximate effect of this curvature by the introduction of an effective plate width. The approximate nature of this investigation was caused by the combination of a final common reducing factor from individual factors from two separate analyses expressed simply as a multiple of the two. Then the work concentrated on the presentation of the two latter multiplying factors. This was done within plate theory by the assumption of the displacement function:

$$w = \cos \frac{x}{a} \cdot \cos \frac{y}{b}$$

Maquoi and Massonet state in their analysis of the state of collapse that the external, longitudinal stress consisted of:

$$\sigma_{\text{dead load}} = 194 \text{ N/mm}^2 \quad (\text{MPa})$$

$$\sigma_{\text{change in temp.}} = 26 \text{ N/mm}^2$$

forming a total applied stress of:

$$\sigma_{\text{total}} = 220 \text{ N/mm}^2$$

The critical load was:

$$\sigma_{\text{crit}} = 217 \text{ N/mm}^2$$

while the yielding limit was:

$$\sigma_{\text{yield}} = 285 \text{ N/mm}^2$$

As Maquoi and Massonet obtained:

$$\sigma_1 = 0.811 ; \sigma_2 = 0.848$$

for the two multiplying factors, they derived the value:

$$\sigma_{\text{limit}} = 0.811 \times 0.848 \times 285 = 196 \text{ N/mm}^2$$

i.e., if the stress, assumed to be uniform along the cross-section, reached this value, then at the marginal point of the stress distribution denoted by curve II in Fig. 4, the cross-section yields.

This investigation contains a number of approximations since the simplified analysis is not able to follow exactly the geometrical form of the cross-section.

3.4 Based on the previous analysis it was an interesting task to plot the entire stress distribution of the cross-section as a function of the longitudinal loading using the FISNAB program.

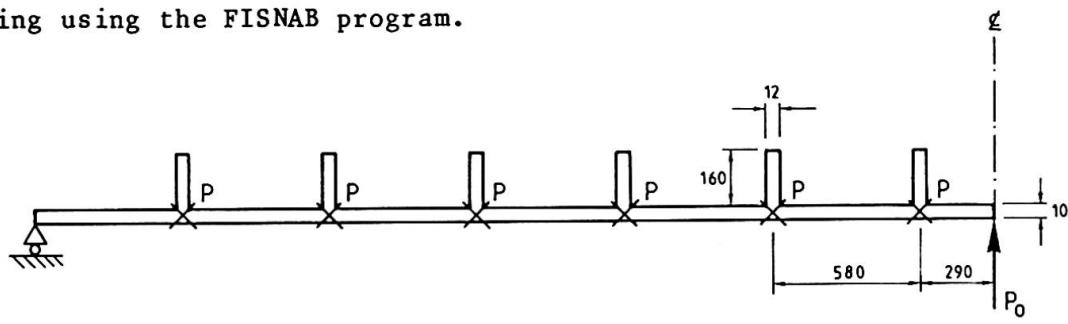


Fig. 5

dimensions in mm



The lower flange was modelled as shown in Fig. 5 making use of the symmetrical geometry of the section. The two extreme points of the cross-section were assumed to have no vertical displacement because of the supporting effect of the webs. The lower flange was considered to be a part of a box girder segment bounded by two stiff diaphragms, and having no intermediate transverse stiffeners. At the two bounding stiffening diaphragms the flange was assumed to be simply supported. A vertical load applied at the centre point of the plate approximated the initial curvature of the lower flange. The geometrical data of the structure are given in Fig. 5.

By increasing the longitudinal horizontal loading formed by concentrated, longitudinal loads along the nodal lines indicated as denoted in Fig. 5, two graphs have been plotted.

The first, Fig. 6, shows the values of longitudinal stress at the centre of the box girder segment plotted against the applied stress due to the longitudinal loads. The two curves show:

- Maximum stress at the central cross-section of flange;
- Minimum stress in the central cross-section of the flange, i.e., at the central point of the flange.

These values correspond to the maximum values of displacement.

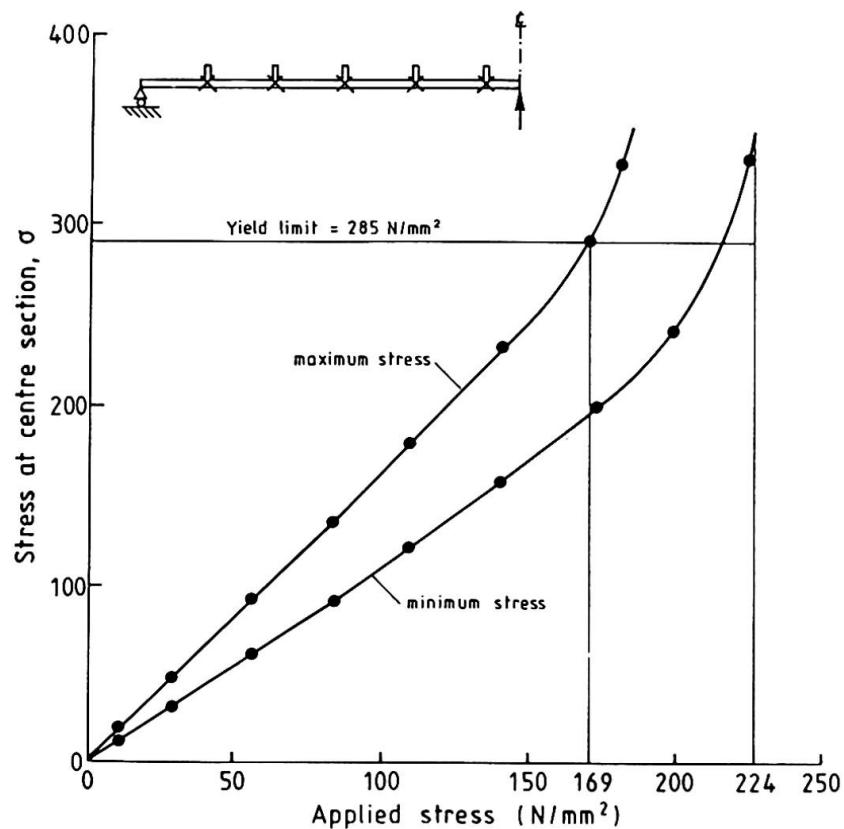


Fig. 6

A horizontal line at the yield value 285 N/mm^2 is drawn in Fig. 6. The point at which the curve of maximum stress crosses this line is the limiting loading stress corresponding to yielding. This value is 169 N/mm^2 . The critical load would correspond to a mid-point stress of 224 N/mm^2 .

Figure 7 shows the distribution of longitudinal stress over the entire cross-section. This figure shows that upon increase of the applied loading

stress, the longitudinal stress increases more rapidly at the line of the cross-section under the webs as compared with the stress increase at the centre line. This means that the curvature of curve I in Fig. 4 is even more pronounced, while the shear lag effect between longitudinal stiffeners (curve II) can also be seen in Fig. 7, particularly at the centre-line.

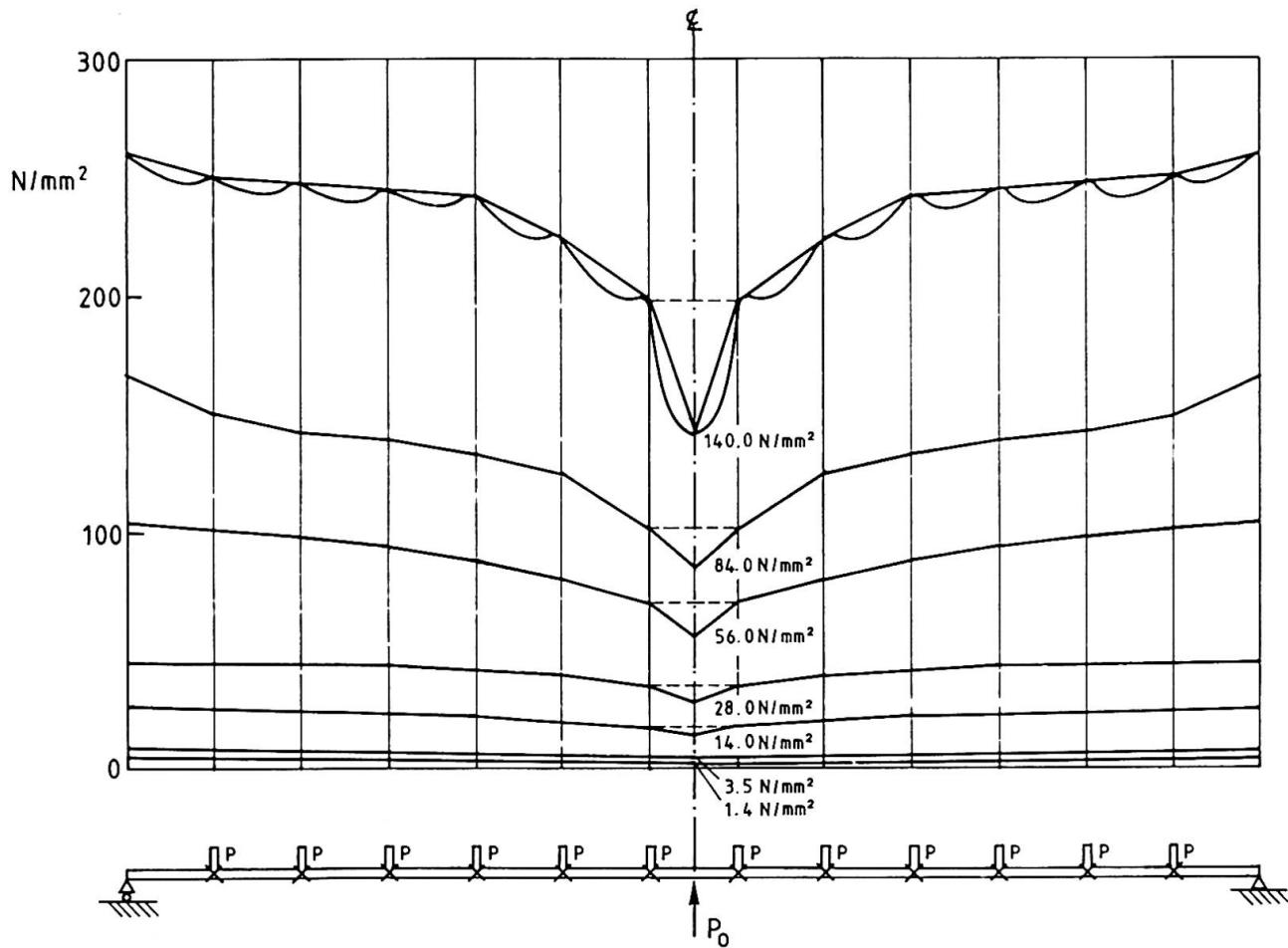


Fig. 7

To summarise, it can be concluded that if the lower flange of the Vienna Danube bridge is analysed by a direct study of the real stress distribution, then:

- the permissible loading stresses given by Sattler [10] are actually 50 N/mm² above the yield limit;
- the limit stress is 169 N/mm², i.e. smaller than the 196 N/mm² obtained by Maquoi and Massonnet [7];
- the collapse was initiated by yield at the web/flange junction of the lower flange.

4. ACKNOWLEDGEMENT

The development and testing of the method were achieved with the aid of Professor Halász, Head of Department of Steel Structures, Technical University, Budapest. The second example was suggested by Professor Halász in the light of



his own studies of this problem. His help is greatly appreciated.

The facilities of, and permission for publication by, the VIDEOTON Development Institute are gratefully acknowledged.

5. REFERENCES

- [1] Y.K. Cheung, Finite Strip Method in Structural Analysis, Pergamon, New York, 1976.
- [2] P. Cicin, *Betrachtungen über die Bruchursachen der neuen Wiener Donaubrücke*, Tiefbau (665-674), 1970.
- [3] T.R. Graves-Smith and S. Sridharan, A Finite Strip Method for the Post-Locally-Buckled Analysis of Plate Structures, *Int. J. Mech. Sci.*, 20 (833-842), 1978.
- [4] G.J. Hancock, Non-Linear Analysis of Thin Sections in Compression, Research Report R 355, University of Sydney, 1979.
- [5] P. Lengyel and A.R. Cusens, A Finite Strip Method for the Geometrically Non-Linear Analysis of Plate Structures, *Int. J. Num. Meths. Engg.*, Vol. 19 (331-340), 1983.
- [6] Y.C. Loo and A.R. Cusens, The Finite Strip Method in Bridge Engineering, Viewpoint Publications, Cement and Concrete Association, 1978.
- [7] R. Maquoi and Ch. Massonnet, *Théorie non-linéaire de la résistance postcritique des grades poutres en caisson raidies*. IABSE Publications 31-II, (91-140), 1971.
- [8] R.J. Plank and W.H. Wittrick, Buckling under Combined Loading of Thin, Flat-Walled Structures by a Complex Finite Strip Method, *Int. J. Num. Meths. Engg.*, 8 (323-339), 1974.
- [9] K. Roik, *Nochmals: Betrachtungen über die Bruchursachen der neuen Wiener Donaubrücke* (1152), 1970.
- [10] K. Sattler, *Nochmals: Betrachtungen über die Bruchursachen der neuen Wiener Donaubrücke* (948-950), 1970.
- [11] R.D. Wood, The Application of Finite Element Methods to Geometrically Non-Linear Structural Analysis, Ph.D. Thesis, University of Wales, Swansea, 1973.
- [12] N. Yamaki, Postbuckling Behaviour of Rectangular Plates with Small Initial Curvature Loaded in Edge Compression, *J. Appl. Mech.* (407-414), 1959.

6. APPENDIX – THE GEOMETRICAL NON-LINEAR FINITE STRIP METHOD

Consider the finite strip denoted by I having the length of b and the width of a of Fig. A1.

Two nodal lines belong to the strip denoted by i and j . The method assumes four displacement components characterised by the following terms:

$$w(x, y) = \sum_{m=1}^r (\alpha_1 + \alpha_2 x + \alpha_3 x^2 + \alpha_4 x^3) \sin k_m y$$

$$\theta(x, y) = \sum_{m=1}^r (\alpha_2 + 2\alpha_3 + 3\alpha_4 x^2) \sin k_m y$$

$$u(x, y) = \sum_{m=1}^r (\alpha_5 + \alpha_6 x) \sin k_m y$$

$$v(x, y) = \sum_{m=1}^r (\alpha_7 + \alpha_8 x) \cos k_m y$$

where $k_m = \frac{m\pi}{a}$

The displacement parameters belonging to the strip I form the vector \underline{e}^I , where:

$$\underline{e}^I = \sum_{m=1}^r \underline{e}_m^I = \sum_{m=1}^r [u_{im}, v_{im}, \theta_{im}, u_{jm}, v_{jm}, w_{jm}, \theta_{jm}]^T$$

These enable us to describe the displacement of nodal lines i and j , utilising the above approximations, by:

$$w_i = \sum_{m=1}^n w_{im} \sin k_m y, \quad w_j = \sum_{m=1}^n w_{jm} \sin k_m y$$

The strain vector of strip I, and harmonic m , contains a linear and non-linear component and takes the form of:

$$\underline{\varepsilon}_m^I = \underline{\varepsilon}_{om}^I + \underline{\varepsilon}_{Nm}^I$$

The corresponding strain-matrix components are:

$$\underline{\underline{B}}_m^I = \underline{\underline{B}}_{om}^I + \underline{\underline{B}}_{Nm}^I$$

where

$$d\underline{\varepsilon}_m^I = \underline{\underline{B}}_m^I d\underline{e}_m^I$$

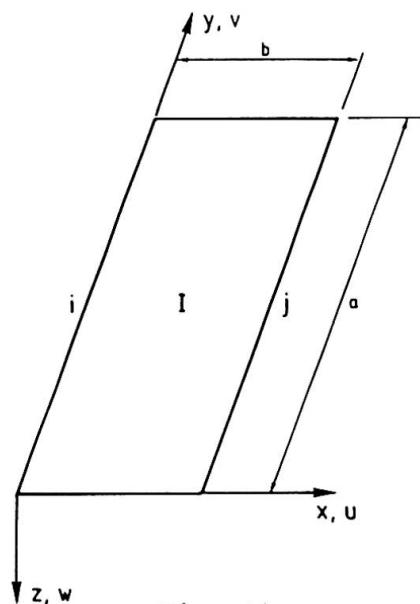


Fig. A1

Following the non-linear finite element solution of [11], the summation of internal and external forces for the total structure and harmonic m results in:

$$\Psi_m = \sum_{I=1}^N \sum_{n=1}^r \int_0^a \int_0^b (B_n^I)^T \sigma_m^I dx dy - \sum_{I=1}^N r_m^I = 0$$

Here the vector σ_m^I contains the stress components, the vector r_m^I , the loading components for harmonic m and strip I .

Making use of the connection of:

$$\sigma_m^I = \underline{D}^I \underline{B}'_m \underline{e}_m^I$$

where \underline{D}^I denotes the elasticity matrix, \underline{B}'_m^I , the strain matrix for non-differential quantities, the following expression is obtained:

$$\Psi_m = \sum_{I=1}^N \sum_{n=1}^r \int_0^a \int_0^b (B_n^I)^T \underline{D}^I \underline{B}'_m^I \underline{e}_m^I dx dy - \sum_{I=1}^N r_m^I$$

where again:

$$\underline{B}'_m^I = \underline{B}'_{om}^I + \underline{B}'_{Nm}^I (\underline{e}_1^I, \underline{e}_2^I, \dots, \underline{e}_r^I)$$

The state of equilibrium of the analysed structure is characterised by the vector series of displacement amplitudes, e_1, e_2, \dots, e_r for the whole structure resulting in a zero vector for:

$$\underline{\Psi} = \sum_{m=1}^r \underline{\Psi}_m$$

The Newton-Raphson method is applied, represented here by:

$$\frac{d\Psi}{de_m^I} = \underline{K}_{Tm}^I$$

where:

$$\underline{K}_{Tm}^I = \underline{K}_{om}^I + \underline{K}_{Nm}^I + \underline{K}_{om}^I$$

and

$$\underline{K}_{om}^I + \underline{K}_{Nm}^I = \sum_{n=1}^r \int_0^a \int_0^b (B_n^I)^T \underline{D}^I \underline{B}'_m^I dx dy$$

with

$$\underline{K}_{om}^I = \sum_{n=1}^r \int_0^a \int_0^b d(B_n^I [e_1^I, e_2^I, \dots, e_r^I])^T \sigma_m^I dx dy$$

and

$$\underline{K}_{om}^I = \underline{K}_{om}^I \cdot de_m^I$$

The last equations define an iterative solution for the determination of vectors e_m ($m = 1, 2, \dots, r$, $I = 1, 2, \dots, N$) resulting in equilibrium based on a non-linear relationship between strains and displacements.

Système intégré de CAO au sein d'une entreprise de travaux publics

Integriertes CAD-System in einer Bauunternehmung

Integrated CAD System in a Construction Company

François MONSARRAT

Bouygues

Clamart, France



Michel DEGUINE

Bouygues

Clamart, France



RESUME

Le Groupe a entrepris depuis deux ans la définition des spécifications et la réalisation du logiciel BDGS. Cette démarche vise à couvrir les besoins de tous les participants à l'acte de concevoir et de construire dans le domaine du bâtiment des travaux publics, et de la construction navale. Cet article expose les grandes lignes du logiciel BDGS, en insistant tout particulièrement sur le domaine de la conception, de l'étude, de la fabrication et de l'exploitation des structures.

ZUSAMMENFASSUNG

Seit zwei Jahren arbeitet die Gruppe an der Definition der spezifischen Bedingungen sowie der Erstellung der BDGS-Software. Hiermit soll den Anforderungen all jener entsprochen werden, die im allgemeinen Bauwesen, im öffentlichen Bauwesen sowie im Schiffsbauwesen an Planung und Konstruktion beteiligt sind. Im Rahmen dieses Artikels legen wir die grossen Linien der BDGS-Software dar. Unsere Unterstützung erstreckt sich vor allem auf das Gebiet der Planung, der Entwicklung, der Fabrikation und der Nutzung der Bauten.

SUMMARY

The Group began defining the specifications for the BDGS software package two years ago. BDGS is designed for the needs of all those who participate in design and construction in building, public works and shipbuilding. This article outlines the main features of the BDGS software, and particularly emphasizes its use in the realm of conception, design, manufacturing and exploitation of structures.



1. LES OBJECTIFS DU PROJET BDGS

Quelques idées de base ont présidé à l'écriture de BDGS :

1. Un ouvrage est le plus souvent un PROTOTYPE.

Les performances de BDGS sont optimisées dans les fonctionnalités les plus utilisées : celles qui simulent les processus d'études les plus courants.

2. Les règlementations sont multiples

BDGS est conçu pour s'adapter à d'autres règlements et se connecter à d'autres logiciels.

Les logiciels d'application sont modulaires, dissociant les couches de logiciel générales (calculs géométriques, résistance des matériaux, etc...), des couches de logiciel spécifiques à un règlement, une norme, un client ...

L'utilisateur peut inclure ses propres logiciels ainsi que des données générées par des produits informatiques différents.

La part de logiciel liée au système d'exploitation est complètement dissociée des programmes constitutifs de manière à faciliter la transportabilité du Produit.

3. Pluralité des intervenants

BDGS est appelé à être utilisé par une grande diversité de professions ; de ce fait il permet d'englober TOUTES les ACTIONS participant à l'ACTE de CONCEVOIR, et, par des logiciels techniques appropriés, de tirer la maquette tous les enseignements : vues, coupes, perspectives, métrés, calculs techniques etc...

BDGS assure le PARTAGE des DONNEES entre de nombreux utilisateurs en gérant les problèmes de SECURITE et de COHERENCE.

4. Coordonnation des interventions

La conception et l'étude d'un ouvrage font appel à un grand nombre d'intervenants de spécialités diverses :

Afin d'éviter l'introduction d'incohérences graves, BDGS offre des possibilités de mise en commun des informations et leur validation au niveau le plus élevé (celui du Chef de Projet), en respectant les deux impératifs ci-dessous :

- L'INDEPENDANCE DES OPERATEURS : Chacun dispose des moyens géométriques, topologiques et technologiques nécessaires à la construction de ses propres maquettes.
- La possibilité de FEDERER au sein d'un projet l'ensemble des parties qui le composent : Un opérateur privilégié (le chef du projet) dispose de moyens permettant de rendre publics, et d'imposer au niveau du projet, les objets, matériaux, attributs et paramètres communs. Ainsi peuvent être assurées l'homogénéité et l'unicité des informations liées au projet.

PARTAGE	UNICITE DES INFORMATIONS
SECURITE	SAUVEGARDE CONFIDENTIALITE INTEGRITE

2. LA MAQUETTE ELECTRONIQUE

2.1 COMMENT EST CONSTITUÉE LA MAQUETTE

La démarche CAO consiste à associer à un ouvrage une **MAQUETTE ELECTRONIQUE TRIDIMENSIONNELLE** intégrant la totalité des données qui le définissent.

Le système est capable de construire cette maquette par assemblage de tous les composants de l'ouvrage, archivés dans des BIBLIOTHEQUES d'objets bi et tri-dimensionnels dont la géométrie peut être paramétrée.

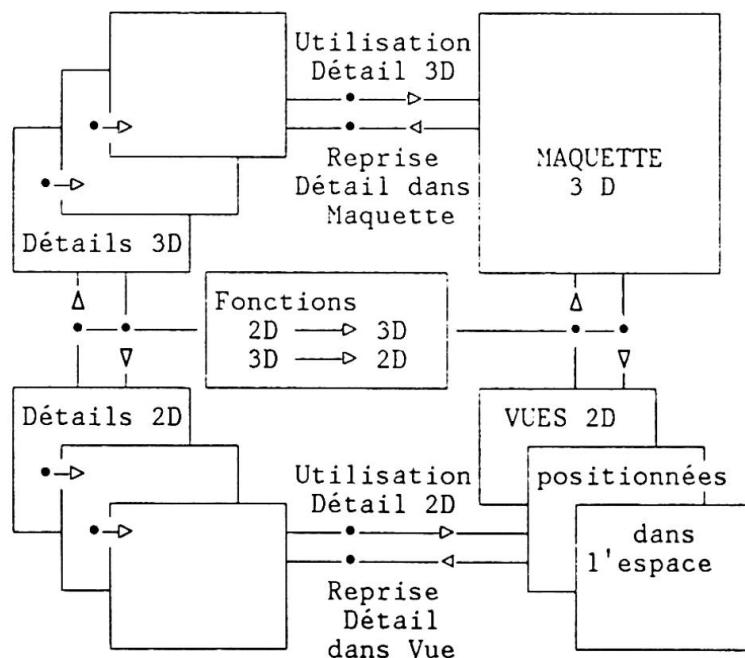
Le système a connaissance des réalités technologiques de chaque matériau, et permet la résolution correcte de leurs interactions et l'exploitation ultérieure de la maquette par l'ensemble des programmes d'application.

Le système est capable à tous moments de modifier simplement la maquette électronique, sans que l'utilisateur ait à redéfinir tous les assemblages.

Pour rendre possible l'étude des plus grands projets, le système peut travailler sur un ESPACE MEMOIRE pratiquement ILLIMITE. L'opérateur peut, à un instant donné, travailler sur un sous-ensemble de cette maquette résidant en unité centrale.

La maquette associe :

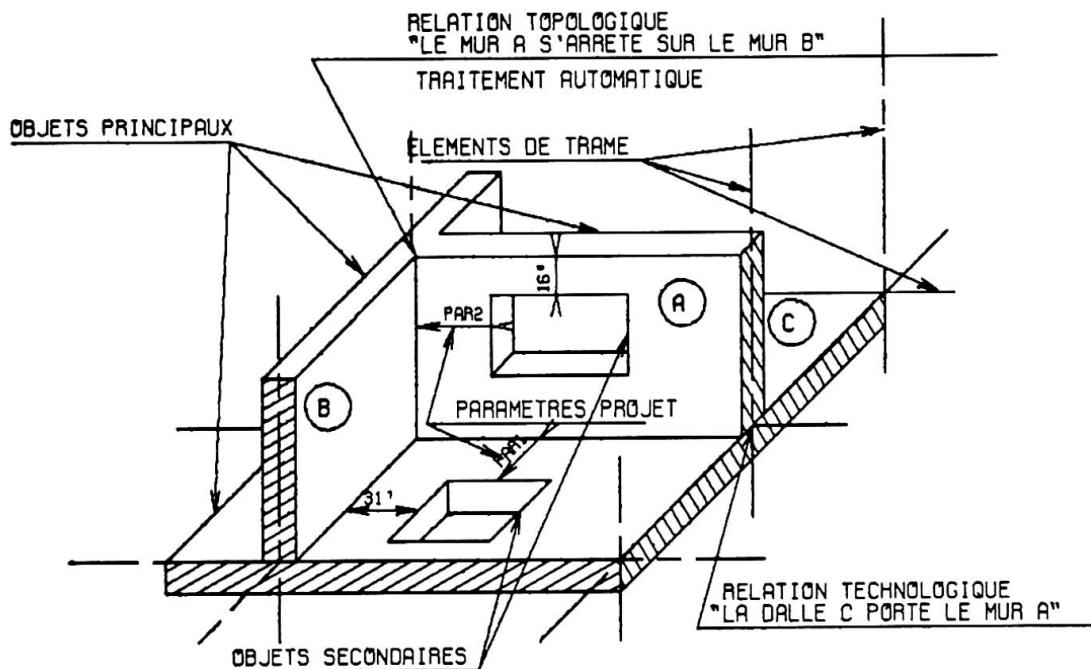
- des vues planes positionnées dans l'espace et constituées d'objets 2D ou de traces d'objets 3D,
- des objets tridimensionnels,
- des **DETAILS BIDIMENSIONNELS**, pouvant être assimilés aux feuilles d'un carnet de croquis, et utilisés au sein d'une vue plane ou d'un autre détail,
- des **DETAILS TRIDIMENSIONNELS**, constitués d'éléments 3D et se comportant, vis-à-vis de la maquette, comme les détails 2D vis-à-vis des vues planes.



Le logiciel associe des fonctions 3D sophistiquées (algèbre générale des volumes et des surfaces), et des fonctions spécifiques pour les "objets du bâtiment" les plus utilisés.

2.2 LES OBJETS DE BDGS - LES RENSEIGNEMENTS QU'ILS PORTENT

Avant de situer un objet dans la BASE de DONNEES graphiques et non graphiques qui constitue la maquette, il est utile de donner quelques définitions des CONCEPTS DE BASE qui ont présidé à la structuration du logiciel BDGS :



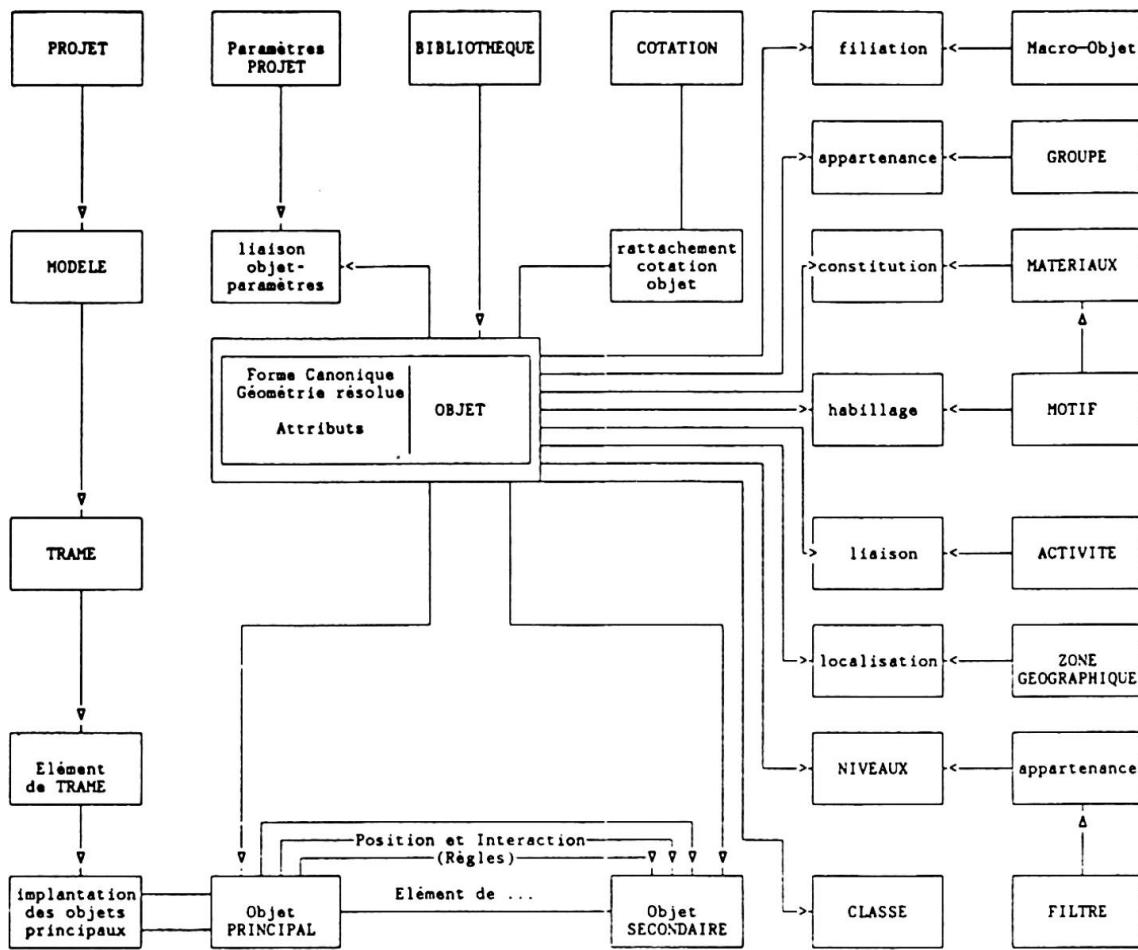
- **PROJET** : Il regroupe et met à disposition de chaque opérateur l'ensemble des objets, informations, standards, normes ,etc... communs à toutes les maquettes qui le détaillent.
- **BIBLIOTHEQUE** : Collection des objets utilisés d'une façon répétitive dans les diverses maquettes, elle fait partie intégrante du projet. Elle peut être constituée à partir de bibliothèques externes plus générales, et enrichie dans le cadre du projet lui-même.
- **LA TRAME** : Ensemble des plans et cylindres verticaux ("axes") et des plans ("niveaux") et cylindres horizontaux qui génèrent une série structurée de référentiels, communs à l'ensemble des maquettes constituant le projet.
- **OBJET PRINCIPAL ET OBJET SECONDAIRE** :
Un objet principal est positionné par rapport à la trame (exemple : murs, poutres, semelles, poteaux ...).
Un objet secondaire appartient à un objet principal et est positionné par rapport à lui (exemple : fenêtre dans mur, trémie dans dalle)
- **PARAMETRE** : Association d'un identificateur et d'une valeur numérique. L'utilisateur, pour définir dans la maquette une épaisseur, une position relative, etc... pourra utiliser l'identificateur d'un paramètre projet en lieu et place de sa valeur. Toute modification de la valeur d'un tel paramètre a des conséquences automatiques sur l'ensemble de la maquette.
- **HISTORIQUE**: Le mode de construction, basé sur une topologie de liaisons et gardé en mémoire doit permettre, en cas de modifications locales ou globales du projet, de remettre à jour automatiquement la maquette.
- **RELATION** :
 - Relation TOPOLOGIQUE
 - Relation TECHNOLOGIQUE: BDGS sait traiter globalement des entités complexes, et porter une connaissance technologique de haut niveau (poutre portée par... et portant..., liaison par encastrement...)
- **REGLES** : Mise en mémoire des règles topologiques régissant les interactions entre objets (qui peuvent utiliser la table des priorités entre matériaux).

- **MACROS** : Sauvegarde de séquences d'interactions aboutissant à la création d'un objet plus ou moins complexe susceptible d'un paramétrage (par paramètres externes ou utilisation de paramètres-projet).
- **MATERIAUX** : Table contenant les descriptions des matériaux utilisés dans un projet, ainsi que tous attributs physiques ou textuels qui leur sont liés. Leur sont associés des Motifs de représentation, ainsi qu'une table de priorités relatives.
- **DETAILS** : Bibliothèque d'objets bi et tridimensionnels. Leur sont associés des Attributs, des Matériaux, des Règles, des Classes, des Activités.
- **MOTIFS** : Représentations surfaciques (hachures, grisés, couleurs, motifs quelconques).
- **COUCHES** : (Layers) Mise en mémoire de l'appartenance d'un objet à un sous ensemble défini d'une façon externe, lequel pourra être traité globalement en visualisation ou/et en plotting.
- **FILTRES** : Mise en mémoire de modes de visualisation diverses.
- **ATTRIBUTS** : Table associant des identificateurs accessibles de façon externe (densité, conductibilité ...) et leurs valeur numériques.
- **CLASSES** : Table permettant le classement des objets en familles ayant les mêmes attributs (éléments de façade, porteurs ...).
- **ACTIVITES** : Table associant objets et activité d'étude (poutre -> élément pour le calcul de structure)
- **COTATION** : La cotation n'est pas un simple renseignement graphique porté sur le plan, perdant toute liaison avec les objets cotés, mais un élément à part entière de la maquette électronique.

2.3 LA RICHESSE DU CONCEPT D'OBJET

OBJET	EST DEFINI	TYPE	GEOMETRIQUE PHYSIQUE TEXTUEL
		ATTRIBUTS TOPOLOGIE CARACTERISTIQUES	Propres (Matériaux ...)
	PROVIENT de ou EST GENERE par		une Bibliothèque un Projet des Paramètres projet un Objet paramétré père ses Liaisons avec autres
			un Macro-objet un Niveau un Filtre de visu. des Groupes des Extraits une Classe des Activités des Domaines géographiq.
	INCLUS dans ou APPARTIENT à		
	LIE à		d'autres objets la Trame la Cotation
	a une HISTOIRE		

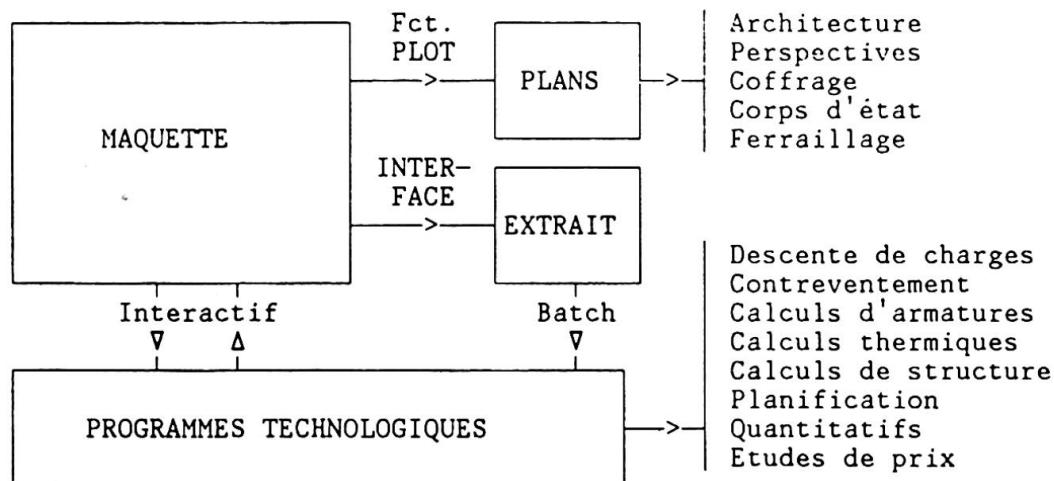
2.4 LA STRUCTURE DE LA BASE DE DONNEES



3. LIAISONS DES PROGRAMMES TECHNOLOGIQUES A LA MAQUETTE

Incluant les nécessaires outils de sécurité, cette fonctionnalité de BDGS est obtenue à l'aide d'un outil que nous nommerons EXTRACTEUR.

Un EXTRACTEUR est une combinaison par des opérateurs Booléens (AND, NOT, OR etc...) de conditions d'extraction élémentaires (types d'objets, codes de visualisation, appartenance à un sous-ensemble, localisation géographique, attributs divers, corps d'états concernés, liaisons topologiques, etc...).



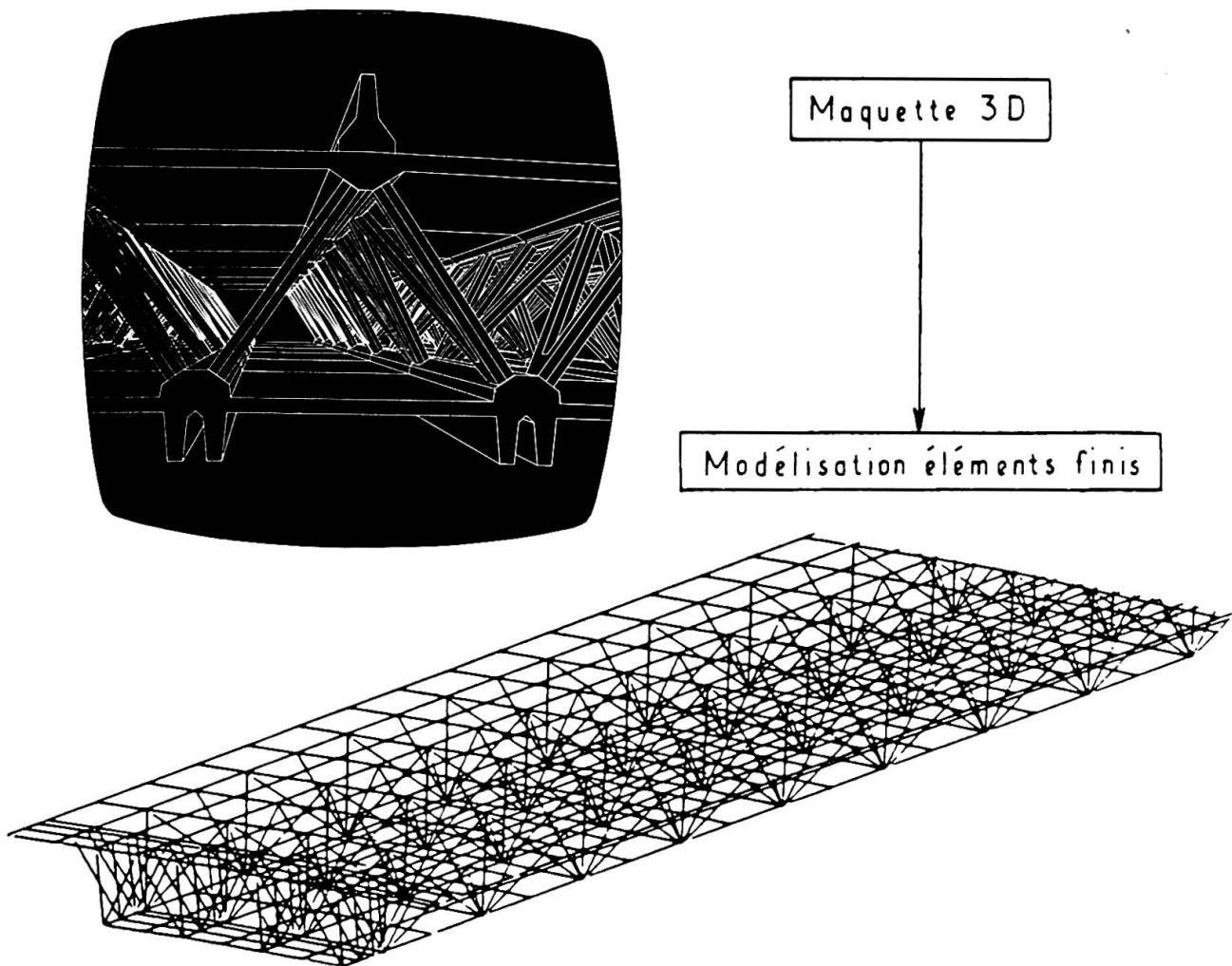
Les applications technologiques associent intimement des actions de type interactif et de type batch.

- INTERACTIF : pour compléter les renseignements préexistants dans la maquette (ajout de charges, choix des éléments participant au contreventement, visualisation de sous-ensembles à métrer, etc...)
- BATCH : une fois que les renseignements nécessaires sont connus, il est inutile de charger l'interactif des calculs complexes que peuvent nécessiter les applications technologiques.

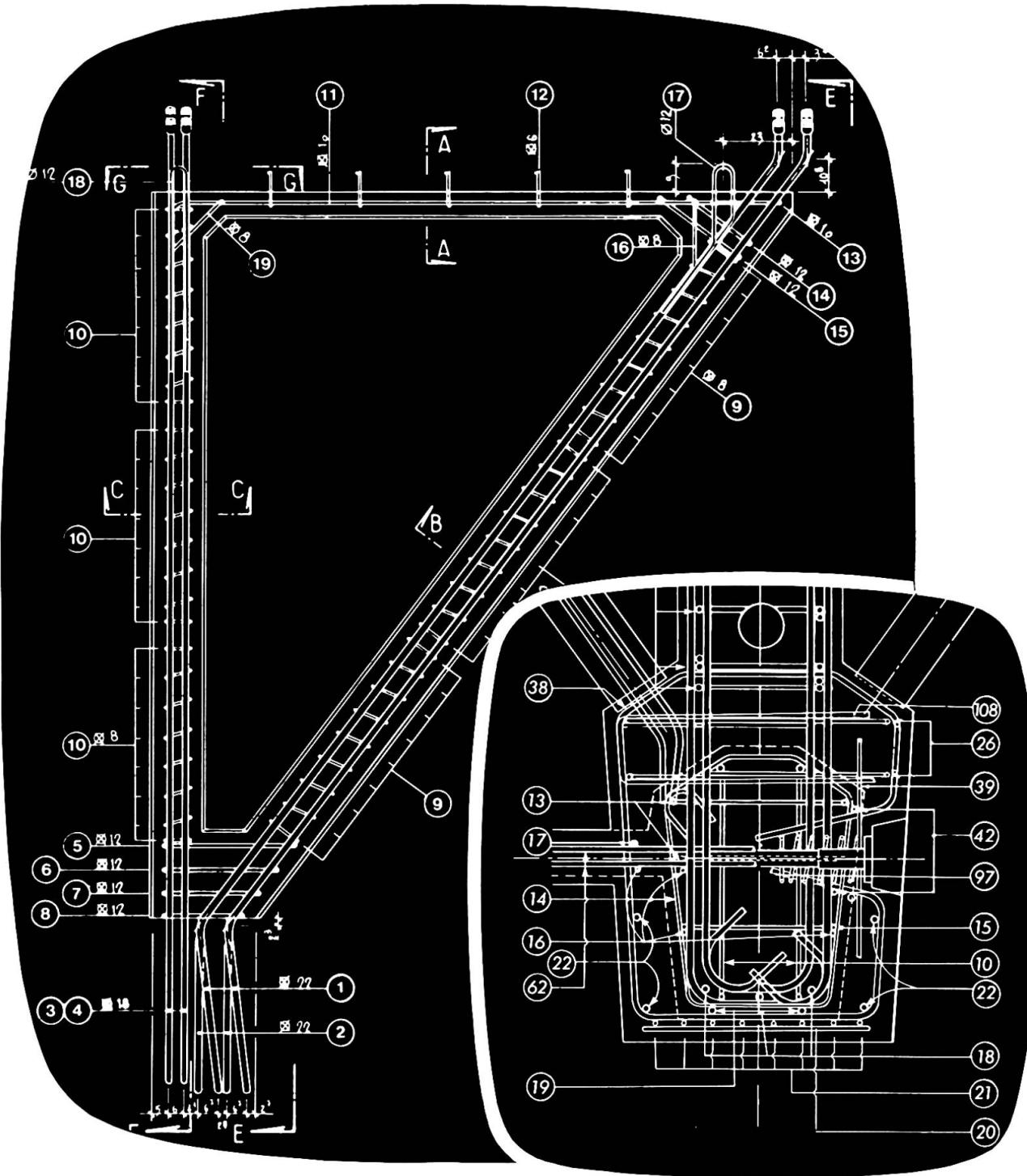
4. EXEMPLES DE PROGRAMMES D'APPLICATION

A l'aide de fonctions technologiques, l'utilisateur enrichit la base de données des informations qui lui sont nécessaires (ex. : éléments portant ou portés pour la descente de charges).

C'est ainsi qu'il peut discréteriser en éléments finis le modèle.



A partir des résultats de l'analyse, l'utilisateur produit l'ensemble des documents de ferrailage (plans, détails, nomenclatures). Il contrôle les procédures de montage des aciers (cheminée de betonnage, préfabrication de panneaux, etc...).



L'utilisateur contrôle la capacité des outils de coffrage et leur modalité de mise en place (déplacement, course de verins). Il optimise leur emploi (cycle de production).

Par exploitation de mesures réalisés lors de la préfabrication et la mise en place des éléments il contrôle la qualité de l'ouvrage.

5. EXPLOITATION DE L'OUVRAGE

Il serait souhaitable que les différents intervenants à l'acte de construire intègrent l'informatique graphique dans leurs relations et qu'ils puissent les uns et les autres fournir tout document sous forme magnétique.

Nonlinear Structural Analysis: the Glass-Box Approach

Analyse structurale nonlinéaire interactive

Interaktive nichtlineare Tragwerkanalyse

E. ANDERHEGGEN

Prof. Dr.
Swiss Inst. of Technol.
Zurich, Switzerland



Prof. Dr. E. Anderheggen, born 1939, Diplom and Ph.D. at the ETH in Civil Engineering. Since 1976 Professor for Applied Computer Sciences at the ETHZ.

SUMMARY

A new approach to incremental nonlinear static and dynamic analysis as implemented in a recently developed general purpose finite element program is presented. The approach is new not because new algorithms or element models are implemented, but because full interaction between the program and its user during the step-by-step analysis process is possible. This means total involvement of the program user with the algorithms and the element models he is using, i.e. the opposite of the "black box" approach generally adopted.

RESUME

Une nouvelle façon de traiter les problèmes d'analyse structurale nonlinéaire est présentée. La nouveauté consiste non pas dans les modèles ou les algorithmes utilisés mais dans le fait que l'utilisateur a la possibilité de suivre en temps réel le développement du calcul sur l'écran de son terminal et d'intervenir à chaque instant pour changer les paramètres et les stratégies utilisés pour la solution du problème.

ZUSAMMENFASSUNG

Eine neue Arbeitsweise bei der Lösung nichtlinearer Probleme der Tragwerksanalyse wird vorgestellt. Neu sind dabei nicht etwa die verwendeten Modelle oder Algorithmen, sondern die dem Programmbeutzer gebotenen Möglichkeiten, den Verlauf der Berechnungen auf seinen Bildschirmterminal zu verfolgen und die Lösungsparameter jederzeit interaktiv zu verändern.



1. INTRODUCTION

Many difficulties encountered when applying sophisticated numerical procedures to real-life structural design problems stem from the fact that the engineer responsible for the design and the computer scientist who developed the FE program used for the analysis can not be the same person. While in the (good) old days all needed know-how was concentrated in one person, today at least two highly specialized but unfortunately totally unrelated professionals are needed. Sometimes a third person comes in to play. This happens when the structural designer seeks the help of a "stress analyst" for modeling his structural problem and running the FE program. In fact structural designers, in many cases, do not have the time and the know-how needed for using a FE program or even for checking that somebody else has used it correctly.

This situation is particularly unfavorable in nonlinear analysis which can only make sense if the approximations involved in the numerical model are understood, if the workings of the iterative algorithms used and their sensitivity to the many parameters involved are known and if the basic design problems the analysis is supposed to solve are clearly formulated. When some of these conditions are not satisfied design and analysis tend to diverge from each other : the designer bases his decisions, just like in the old days, only on common sense, experience and simple hand calculations while the "sophisticated" numerical results obtained by the stress analyst are often accepted only if they confirm the expectations of the designer. Sometimes they are considered a waste of time just needed to make contractors and regulators happy.

What can the computer scientist do to enhance the confidence of the designer in FE analysis ? An obvious answer is that adequate teaching and "user friendliness" of the computer programs are of great importance. In addition, at least for non-linear analysis, it is certainly necessary for the program user to truly understand the algorithm and the element models he is using.

Based on these considerations a new general purpose FE-program called FLOWERS capable of linear and nonlinear static and dynamic analysis for different types of structures was developed. The most remarkable feature of this program is the approach adopted in nonlinear static and dynamic analysis: the program allows full interaction between the computer and the program user during the execution of the interactive step-by-step procedures needed in nonlinear analysis. This means that the program user can sit in front of a terminal and follow on the screen the development of the numerical computations in real-time. At any time he can stop execution, look at different partial results, change solution strategies, go back to some predefined restart point, repeat some parts of the analysis with different solution parameters, compare results, etc. It is the aim of this paper to discuss this approach as implemented in the program FLOWERS.

2. NONLINEAR ANALYSIS WITH THE PROGRAM "FLOWERS" : GENERALITIES

The program FLOWERS consists of eight independent programs, called modules, which communicate with each other through secondary storage. For nonlinear analysis the following four modules are used:

SYSIN (SYStem INput module) for inputting system, element and load data for both linear and nonlinear static and dynamic problems

NOMBA (NONlinear BAtch module) for inputting additional specification needed for nonlinear static and dynamic analysis

NONIN (NONlinear INteractive module) for the step-by-step solution of nonlinear static and dynamic problems

GRAPH for graphical output.

The FLOWERS modules are made up of two different kinds of interconnected FORTRAN routines: the module routines and the element routines. The module routines build the main body of each module. They perform the processing steps needed for any kind of finite element model independently of specific applications. The modules NONBA and GRAPH are application independent being made up exclusively of module routines. The element routines are related to specific library element models used for implementing a variety of finite element procedures for different applications. The main task is to compute the numerical coefficients describing the physical behaviour of each element (i.e. the local element matrices) to be transmitted to the module routines for further processing. Library element routines are attached to the modules SYSIN and NONIN. Therefore these modules are application dependent. In fact for different classes of applications (like plane trusses, space frames and shells, three dimensional continua, etc.) separate versions of these modules comprising different element routines identified by their library names exist.

All FLOWERS modules with the exception of NONIN are batch programs in the sense that they communicate with their users only through input and output textfiles. The syntax and, to a large extent, the semantics, i.e. the physical meaning of the free-format, problem-oriented input language used for specifying the input data (i.e. for writing the input file) are defined and described by syntax diagrams similar to those often used for defining programming languages (e.g. PASCAL). Fig. 1 shows the syntax diagrams of the module NONBA. The very simple conventions needed to understand how to use these diagrams shall not be explained here (see (1)). It should be stressed however, that we believe, and practical experience with FLOWERS and other programs confirm, that syntax-diagrams are an extremely effective way, in fact probably the only acceptable way if real user friendliness is sought, for defining and describing batch oriented input specifications.

Contrary to all other FLOWERS modules, the module NONIN is an interactive program expecting all needed input data to be typed by a user sitting in front of a terminal.

During the execution of the module NONIN the nonlinear computations occur. That is, for a given time history of the external loads, the joint displacements and all related state variables (e.g. the element stress components) are evaluated time-step-by-time-step. In static analysis (where time represents a load parameter) the modified Newton-Raphson algorithm ranging from the initial stress method to the original, non-modified Newton-Raphson method is implemented. For nonlinear (or linear) dynamic analysis the explicit central difference method as well as two implicit algorithms, the Newmark method and the so-called α - method developed within our group (see (2)) are at the present time implemented.

These algorithms are implemented at the global system level, i.e. by means of the application independent module routines. These are interfaced to the element routines which must provide all numerical coefficients describing the nonlinear



element behaviour as needed by the particular algorithm being used. Therefore all kinds of nonlinear effects (including e.g. contact problems) are treated at the local element level by the element routines. These, as explained below ("Single value output state variables", "Direct element output"), also have to provide different kinds of output data (e.g. stress components, plastic work, etc.) needed to inform the program user about the current state of the elements.

3. PREPARING FOR NONLINEAR ANALYSIS

The step-by-step solution of a nonlinear static or dynamic problem for a given time history of the external loads requires the execution of the preparatory batch modules SYSIN (SYStem INput) and NONBA (NONlinear BAtch), followed by the interactive module NONIN (NONlinear INteractive).

The module SYSIN is used both in linear and nonlinear analysis for specifying global system data (joint coordinates, element incidences, etc.), element data (e.g. material properties) and time independent load data. After the execution of the module SYSIN (and perhaps of the module GRAPH for plotting the element mesh) some additional data specifically needed for nonlinear analysis have to be inputted by executing the batch module NONBA. Its input statements are described by the syntax diagram of Fig. 1. The type of input data they refer to can be summarized as follows:

- 1) ANALYSIS TYPE (static and/or dynamic) and algorithm(s) to be used. It should be noted that static analysis can precede dynamic analysis and that explicit and implicit dynamic integration algorithms can be mixed in the time domain (however not within the element mesh).
- 2) TIME HISTORY of the external loads, i.e. the time functions associated with each of the independent loadcases previously defined.
- 3) INITIAL DYNAMIC CONDITIONS referring to nodal displacements, velocities and accelerations at the beginning of dynamic analysis.
- 4) DAMPING coefficients building the global damping matrix used for taking linear viscous damping into account. Nonlinear and nonviscous damping is to be treated at the element level.
- 5) CONVERGENCE TOLERANCE DEFINITION: with the exception of the explicit central difference method, all algorithms mentioned above have to check for convergence after each equilibrium iteration within each time-step. The vector norms used to do this have to be defined and identified by names ('ident') chosen by the program user. The actual numerical tolerance values associated with these norms are specified (resp. modified, activated or deactivated) interactively during the execution of the module NONIN.
- 6) SINGLE VALUE OUTPUT STATE VARIABLES associated either with nodal information (displacements, unbalanced loads, etc.) or with element information provided by the element routines (stress components, plastic work, etc.) have to be specified and identified by names ('ident') chosen by the program user. These variables can then be displayed on terminal, printed or saved for graphical postprocessing by the module GRAPH.

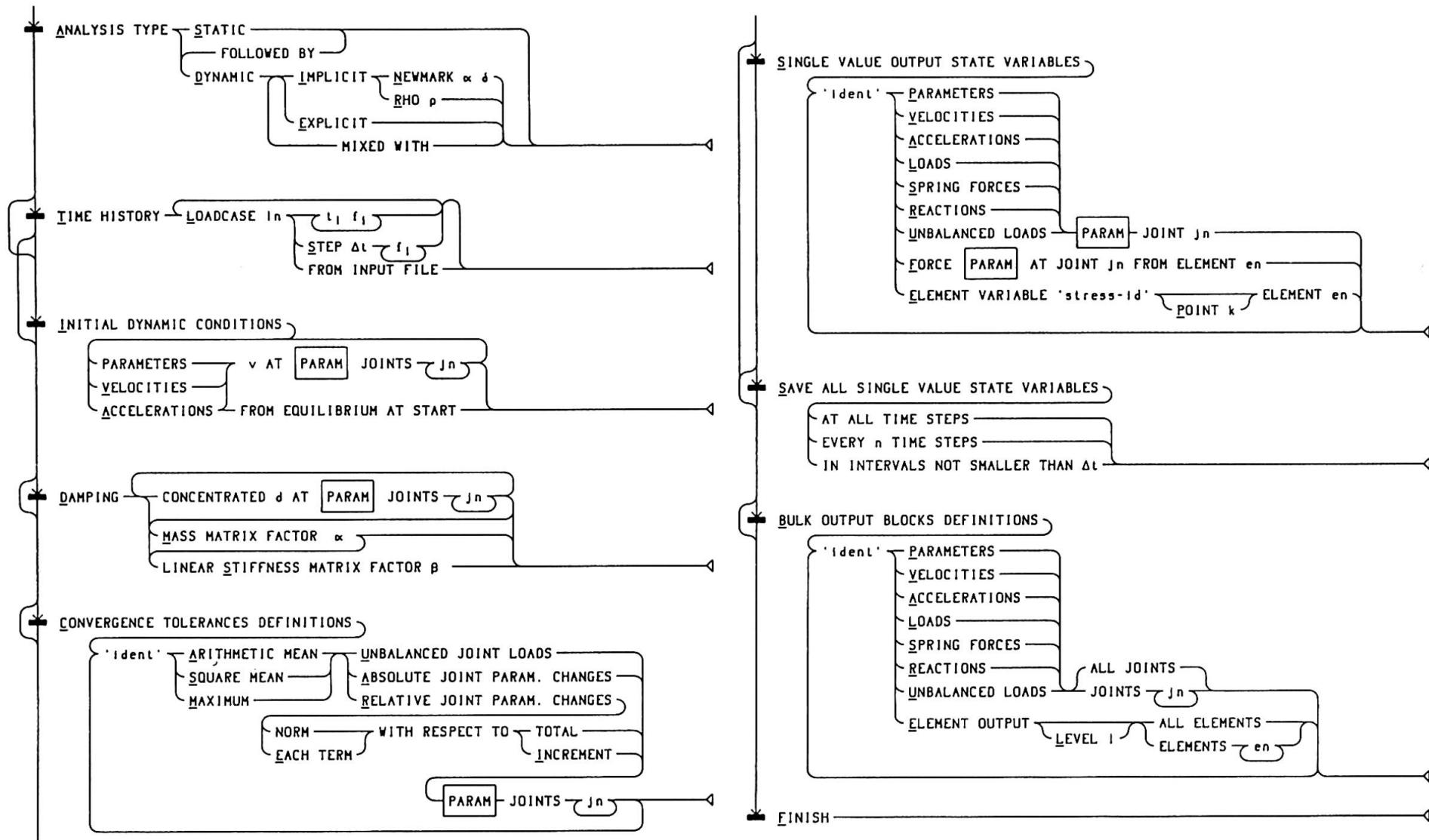


Figure 3 Syntax diagram of the module NONBA



7) BULK OUTPUT BLOCKS DEFINITIONS referring to nodal or element data to be printed when requested by the program user during the execution of the module NONIN. Again these data blocks have to be identified by names ('ident') chosen by the program user.

4. RUNNING NONLINEAR ANALYSIS

Fig. 2 shows the workings of the interactive module NONIN. Whenever requested by the program user, as well as at the beginning and shortly before the end of the interactive session, the main input menu, called the MONITOR menu, appears on the terminal screen. The program user has then the options of pressing on his

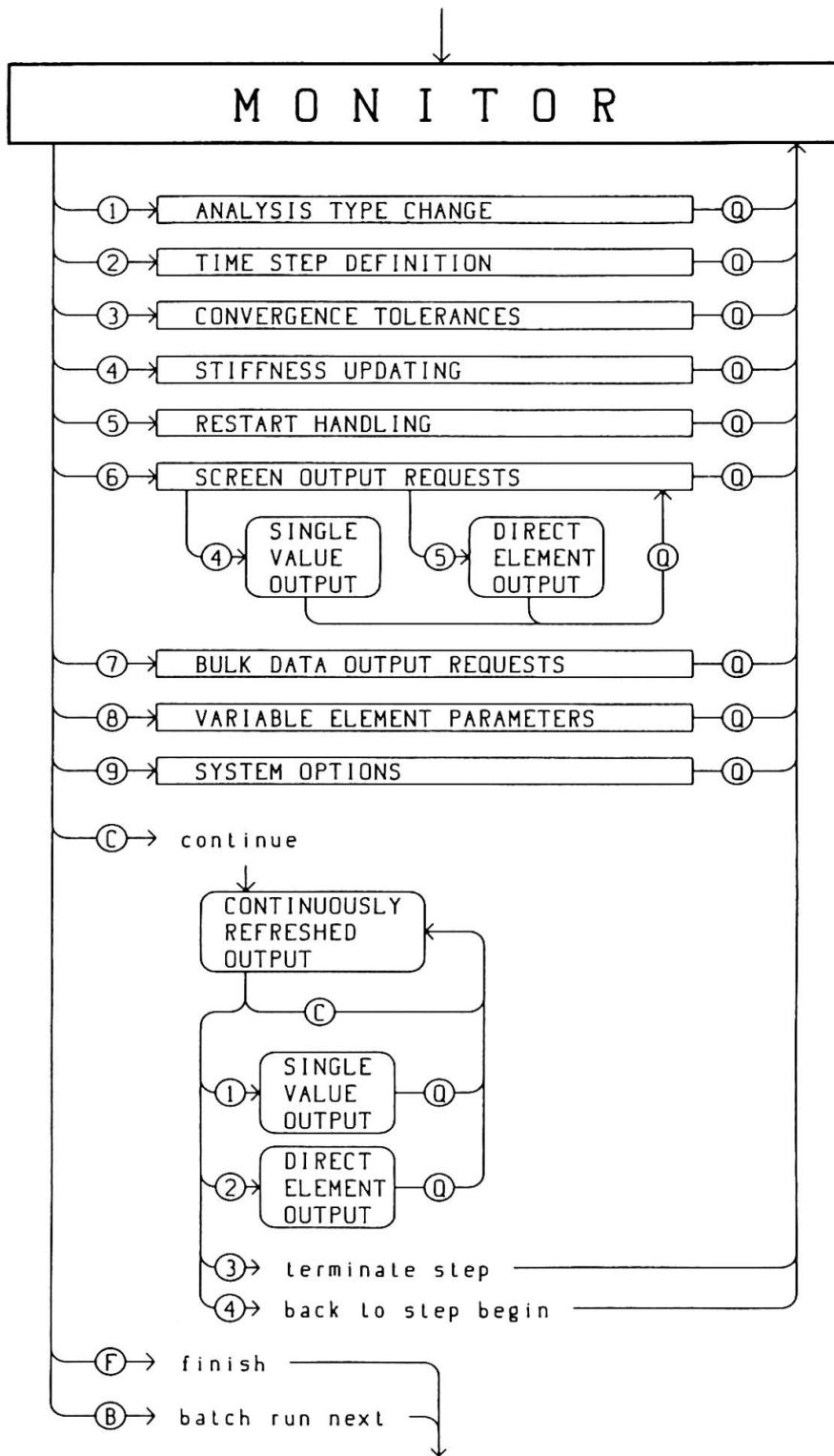


Figure 5 Interactive I/O of the module NONIN



keyboard either one of the digits "1" to "9" for calling the corresponding input sub-menu or one of the letters "C", "B" or "F" for continuing or terminating execution (these digits and letters appear on the left side of Fig. 2 where they are enclosed in circles).

The input data the user can specify by means of the sub-menus 1 to 7 can be summarized as follows (for brevity the sub-menus 8 and 9 shall not be considered here):

- 1) ANALYSIS TYPE CHANGE menu used for switching from static to dynamic analysis or for changing integration algorithms during dynamic analysis (e.g. from implicit to explicit or vice-versa).
- 2) TIME STEP DEFINITION menu used for specifying the length and the number of load history time steps until the next adjustment in time step length is needed or until the last load history step is reached. An elaborate procedure for automatically reducing the step length when convergence difficulties occur has also been introduced.
- 3) CONVERGENCE TOLERANCES menu used for specifying numerical tolerance values to be compared with the tolerance vector norms defined by the module NONBA. Convergence checks can also be selectively activated or deactivated.
- 4) STIFFNESS UPDATING menu used for specifying the strategy to be adopted for rebuilding the global stiffness matrix (e.g. every 3 time steps at the beginning of the step, or after 7, 13 and 25 unsuccessful equilibrium iterations within any step, etc.). These options are relevant both in static analysis by the modified Newton-Raphson algorithm and in dynamic analysis when implicit algorithms are used.
- 5) RESTART HANDLING menu used both for defining so-called "restart points" where all relevant problem data are saved on secondary storage, and for repeating one portion of the analysis starting from a previously defined restart point. This is an essential feature in nonlinear analysis as it allows to compare results obtained by different procedures (e.g. different time steps, different stiffness updating strategies, different dynamic integration algorithms) and also to overcome difficulties due to an unfavorable choice of solution strategies (e.g. a singular tangent stiffness matrix).
- 6) SCREEN OUTPUT REQUEST menu used for specifying which of the single value output state variables are to be displayed continuously on the terminal screen. This menu is also used for displaying the current values of all these variables (SINGLE VALUE OUTPUT) or for requesting the element routines of some elements to display their so called "DIRECT ELEMENT OUTPUT" containing appropriate information on the current state of the element.
- 7) BULK DATA OUTPUT REQUESTS menu used for specifying the data blocks to be printed or saved for graphical postprocessing and the load history times when this has to occur.

When from the MONITOR menu the program user types the letter "C" the nonlinear analysis process is continued (or started). On the terminal screen different data are then displayed and updated at the end of each equilibrium iteration within each time step. These data include the values of the active convergence tolerance vector norms expressed as a percentage of the corresponding tolerance values. The single value output state variables selected in the sub-menu no. 6



(up to 12 of them) are also continuously displayed as well as error and non-error messages sent by the element routines. All this information is displayed without interrupting execution and thus at a speed which depends greatly on the problem size and on the performance of the computer used (the program FLOWERS is being developed on a DEC-10 computer). According to different system options specified in the sub-menu no. 9, execution can be stopped at the end of each equilibrium iteration, at the end of each time step when convergence is reached, when an element sends a message, or only when all time steps defined in the sub-menu no. 2 have been analysed. However the program user also has the option to stop execution simply by pressing any key on his keyboard.

When execution is stopped different options are open. It is possible just to continue execution, to have all single value state variables displayed (SINGLE VALUE OUTPUT), to require some elements to display their "DIRECT ELEMENT OUTPUT" or to go back to the MONITOR menu. In this last case, if convergence was not reached, the program user has the option of either terminating the present step and proceeding to the next step just as if convergence was reached, or to go back to the step beginning, change solution parameters and repeat the same step.

Two different ways exist for terminating the interactive session. By typing the letter "F" the interactive session is terminated. At a later time a new interactive session beginning from any of the defined restart points is then possible. By typing the letter "B" the interactive session or rather a batch job, without any input or output from the terminal, i.e. with only printer and perhaps post-processor output. When this option is chosen the program user has to specify different data (e.g. a sufficient number of time steps) so that during the following batch job no input data will be needed. After terminating an interactive session in this way, the subsequent batch job does not require the physical presence of the program user in front of the terminal. Upon termination of this batch run a new restart point is automatically set. The next run will then have to be an interactive session again.

5. CONCLUSION

A new approach to nonlinear structural analysis called the "glass box" approach was described. Its principal goal is to get the program user involved with the solution algorithms and the element models he uses by allowing him to follow and to steer the internal workings of the program on a terminal screen. In nonlinear analysis this seems to make sense. In fact its success depends on many parameters which in most cases can only be determined by trial and error, and trial-and-error procedures are much better implemented in an interactive environment than with a batch program. Of course the program user must know exactly what he is doing, but this is certainly necessary whatever program he uses. The black-box approach can not work in nonlinear analysis. Finally it is obvious, that for real-life problems, if the response times during interaction are to remain acceptable, a large amount of computer power will be needed. This however, should become less and less of a problem in the future.

REFERENCES

- (1) E. Anderheggen, G. Bazzi, H. Elmer, T. Friedrich, H. Maag, J. Theiler: "FLOWERS User's Manual", 2nd edition, April 1983, Institut für Informatik, ETH-Zürich.
- (2) G. Bazzi, E. Anderheggen, "The Ω -Family of Algorithms for Time-Step Integration with Improved Numerical Dissipation, J. of Earthquake Engineering and Structural Dynamics, Vol. 10, 537-550, 1982.

Interactive Limit States Design of Steel Frames

Calcul interactif, à l'état limite, de cadres métalliques

Interaktive Bruchbemessung von Stahlrahmen

Carlos I. PESQUERA

Syst. Dev. Eng.
3D/EYE, Inc.
Ithaca, NY, USA



Carlos I. Pesquera, born in 1956, in San Juan, Puerto Rico, obtained his Civil Engineering degree at the University of P.R. In January of 1984, he received a Ph.D. in Structural Engineering from Cornell University. Since August of 1983 he has been working on the development of CAD systems for structural engineering applications.

William McGuire

Prof. of Struct. Eng.
Cornell University
Ithaca, NY, USA



William McGuire, born in 1920, received his BSCE degree from Bucknell University and his MCE degree from Cornell University. He is now on the faculty of Cornell University.

SUMMARY

A computer-aided approach for the interactive limit-states design of steel frames is presented. A flexible scheme permits the engineer to define the design equations interactively. Both first- and second-order elastic analyses can be employed to predict the response of the structure to the applied loads.

RESUME

Une méthode de traitement assisté sur ordinateur pour la conception et l'analyse de structures en acier est présentée, pour les états limites. La méthode permet à l'ingénieur de définir les équations de manière interactive. L'analyse des forces élastiques du premier et du deuxième ordre peut être utilisée afin de prédire le comportement d'une structure à la charge qui lui est appliquée.

ZUSAMMENFASSUNG

Eine interaktive, computerunterstützte Methode für die Bruchbemessung von Stahlrahmenkonstruktionen unter extremen Bedingungen wird aufgezeigt. Ein anpassungsfähiges Schema erlaubt es dem Ingenieur, die Entwurfskriterien für gegebene Anforderungen zu bestimmen. Elastische Analysen sowohl erster als auch zweiter Ordnung werden angewendet, um die Reaktion der Konstruktion auf die Belastungen vorauszusagen.



1. INTRODUCTION

The authors and their colleagues have developed an interactive computer aided design system for three dimensional steel frame structures [References 1-4]. The system accommodates design by either a limit states, code related approach [Ref. 5], or through code independent methods that involve the use of geometric and material nonlinear analysis. In this paper the code based method is discussed. The assumptions and limitations are listed, some of the novel features of the system are described, and an illustrative example is presented. The use of full nonlinear analysis is also illustrated in the example.

2. FEATURES, ASSUMPTIONS, LIMITATIONS

In a typical problem, the structure geometry, boundary conditions, trial member sizes, and loads are first specified in the problem definition module -- an interactive graphics "preprocessor" [Ref. 2]. The load combinations for the limit states under consideration are then defined, the type of analysis to be used is selected, and the analysis is performed. The computer then evaluates the trial design by applying the relevant code equations to each member. Graphical displays aid in interpreting the results and the behavior of the structure. Successive iterations of re-design and analysis can be performed either automatically or under the direct, interactive control of the engineer until a system with well balanced strength properties is obtained. An additional feature, which is described in References 1 & 4 but not discussed here, is an optimality criterion based procedure for satisfying displacement limits.

In the present state of the development of the system all members are assumed to be compact, lateral buckling is prevented, and all connections are rigid. Work is in progress to remove these limitations.

3. STRENGTH DESIGN PROCEDURES

Figure 1 is a flow chart of the code related procedure for meeting strength requirements. The design algorithm can be summarized as follows: 1) An initial set of member sizes (a "design vector") is selected based on experience or rough preliminary calculations. 2) All the design rules from the applicable specification are defined as design equations. 3) For a single load combination, the structure is analyzed to obtain internal member forces and joint displacements. 4) The response of the structure to this load combination is evaluated. If the response is not satisfactory a new design is generated, either automatically or interactively. In the former case the program selects the lightest members that satisfy the controlling design equations. In the other option, the user selects the members. The results of the new design are stored in the "next design vector" as described in Section 3.1 below. 5) Steps 3 and 4 are repeated for each load combination. A design is considered feasible when for all possible load combinations the structural specifications are not violated. The ultimate goal is to find for each group of members an acceptable section with the least possible weight. After the last load combination is considered, a new design iteration is begun only if changes have occurred in the design vector since the previous iteration.

The design procedure is not a fully automatic one. At all times the engineer has control over the path to be followed. He can exit the loop at any time to modify the structure or to change any of the parameters that control the design and analysis routines.

In the following subsections each of the major components of the method are described.

3.1 Design Vectors

The term "design vector" is used to describe the list of member sizes employed at any stage of the process. It is the vector that is progressively refined from the initial to the final design. In the manual, i.e., interactive option, a member is selected or changed by pointing to an item on a displayed list of conventional designations (W18x40, W36x300, etc.) and then pointing to the member on a line diagram of the frame. The properties of that section, which are in the system database, are assigned to that member by the program.

At any stage of the process, three design vectors are stored: the "previous", "current", and "next" vectors. The current vector lists the members whose properties are being used at the current stage of the program in all calculations. The new trial design suggested by the computer or interactively by the user is stored as the next vector. This multiple, progressively updated, record enables the designer to recover a design if a subsequent review of the new trial design produces less satisfactory results than the current one.

3.2 Design Equations

One aim is to make these design tools applicable to a wide range of structural design codes. This is a difficult task since at present there is no uniform format for codes. It is for this reason that the scope of the elastic analysis based procedures has been limited to codes employing the limit states design philosophy. The format of the design equations in this type of code can be given rather generally as:

$$\frac{\text{calculated load effect at factored loads, } (S_i)}{\text{estimated resistance to factored load effect, } (R_i)} \leq 1 \quad (1)$$

To achieve a degree of code independence the design procedure has been implemented in the following way: 1) The user is afforded full control over the specification of load and resistance factors. 2) In cases in which different codes specify different formulas for the calculation of resistance to a given load effect, for example, to direct compression, several commonly used formulas are included in the program to give the designer a choice. 3) To accommodate the use of interaction equations commonly found in codes, the scheme described below has been used to define design equations that combine different load effects.

It is possible to define equations of the following type:

$$g = \sum_{i=1}^7 \delta_i C_i \left(\frac{S_i}{R_i} \right)^{e_i} \leq \bar{g} \quad (2)$$

in which δ_i is 1 or 0 depending on whether the i^{th} load effect is active or inactive, C_i is a coefficient, e_i is an exponent, \bar{g} is a function of the load effects and member resistance, and \bar{g} is the limit imposed on the design equation (usually unity). In cases in which a structural member is subjected to two or more load effects that cause direct stresses in the same direction, structural specifications prescribe so-called "interaction equations" for design. This type of equation can be defined by activating the corresponding load effects in eqn. 2 [1]. However, to accommodate the definition of interaction equations that consider the stability of individual members it is necessary to modify eqn. 2 since the bending terms in such equations need to include



both an amplification factor and an equivalent moment factor.

Figure 2 shows the menu page used for the interactive definition of the design equations. The equation displayed is an interaction equation for compression and bending. The load effects that are active are indicated by a double line box. This and the design equations to be considered in the subsequent examples are taken from Ref. [5].

3.3 Load Combinations

To verify a limit state it is generally necessary to check the adequacy of the structure for more than one load combination. The load combinations to be considered in the example to follow are those given in Ref. [5].

3.4 Analysis

The object of the analysis is to calculate the internal member forces and moments due to the applied loads. The applied loads can be either the specified service loads or factored loads, depending on the limit state under consideration. Most modern specification contain provisions for the use of first- or second-order analysis to estimate the response of the structure. The difference in the application of specification clauses in these two cases is often restricted to the consideration of stability effects. For example, if the second-order effects are taken into account directly in the analysis, the Canadian Limit States Design Specification allows the use of effective length factors and uniform bending coefficients calculated on the assumption that sideway is prevented.

Conventional stiffness method procedures are employed to calculate member forces when a first-order elastic analysis is requested. To solve for the structural response for the second and subsequent load combinations, advantage is taken of the already decomposed structure stiffness matrix from the first load combination.

The column effective length factors required when the design is based on first-order elastic analysis are computed by performing a separate elastic critical load analysis of planar bents in the structure. The reference loads used in the elastic critical load analysis of the example to be presented correspond to specified gravity loads. The effective length factors are calculated from the member axial loads at the elastic critical load [1].

For second-order analysis either iterative or incremental procedures can be employed to solve the finite element equilibrium equation. The iterative procedures provide a more accurate approach to equilibrium at the expense of computational effort. Although the incremental procedures are approximate, it has been found that for steel frames of regular geometry and loading, results accurate for design can be obtained even if steps of 10 to 20% of the applied loads are specified. Unless otherwise specified, the incremental predictor-corrector method [3] will be used in all subsequent analyses which consider elastic second-order effects.

3.5 Design Evaluation

There are two basic ways to evaluate a design. One is based on the overall behavior of the structure and the other concentrates on the performance of individual members. Several tools have been implemented which, in combination with engineering judgment, enable the designer to decide when a trial design is adequate.

To aid in the evaluation of a design based on overall behavior, the tools implemented include the display of the deflected shape of the structure, force and

and moment diagrams, and plots of the total or story lateral drift.

The performance of individual members is evaluated through the design equations. The objective is to identify those members that are either over- or under-designed. The graphical technique used for this purpose identifies those members for which the left hand side of the design equations give values between two limits specified by the designer. Figure 3 shows an image displayed for this purpose. The information in the upper right of the figure shows that the maximum value for the left hand side of the equation under consideration is 1.195. In this illustration the designer is looking for those members for which the design equation is violated, i.e., those members for which the value of the left hand side of the design equation is greater than 1.0. As an aid to detecting such conditions, the members which possess values in this — or any other — specified range are shown in dashed lines on the displayed image of the frame.

3.6 Re-Design

If the design evaluation process indicates that only a few members require modification it is probably best to select new trial sections based on judgment. The selection of new sections can then be performed interactively by replacing the current section by another of the 210 standard sections in the database. When a substantial number of members requires modification, the use of this "manual" approach becomes impractical and the automatic procedure is indicated.

The automatic re-design procedure suggests to the designer a new design vector for which the design equations are satisfied. The term "suggest" is used because the designer can modify the selected sections if a suggested section is impractical or if it violates functional requirements, such as a rule that the column depth at a given level not be greater than the depth of the columns at lower levels. A search procedure has been implemented to select from the available sections the one that meets the design equations and any user specified depth limit [1]. When the designer is satisfied with the new trial list of sections, the vectors are updated.

4. EXAMPLE

The fictitious nonsymmetric three-dimensional frame shown in Fig. 4 has been selected to demonstrate the application of the strength design procedure. A plan view of the structure, its material properties, and the specified service loads are shown in Fig. 4b. The bents spanning in the E-W direction are identified by a letter, while those spanning in the N-S direction are identified by a number.

The structure is designed for three possible load combinations, gravity loads acting alone and gravity loads plus the full wind load acting in either the West-East or the South-North directions.

In this example live load reduction provisions are not explicitly considered. To compensate for this, the live load specified in Fig. 4b is less than the value one would normally specify for a structure of this type. In addition, placement of the live load in the most unfavorable locations (pattern loading) for individual members has not been considered.

Two alternative designs are obtained for the structure. The first, Design I, is based on the use of three dimensional second-order elastic analysis. The second design, Design II, is obtained by using three dimensional first-order elastic analysis and sidesway based effective length factors to compensate for the neglected second-order effects.

The design equations used in this example are those specified in the Canadian Limit States Design Specification. The load factors and the load combinations

considered are also taken from Ref. [5]. Following conventional practice, the members in the structure are grouped to satisfy certain symmetry and functional requirements. The most critical member in a group governs the design of all the members in that group. A total of 64 members groups is used. Member groups 1-34 are beams, and the remainder are columns. Details of the grouping arrangement can be found in Ref. [1]. For the initial trial design, all columns were arbitrarily defined as W14x90 and the beams as W18x50. An upper limit of 61.0 cm is specified for the beams depths, while a maximum depth of 35.6 cm is prescribed for the columns. In both cases a minimum depth of 15.2 cm is designated. The distance between floors is 3.65 m.

The final sections obtained by the two design procedures are given in Table 1. By considering the second-order effects directly in the analysis, one attains a steel frame weight about 6% less than that obtained with the conventional elastic design procedure.

A comparison of the two designs shows that the beam sections in Design I are generally heavier than their counterparts in Design II. This is due to the consideration of second-order moments in the first design. The column sections, however, are generally smaller in Design I because inclusion of the second-order moments in their design does not completely offset the reduction in columns sizes achieved by assuming effective length factors of unity. The ratio of total beam weight in Design I to that of Design II is 1.08, and the analogous ratio for columns is 0.80.

As an extension to this example and an illustration of the capabilities of this system for full nonlinear (geometric and material) analysis Designs I and II have been analyzed to collapse for each of the load combinations considered. A full report of the results can be found in Reference [1]. In summary, all the collapse load factors computed exceed the minimum required. For instance, the collapse load factors for the most critical combined load case (wind + gravity) are 1.51 and 1.59 for Designs I and II, respectively.

5. SUMMARY

This paper has presented an approach to the interactive limit states design of steel frames. Computerized design aids are provided that enable an engineer to systematically and efficiently go through the iterative process typical of structural design. At all times the engineer has the option to override the suggestions made by the computer. The application of these tools to the design of one three-dimensional structure shows a 6% saving of steel through consideration of second order effects in the analysis.

REFERENCES

1. PESQUERA, C. I., "Integrated Analysis and Design of Steel Frames with Interactive Computer Graphics," Ph.D. Thesis, Department of Structural Engineering, Cornell University, January 1984.
2. PESQUERA, C. I., MCGUIRE, W., and ABEL, J. F., "Interactive Graphical Preprocessing of Three-Dimensional Framed Structures," Computers and Structures, Vol. 17, 1983, pp. 1-12.
3. ORBISON, J.G., "Nonlinear Static Analysis of Three-Dimensional Frames," Ph.D. Thesis, Department of Structural Engineering, Cornell University, May 1982.
4. PESQUERA, C. I., and MCGUIRE, W., "Design of Steel Frames with Interactive

Computer Graphics," Instability and Plastic Collapse of Steel Structures, Granada Publishing Ltd., London, 1983.

5. Limit States Design Steel Manual, First Edition, Canadian Institute of Steel Construction, Toronto, Ontario, 1977.

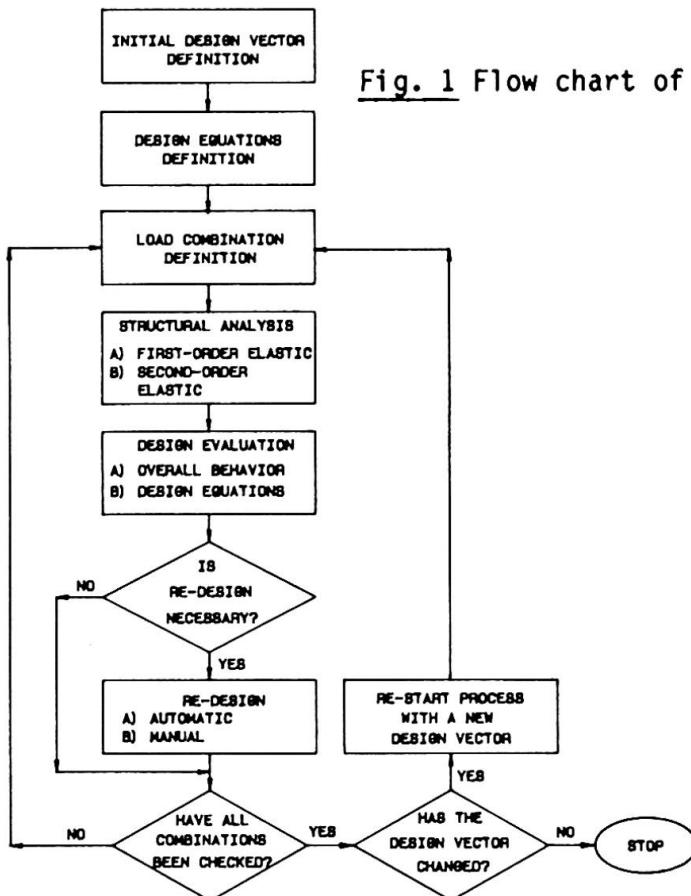


Fig. 1 Flow chart of design procedures.

10 STORY L-SHAPED FRAME						3-FEB-84													
DESIGN EQUATIONS																			
LOAD EFFECT	TENSION	COMPRES	SHEAR Y	SHEAR Z	TORSION	MOMENT Y	MOMENT Z												
COEFFICIENT	1.000	1.000	1.000	1.000	1.000	1.000	1.000												
EXponent	1.000	1.000	1.000	1.000	1.000	1.000	1.000												
PER. FACTOR	0.800	0.800	0.800	0.800	0.800	0.800	0.800												
AMPL. FACTOR							YES												
SECOND-TERM COEFF.							YES												
$C_0 < \text{PRIMAF}_y$							YES												
<table border="1"> <tr> <td>TYPE OF CHECK</td> <td>TYPE OF EQUATION</td> <td>NAME OF EQUATION</td> </tr> <tr> <td>LOAD</td> <td>USER DEF.</td> <td>BEAM</td> </tr> <tr> <td>RESISTANCE</td> <td>YIELD SUR.</td> <td>COLUMN</td> </tr> <tr> <td>RATIO</td> <td></td> <td>AXIAL</td> </tr> </table>							TYPE OF CHECK	TYPE OF EQUATION	NAME OF EQUATION	LOAD	USER DEF.	BEAM	RESISTANCE	YIELD SUR.	COLUMN	RATIO		AXIAL	POINT OF COMPUTATION MAX. ENDS
TYPE OF CHECK	TYPE OF EQUATION	NAME OF EQUATION																	
LOAD	USER DEF.	BEAM																	
RESISTANCE	YIELD SUR.	COLUMN																	
RATIO		AXIAL																	
$1.000 \frac{C_y}{C_a} + 1.000 \frac{M_y}{M_a (1 - \frac{C_y}{C_a})} + 1.000 \frac{M_z}{M_a (1 - \frac{C_y}{C_a})} \leq 1.000$																			
EQNATION	-	+	STATUS	ACTIVE	LIMIT	RETURN													

Fig. 2 Definition of design equations.

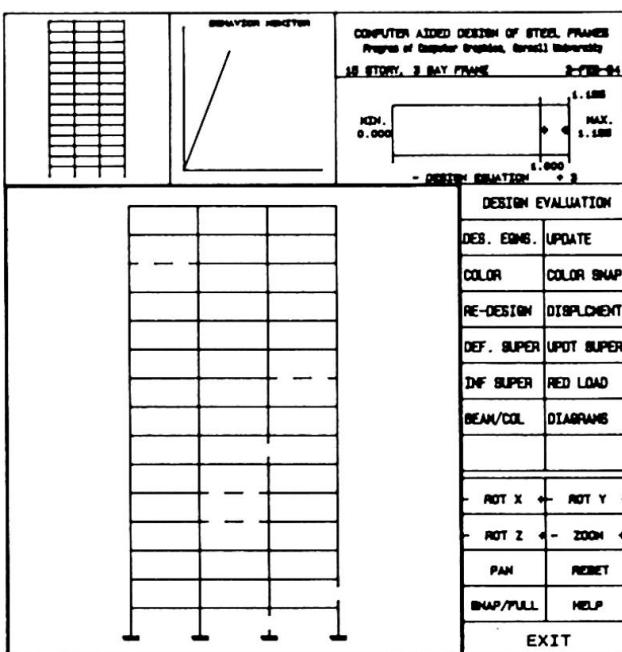


Fig. 3 Identification of overstressed members.

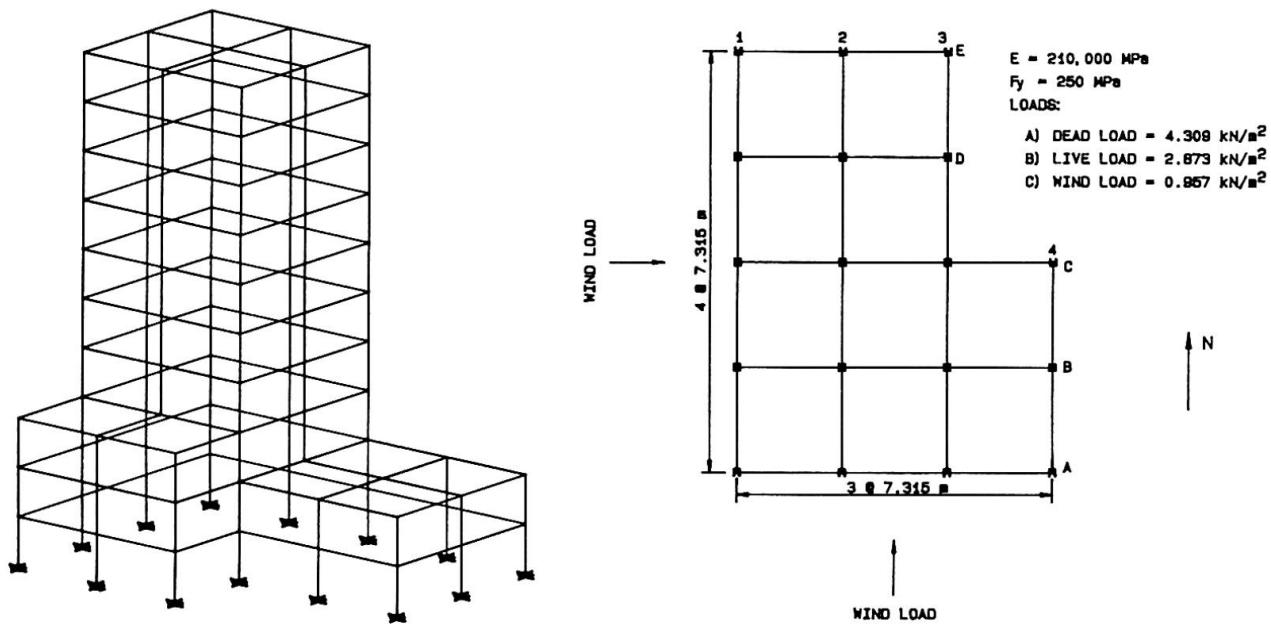


Fig. 4 a) Simplified isometric view of example structure, **b)** Dimension, material properties, and specified loads.

DESIGN I		
1 W16X 26	33 W14X 22	
2 W16X 26	34 W12X 22	
3 W16X 26	35 W12X 72	
4 W14X 22	36 W14X 109	
5 W12X 22	37 W8 X 24	
6 W18X 40	38 W12X 65	
7 W21X 44	39 W12X 87	
8 W18X 40	40 W14X 43	
9 W18X 35	41 W14X 43	
10 W16X 31	42 W12X 72	
11 W18X 35	43 W10X 39	
12 W16X 26	44 W10X 49	
13 W16X 26	45 W12X 40	
14 W14X 22	46 W8 X 35	
15 W12X 22	47 W14X 109	
16 W14X 30	48 W14X 176	
17 W12X 22	49 W10X 39	
18 W16X 26	50 W12X 79	
19 W16X 26	51 W14X 145	
20 W16X 26	52 W14X 38	
21 W14X 22	53 W12X 72	
22 W12X 22	54 W14X 109	
23 W21X 50	55 W10X 49	
24 W21X 50	56 W12X 79	
25 W21X 44	57 W10X 33	
26 W18X 40	58 W12X 45	
27 W18X 35	59 W12X 26	
28 W18X 40	60 W10X 49	
29 W18X 40	61 W12X 53	
30 W16X 26	62 W8 X 40	
31 W14X 22	63 W10X 49	
32 W12X 22	64 W8 X 48	
89900 kg		

DESIGN II		
1 W14X 22	33 W12X 22	
2 W16X 26	34 W12X 22	
3 W14X 22	35 W14X 90	
4 W12X 22	36 W14X 132	
5 W12X 19	37 W10X 33	
6 W18X 40	38 W12X 87	
7 W18X 40	39 W14X 109	
8 W18X 40	40 W12X 53	
9 W18X 35	41 W12X 53	
10 W14X 30	42 W14X 90	
11 W18X 35	43 W10X 49	
12 W16X 26	44 W12X 72	
13 W14X 22	45 W10X 49	
14 W12X 22	46 W10X 49	
15 W12X 19	47 W14X 132	
16 W14X 30	48 W14X 211	
17 W12X 19	49 W14X 48	
18 W16X 26	50 W14X 90	
19 W16X 26	51 W14X 176	
20 W12X 26	52 W14X 48	
21 W12X 22	53 W14X 90	
22 W12X 19	54 W14X 145	
23 W18X 40	55 W12X 58	
24 W21X 44	56 W14X 90	
25 W18X 40	57 W12X 40	
26 W18X 35	58 W8 X 48	
27 W16X 31	59 W10X 39	
28 W18X 35	60 W12X 65	
29 W18X 35	61 W12X 65	
30 W12X 26	62 W10X 49	
31 W12X 22	63 W12X 65	
32 W12X 19	64 W12X 65	
95345 kg		

Table 1 Final sections and total weight obtained by Design I and Design II.

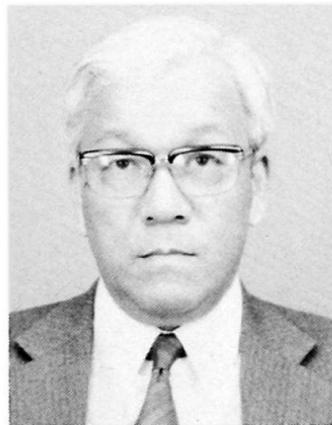
Optimum Design of Framed Structures using a Personal Computer

Projet optimal de charpentes à l'aide d'un ordinateur personnel

Optimaler Entwurf von Rahmentragwerken mit Hilfe eines "Personal-Computers"

Yasunori KONISHI

Prof. Dr.
Nagasaki Univ.
Nagasaki, Japan



Yasunori Konishi, born 1926, received his B.E. and Dr. Eng. from the University of Osaka, Suita. For 22 years he was engaged in the design of bridges for the Japan Bridge company. He is now Professor at Nagasaki University, and is engaged in research in the optimum design of Civil Engineering Structures.

Yukio MAEDA

Prof. Dr.
Osaka Univ.
Osaka, Japan



Yukio Maeda, born in 1922, received his B.E. at Hokkaido University and Dr. Eng. from the University of Tokyo. Since 1969 he has been Professor of Structural Engineering, Osaka University. At present, he is Vice Chairman of Technical Committee, IABSE.

SUMMARY

Since the number of design variables and constraints increases for large-span bridges the numerical calculation for the optimum design becomes difficult. It is possible to carry out an optimum design of a complicated framed structure, using a personal computer, by the suboptimization method. The optimum design of steel highway bridges of statically indeterminate truss form has been carried out with sufficient accuracy.

RESUME

Pour la construction de ponts de grande portée, il est difficile d'en établir le projet optimal à cause de la multiplication des variables et des conditions de limite, et de la complication des calculs numériques. L'optimisation d'un projet de structure compliquée est cependant possible en utilisant la méthode de sub-optimisation, à l'aide d'un ordinateur personnel. Le projet optimal de ponts-routes en treillis hyperstatique démontre l'efficacité et la précision suffisante de cette méthode.

ZUSAMMENFASSUNG

Beim Entwurf von Brücken mit grossen Spannweiten erhöht sich die Anzahl der Variablen und Randbedingungen. Die numerische Erfassung des optimalen Entwurfs wird schwierig. Eine Optimalisierung mit Hilfe eines Personal-Computers ermöglicht auch für komplizierte Rahmentragwerke einen bestmöglichen Entwurf. Die Entwurfsmethode wird am Beispiel von Autobahnbrücken aus Stahl, bestehend aus statisch unbestimmten Fachwerken, dargestellt und zeigt eine Anwendbarkeit für die Ausarbeitung des tatsächlichen Entwurfs.



1. INTRODUCTION

Prof. Fenves emphasized the main point of the theme in his introductory report to be that the emergence of powerful personal computers, vastly expanded computer graphics, widely accessible distributed databases, microprocessor-controlled intelligent construction equipment, and knowledge-based expert systems will all drastically change structural engineering design and construction practices, and even the nature of the structures we design, build and operate.¹⁾ Based on this point, we are submitting the present paper. In this study, we are successful in generating an optimum design of a complicated structure using a personal computer.

In general, for large-span bridges, the number of design variables and constraints increases, and the numerical calculation for their optimum design has become difficult without using a large electronic computer. It is possible to carry out an optimum design of a complicated structure, using a personal computer, by suboptimization method. In a statically determinate structure, when only governed by stress, the correct optimum values of the design variables are obtained. In a statically indeterminate structure and a statically determinate structure governed by deflection, the approximate values of the design variables are obtained. In this case, it is possible to obtain comparatively more accurate values using a fully-stressed design method, and an optimality-criterion-based algorithm, respectively.

For the structural analysis a personal computer has a limited memory capacity. Therefore, in the analysis of a statically indeterminate structure, the method of forces or the method of deformations, is used depending upon the structural type. These methods, which are used for the analyses of a wide variety of structures, are nearly as satisfactory for obtaining rigorous results as the finite-element-method. Additionally, the results of the optimum design will make it possible to carry out the automated design with an XY plotter.

A personal computer, Nihon Denki model PC-9800 with 640k byte memory, was used for this investigation. The FORTRAN Language with MS-DOS control was used for the program, where MS is a trademark of Microsoft Corporation and Dos means Disk Operating System.

2. METHOD OF OPTIMUM DESIGN USING SUBOPTIMIZATION

This method is mentioned in detail by the authors of this paper in ref. 2). A method of optimum design using suboptimization is outlined as follows: The design variables are divided into two groups, the variables \mathbf{x} of individual sections of elements and the variables \mathbf{y} of sections common to the whole structure.

In general, the constraints are given by:

$$g_{iI}(\mathbf{x}_I, \mathbf{x}_{II}, \dots, \mathbf{x}_N, \mathbf{y}) \leq 0 \quad (i=1, \dots, m_I), \\ g_{III}(\mathbf{x}_I, \mathbf{x}_{II}, \dots, \mathbf{x}_N, \mathbf{y}) \leq 0 \quad (i=1, \dots, m_{II}), \quad \dots \dots \dots \quad (1)$$

$$g_{i_N}(\mathbf{x}_1, \mathbf{x}_2, \dots, \mathbf{x}_{N-1}, \mathbf{y}) \leq 0 \quad (i_N = 1, \dots, m_N),$$

The expressions of functions of only y are given by

where, the variables x belonging to I, II,...and N are called x_I , x_{II} ..and x_N , respectively, and the constraints belonging to I, II,...and N are given by g_{iI}, g_{iII}, \dots and g_{iN} , and N analytical member elements of a structure are by I, II,...and N. The expression of an objective function is given by

Then for the objective function and constraints, the groups of the variables x_I, x_{II}, \dots and x_N belonging to the element I, II, ..., and N, respectively, are

independent of each other. In the case of a statically determinate structure, the change of its sectional dimensions does not have an effect on the distribution of the internal stresses. Then, for this case, too, as to the objective function and constraints, the groups of the variables x_1, x_{II}, \dots and x_N are independent of each other. Therefore, the constraints are given by

$$\begin{aligned} g_{iI}(\mathbf{x}_I, \mathbf{y}) &\leq 0 \quad (i_I = 1, \dots, m_I), \\ g_{III}(\mathbf{x}_{III}, \mathbf{y}) &\leq 0 \quad (i_{III} = 1, \dots, m_{III}), \end{aligned} \quad \dots \dots \dots \quad (4)$$

and the objective function z is expressed by

where

$$\begin{aligned} Z_I &= f_I(x_I, y) \\ Z_{II} &= f_{II}(x_{II}, y), \\ \dots & \dots \\ Z_N &= f_N(x_N, y). \end{aligned} \quad \dots \dots \dots \quad (6) a$$

Show the objective functions of the element I, II,...and N, respectively.

In case of a statically indeterminate structure, Eqs.(4) and (6) a can be approximately applicable. Then, the following relations are obtained the optimum values of x for any constant value of y , for

$$\begin{aligned} x_I &= h_I(y), \\ x_{II} &= h_{II}(y), \\ \dots & \dots \\ x_m &= h_m(y). \end{aligned} \quad \dots \dots \dots \quad (7)$$

Then, by substituting Eqs.(7) into Eq.(6) the following expression is obtained:

$$z = f_1\{h_1(y), y\} + f_{11}\{h_{11}(y), y\} + \dots + f_n\{h_n(y), y\} \quad \text{minimize.....(8)}$$

The constraints will be reduced to only eq.(5) and $y < 0$(9). It is then necessary to obtain the optimum solutions of the variables y by minimizing Eq.(8) under the constraints of Eqs.(5) and (9).

3. EXAMPLE OF OPTIMUM DESIGN

3.1 Example I :Two-panel two-span continuous truss

As an example I of an optimum design using a personal computer by the proposed method, the optimum design of a two-panel two-span continuous truss, as shown in Fig.1, is carried out. In this figure, ①, ②, ③ and ④ show the number of members, and (1), (2), (3) and (4) show the classification of the member sections. In this case, loads of $P=6000$ tons are applied to the panel point of the upper chord.

3.1.1 Design variables

Among design variables the grade of material S the upper flange plate and web plate thicknesses T_u , T_w , respectively, of the classification of the sections (1) and (2), the web plate thickness T_w of classification (3), and the flange and web plate thickness T_u, T_w , respectively, and the flange width B_u of the classification (4), are related to x , and the truss height H , the truss chord width BF and the truss upper chord height BW are related to y . Concerning S , the steel of 41 kg/mm^2 in tensile strength is expressed by 4, the steel of 50 kg/mm^2 is expressed by 5 and 58 kg/mm^2 by 6.

3.1.2 Constraints

In the case of the suboptimization, the constraints contain the limit of stress, the limits of values of the design variables at the limit of ratio of plate width to thickness below 80, the limit of the slenderness ratio of $l/r \leq 1/r$ where r_x and r_y are the radii of gyration about the x and y axes



respectively, and $1/r \leq 200$ at the section (1); the limit of ratio of plate width to thickness for prevention from local buckling, the limit of slenderness ratio of $1/r \leq 1/r_x$, and $1/r \leq 120$ at the section (2); the limit of limit of slenderness ratio of $1/r \leq 200$ at the section (3); the limit of ratio of plate width to thickness for prevention of local buckling and $1/r \leq 120$. In case of overall optimization, the constraints are only the upper and lower limits of the variables H , BF and BW .

The allowable stresses are a function of the discrete variable S , but it is considered to be a continuous as shown in the following equations:

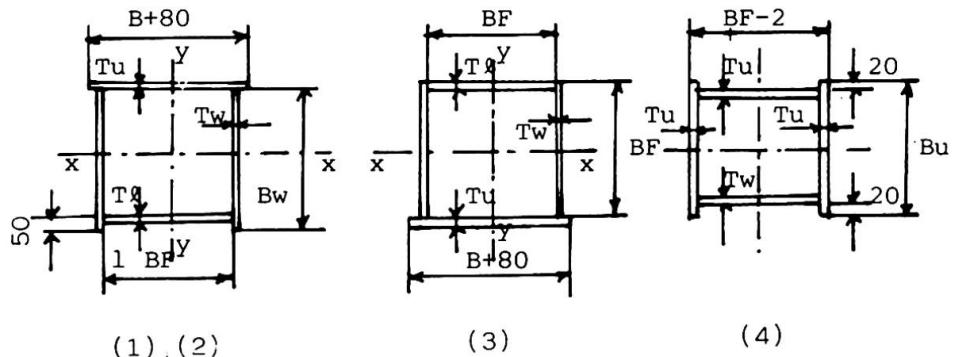


Fig.1 Notations of 2-span continuous truss (unit:mm)

$$\begin{aligned} FG1 &= 4S^2 - 41S + 120, \\ FG2 &= -13S + 145, \\ \sigma_{ua2} &= -2,2S^2 - 15.2S + 34, \\ \sigma_{ua3} &= 100S^2 - 2600S + 15500, \end{aligned} \dots \dots \dots \quad (10)$$

and

$$\sigma_{ta} = 100S^2 - 400S + 1400.$$

The compressive allowable stress without considering local buckling, σ_{cag} , is obtained from the following equations:

$$\begin{aligned} \ell/r \leq FG1: \sigma_{cag} &= \sigma_{ta}, \\ FG1 < \ell/r < FG2: \sigma_{cag} &= \sigma_{ta} - \sigma_{ua2}(\ell/r - FG1), \\ FG2 < \ell/r: \sigma_{cag} &= 12000000/\{\sigma_{ua3} + (\ell/r)^2\} \end{aligned} \dots \dots \dots \quad (11)$$

The compressive allowable stress against local buckling, σ_{cal} , is obtained from the following equations:

$$\gamma_d = 0.35S^2 - 8.75S + 69.0 \dots \dots \dots \quad (12)$$

$$C_d < \gamma_d: \sigma_{cal} = \sigma_{ta} \dots \dots \dots \quad (13)$$

$$C_d \gamma_d: \sigma_{cal} = 2200000(1/C_d)^2$$

where

$$C_d = B/T$$

B : the width of a plate,

T : the thickness of a plate.

Then, the compressive allowable stress σ_{ca} is obtained from the following equation:

$$\sigma_{ca} = \sigma_{cag} \times \sigma_{cal} / \sigma_{ta} \dots \dots \dots \quad (14)$$

The ratio, γ , of plate width to thickness for prevention of buckling of the compression members is expressed as a function of S as follows:

$$\gamma = -6S + 64.0 \dots \dots \dots \quad (15)$$

A limit of the ratio of plate width to thickness is set up for the tension members to be 80.

be calculated by the following equation:

$$\rho_s = -335s^2 + 3955s - 6580 \quad \dots \dots \dots \quad (25)$$

\tilde{H}_{k1} is calculated by the following expression:

$$\tilde{H}_1 = 7.0 \times Al_2 + 56 + 35/NP, \quad \dots \dots \dots \quad (26)$$

where

NP: the number of panels.

and

$$A_{\ell_2} \leq 0, \quad \dots \dots \dots \quad (27)$$

where

l : the length of a member (m).

Then, the optimum values of the variables x are obtained with a fixed value of y for each element, by solving the objective function (16) under the above-mentioned constraints with the SLP method.

3.1.4 Overall optimization

Among the design variables, BF , BW and H are related to y . Since Eq.(7) which is obtained by suboptimization, satisfies all of the constraints except Eq.(5), it would be enough if only the constraints could satisfy Eq.(5), in overall optimization. The above-mentioned SUMT will be applied to the method of the overall optimization.

As S is a discrete variable, the integral value is obtained by the branch and prune method.

3.1.5 Results of calculation and discussions

The optimum values of the variables, penalty function and objective function are shown in Table 1, from the results of optimum design by the proposed method.

K	IS	Bf (cm)	Bw (cm)	H (cm)	F (1000yen)	Z (1000yen)	Rk
1	0	90.0	85.0	600.0	10457.0	7228.2	50.0
1	1	90.9	88.8	620.6	9816.5	7116.7	50.0
1	2	95.8	86.8	703.5	9404.0	6942.3	50.0
2	0	95.8	86.8	703.5	9597.3	6942.3	1.0
2	1	95.2	89.2	725.7	6961.4	6905.8	1.0
2	2	93.9	88.2	821.3	6899.4	6842.8	1.0
2	3	95.2	88.2	827.6	6896.4	6838.9	1.0
3	0	95.2	88.2	827.6	6901.7	6838.9	0.02
3	1	95.2	88.4	828.5	6839.9	6838.7	0.02
3	2	95.3	88.4	835.7	6839.6	6838.3	0.02
3	3	95.2	88.4	836.2	6839.6	6838.3	0.02

Table 1 State of convergence for 2-panel 2-span continuous truss

The results of a two-panel two-span continuous truss calculated by this method are summarized in Table 2. In this table, H_0 shows the initial value of H_0 and the values of the design variables and the penalty and objective functions are shown depending on the values of H_0 . Case 1, Case 2 and Case 3 in table 2 show the different optimum values for the initial values of $H_0 = 500(\text{cm})$, $H_0 = 600(\text{cm})$ and $H_0 = 700(\text{cm})$, respectively. According to the results of the optimum design, the values of Case 1, Case 2 and Case 3 are almost equal.

global minimum value. In this optimum design, SMH=4000 (yen/hour), CM=80000

(yen/ton) and $\mu=0.05$ are employed.

Case	H_0 (cm)	B_f (cm)	B_w (cm)	H (cm)	S_i	S_2	S_3	S_4	T_{u1} (cm)	T_{u2} (cm)
1	500.0	95.2	88.4	832.3	4.0	6.0	5.0	6.0	1.19	3.17
2	600.0	95.2	88.4	836.2	4.0	6.0	5.0	6.0	1.19	3.17
3	700.0	95.3	88.3	836.6	4.0	6.0	5.0	6.0	1.19	3.10
Case	T_{u4} (cm)	T_{w1} (cm)	T_{w2} (cm)	T_{w3} (cm)	T_{w4} (cm)	B_{u4} (cm)	F (1000yen)	Z (1000yen)		
1	3.86	1.04	2.76	1.66	2.95	124.1	6839.7	6838.4		
2	3.86	1.04	2.81	1.25	2.95	124.0	6839.6	6838.3		
3	3.88	1.04	2.69	1.53	2.95	124.7	6839.4	6838.3		

Table 2 Comparison of optimum values for 2-panel 2-span continuous truss

3.2 Example II: Ten-panel 2-span continuous truss

Secondly, as another example of an optimum design using the proposed method, the optimum design of a steel highway bridge of ten-panel 2-span continuous truss type with a live load of TL20, as shown in Fig. 2, is carried out. For

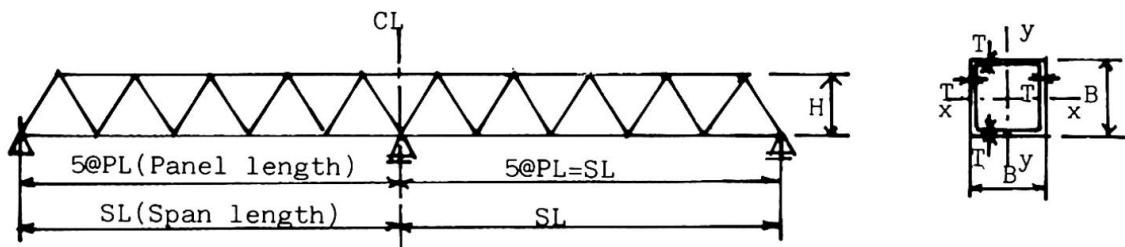


Fig.2 10-panel 2-Span continuous truss and notation

Member	Upper chord					End post	Lower						
	Case	L (cm)	H (cm)	B (cm)	$S_1 - S_{20}$	T_1 (cm)	T_2 (cm)	T_3 (cm)	T_4 (cm)	T_5 (cm)	T_6 (cm)	T_7 (cm)	T_8 (cm)
1	2@25=50	328.4	17.2	4.0	4.0	3.22	4.62	3.02	0.90	2.64	1.97	0.90	1.62
2	2@30=60	384.1	23.8	4.0	4.0	1.97	2.41	1.58	0.91	2.62	1.29	0.90	1.23
3	2@35=70	424.4	28.1	4.0	4.0	1.99	2.66	1.91	0.90	2.31	1.24	0.90	1.20
chord		Diagonal											Z
Case	T_9 (cm)	T_{10} (cm)	T_{11} (cm)	T_{12} (cm)	T_{13} (cm)	T_{14} (cm)	T_{15} (cm)	T_{16} (cm)	T_{17} (cm)	T_{18} (cm)	T_{19} (cm)	T_{20} (cm)	(¥1000)
1	1.57	0.90	3.91	0.90	0.90	0.90	1.17	0.90	0.90	0.90	1.03	1.93	33518.4
2	1.09	0.90	3.32	0.90	0.90	0.90	1.02	0.90	0.90	0.90	0.90	1.43	36202.8
3	1.16	0.90	2.32	0.90	0.90	0.90	0.99	0.90	0.90	0.90	0.90	1.33	38989.2

Table 3 Optimum values

examples of the optimum design of the bridge, 2@25(m), 2@30(m) and 2@35(m) in its effective span length and 10(m) in its effective roadway width are treated.



The variables, constraints and objective function of example II are almost the same as those of example I, S and T among the design variables are related to x , and B and H are related to y . The constraints contain the limit of stress, the limit of deflection, the upper and lower limits of values of the design variables, the limit of ratio of plate width to thickness for prevention from local buckling and $1/r < 120$. The objective function is the same as the two-panel two-span continuous truss. Also, $SMH=4000$ (yen/hour), $CM=80.000$ (yen/ton) and $\mu = 0.05$ are applied to this example. The results of the optimum design are given in Table 3. Case 1, Case 2 and Case 3 show the case of the span length of 2@25(m), 2@30(m) and 2@35(m), respectively, and the effective width of 10(m).

4. CONCLUSION

As the present study demonstrates for a statically indeterminate structure and a statically determinate structure with restriction of displacement, it is a little insufficient from the accuracy and community to any structural type, but it is sufficient as a useful method for an actual design of bridges. In addition, the result of the optimum design will make it possible to carry out an automated design with an XY plotter.

REFERENCES

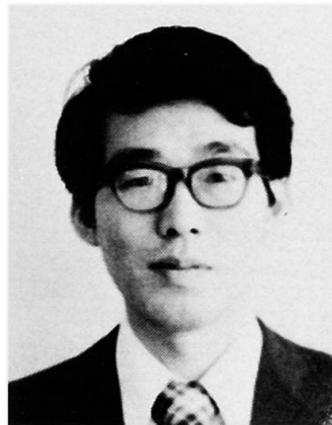
- 1) Steven J. Fenves: Computer Aided Structural Engineering, 12th congress of IABSE Introductory Report, 1983, pp 77.
- 2) Y. Konishi and Y. Maeda: Optimum Design of Trusses using Suboptimization, Proc. of JSCE, No.333, May 1983, PP 173-181.
- 3) The Association of Japan Bridge Construction: Prime Cost Data of Steel Highway Bridges in Japan in 1975 (in Japanese).
- 4) The Association of Japan Bridge Construction: Prime Cost Data of Steel Highway Bridges in Japan in 1972, (in Japanese).

CAD/CAM for the Fabrication of Steel Structures

CAO/CAM pour la fabrication de charpentes métalliques

CAD/CAM für die Herstellung von Stahlbauteilen

Seiji HOJO
 Structural Engineer
 Yokogawa Bridge Works, Ltd.
 Tokyo, Japan



SUMMARY

This paper presents an outline of the whole system with the basic concept. This system will be in promoting the systematization in the industry of multi kind and small quantity production type. The main function of this system are automated drafting and manufacturing.

RESUME

Cet article trace les grandes lignes du système entier, en montrant le concept de base. Ce système permet la production systématique et industrielle de nombreux produits en petites quantités. Les principales fonctions de ce système sont le dessin et la fabrication automatisés.

ZUSAMMENFASSUNG

Der Beitrag gibt einen Überblick über das ganze System mit dem zu Grunde liegenden Konzept. Das System wird die Systematisierung in Industrien mit kleinen Stückzahlen und Modellvielfalt unterstützen. Die Hauptfunktionen des Systems sind das automatische Zeichnen und die automatische Fabrikation.



1. INTRODUCTION

Steel structures are to be produced on orders. With change of the times, their production has been more diversified and complicated in process and larger in scale. The structural steel fabricating industry is now strongly required to improve the productivity while meeting such social conditions.

Our company has been making efforts to develop an automated production system using digital computer and NC machine for bridges and steel structures. Such system is already in actual operation in the Bridge Division. In the Steel Structure Division, however, development of such system was found far more difficult than expected. We have not yet formed a clear view also about the effect and availability of such system at the stage of application.

Meanwhile, basic conditions for the development of such system have been gradually established by introduction of hardwares such as large capacity computer and NC machines, development of basic softwares such as languages and data bases for graphic data processing and introduction of on-line system. At the same time, investigations were also made on the shop drawings for automatic drawing. As the result, it was decided to adopt the form capable of corresponding also to the flow of manufacturing particularly through simplification of expression method and classification of drawing system by functions.

Then, with these circumstances for a back ground, we intended to establish the more advanced production system by promoting the systematization from a new point of view.

This report presents an outline of the whole system with the basic concept of such system brought into focus.

2. OUTLINE OF SYSTEM

2.1 General

As for the automation of steel structure production, partial systematization in which drawing and manufacturing processes will be individually handled can hardly be expected to produce any effect. For this reason, it is necessary to realize a total system covering all kinds of information concerning the steel structure production such as preparation of shop drawings, documents on materials and various data for fabrication, full scale drawings, and operation tapes and documents for NC machines. The key concept of this total system is information control. Therefore, all items of information in the system must be under concentrated and reasonable control without overlapping and be always ready for change, alteration, modification and addition. The present system developed on the basis of this concept makes it possible to realize labor-saving, reduction of construction term and improvement of product quality by securing clear and smooth transmission of information including the unified application of information.

In addition to the above-mentioned points, systematization should be made with due regard to processing of graphic information, flexible application and control of vast volume of data and labor-saving of input data. To solve these problems, we have positively proceeded with the development of languages for graphic data processing, introduction of data bases into system and consolidation of on-line system, aiming at the higher efficiency of development.

With the above-mentioned facts kept in mind, systems configuration together with the organization for systems application and information control will be described hereinbelow.

3. DETAILS OF SYSTEM

3.1 Systems configuration

The whole system can be largely divided into external sub-systems and internal ones. The external sub-systems mean the information control and factory manufacturing, the organizations for transmitting any information to internal and external organizations through the system. The information control serves to perform activities such as prearrangements with external organization (client), preparation of input data, preparation of changed data, checking of output results, and transmission of information to each work unit in the factory manufacturing. This is therefore regarded as a kind of information center and is involved also in integration of conventional operations. The factory manufacturing means the production line which receives any production information and performs fabrication and manufacturing based on such information. In this line are now installed NC machines as particularly direct labor-saving machines including the layout machine, cutting machine and drilling machine, this making it easier to shift to the automated production line in future. In contrast to this, the internal sub-systems consist of hardwares and softwares mainly for information processing by computer. The configuration of internal sub-systems is shown in more detail in Fig. 1.

In the following, mention will be made of the relation between structural elements of information processing system such as input data, data base registration programs, file organization and processing programs with the data bases taken as main subject to explain our way of thinking about the steel structures.

3.2 Input data

(1) Input data for drawing

These data consist of a kind of general common items (general data), those for definition of framed construction of steel structures (structural data), those for giving the characteristics parameters of each steel member (member data), and those for standardization of bolt joints, sectional shape, etc. (standard data), all of which are to be handled at a drawing level.

(2) Inputs for materials

These inputs are classified by the contents and accuracy of data into those for estimation, those for arrangement for materials and those for material list to be used at the time of manufacturing.

(3) Inputs for fabrication

These inputs contain the data to be added to the information at the level of drawing in consideration of the conditions at the time of fabrication. These data consist of cutting allowances at the time of cutting, drilling data on the relation between bolt diameter and bolt hole at the time of drilling, welding data on shrinkage at the time of welding, and assembling allowance.

(4) Conversational inputs

These inputs are to make alterations and/or modifications to the registered data. On the assumption that an on-line terminal device or graphic display may be used to input these data, the information controller can intervene freely in the data. These inputs contain all of the above-mentioned items. As for the graphic information, these inputs serve as a means for expressing the graphic data on the graphic display and registering them after modification.

3.3 Data base registration program

(1) General file

This file can freely register simple numerical values or data necessary for



steel member manufacturing operations after the predetermined variable code name has been allotted to each of them according to the contents of general data.

(2) Structure (skelton) file registration

The lines to be taken as a basic line for framing in the steel structure and the lines on which members are actually arranged (member lines) are registered as geometrical data on the three-dimensional coordinates. Basically, they are expressed by the use of the concept of "plane, line and point" and correlation between plane, line and point. The registration program serves to perform automatic generation of data through the utilization of part program for each kind of structure made up in consideration of the characteristic features of building construction and register such data in the data base.

(3) Member file registration

This is to register the characteristics of member picked up one by one from the drawing. In this sense, the data tend to be larger in volume, thus leading to the very low efficiency of operation. For this reason, functions for automatic generation of members (manufacturing and treatment of members) and members are systematized and classified by the relation between members. And, using the conversational programs prepared according to such classification, it is intended to improve the efficiency of operation.

(4) Standard file registration

The standard data such as hole patterns of bolt joints, sectional shapes of members to be used, graphics to be used in great number at the time of fabrication and on the drawings, and mutually fixed condition of members are registered.

3.4

(1) General file

This file handles comparatively simple data such as a kind of general common items in drawing and production, previous steps to be taken in the applicable job, and control data in batch processing. The data contain:

- Title of job, job No., name of owner, place of construction, date
- Basic beam arrangement in each floor, basic data on bolt hole, basic sectional size for each kind of member
- Relation between the whole structure and orientation, direction to be followed at the time of erection
- Table of design mark and section code relating to the section of member, table of design mark and joint mark

By rational application of data in the general file, the system can show its effect of significantly reducing the inputting operation concerning the structure and members.

(2) Standard file

This file handles the bolt joint standards, member section standards, members installation standards and shape standards. Basically, the data are divided into those to be applied to all jobs and those to be applied only to the specific job. This file consists of:

- Splice file: Bolt hole patterns of bolt joints are stored as simple data for pitch and gauge with a hole pattern mark allotted to each of them. It is also possible to make a master hole pattern by combination of these basis patterns. This file is so designed that it may handle even more complicated patterns.
- Steel section file and variable section file: The steel section file handles basic sectional shapes. The variable section file has combinations and changed positions of steel section file on the member axis. This means that, no matter how significant sectional change any one member may show, its section can be expressed by marks on the variable section file.
- Fitting file: This file has an information on mutual fitting of members,

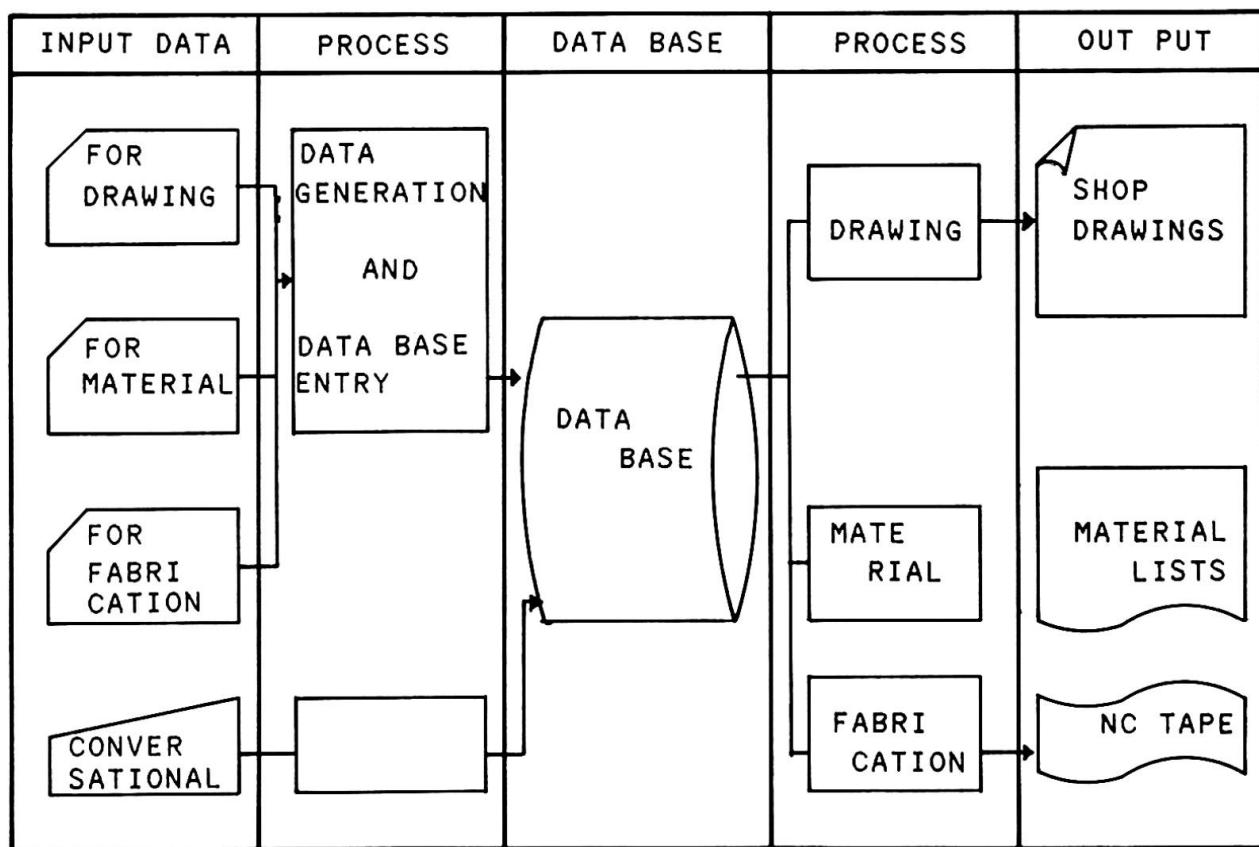


FIG .1 INTEGRATE-SYSTEM

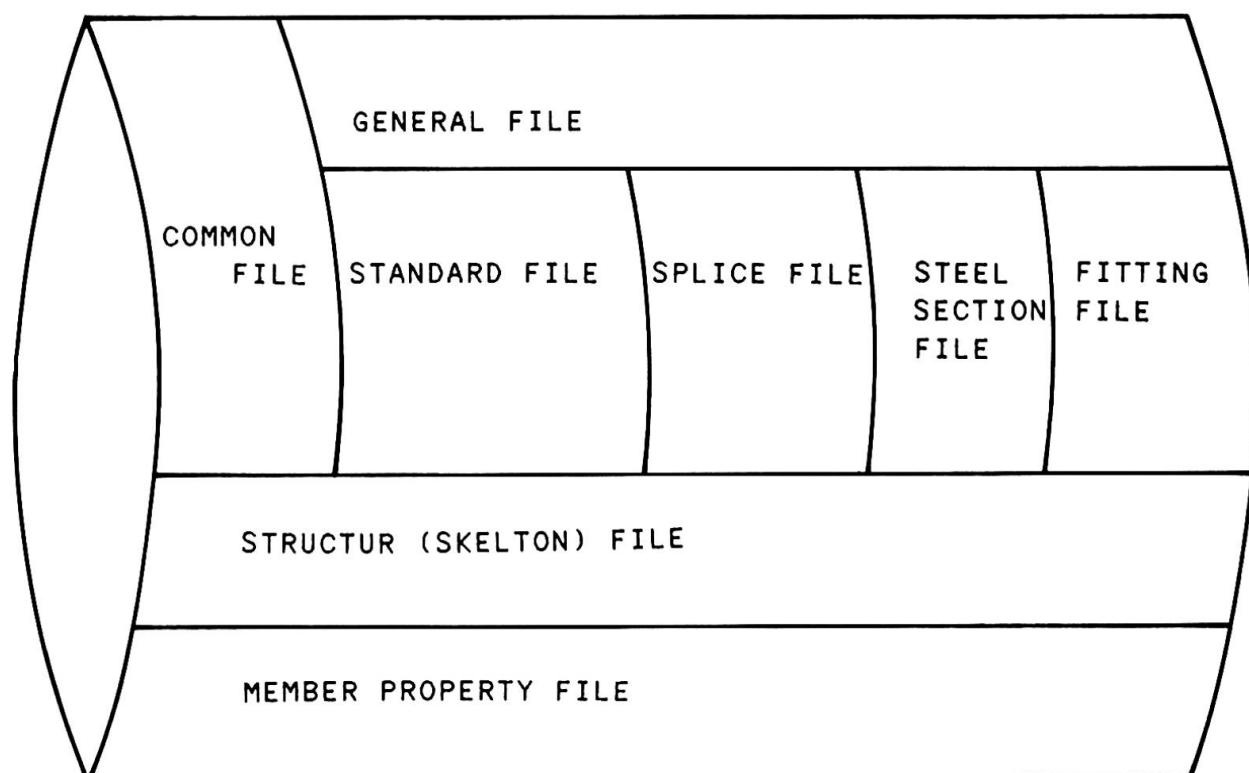


FIG .2 DATA BASE

consisting of fitting marks and fitting positions and fitting angles represented by three-dimensional coordinate values.

(3) Member (property) file

The member file serves to express the characteristics of members constituting the steel structure and handle the relation between the standard file and structure file. This file has a function as a master file in automatic drawing processing and is featured in that it contains all necessary data for automatic drawing as parameters.

Now, mention is made of systematic classification of steel structural members made by our company according to their respective characteristic features.

Classification of members by composition:

- Block members ... Unit member as product or for dispatch
- Sub-block members ... Unit member as intermediate product. These members are line members to be handled as calculation elements in structural design and are corresponding in this unit to the structure file.
- Single piece members ... The minimum unit member for fabrication

This classification corresponds also to the file organization in the member file and the system of shop drawings.

Classification of single piece members by purposes of use:

- Main piece members ... Of single piece members constituting the sub-block member, those pieces constituting its main line member
- Joint piece members ... Pieces constituting the joint in the block or sub-block members (for example, gussets, splices, fillers, etc.)
- Stiffening piece members ... Pieces for stiffening in structural design (for example, stiffeners, diaphragms, etc.)
- Equipment piece members ... Pieces for building equipment (for example, sleeves, fasteners, etc.)
- Erection piece members ... Pieces necessary for site erection (for example, hangers, ladders, etc.)
- Others

This classification of single piece members relates to member preparation processing. Therefore, if the parameters such as fitting positions, shapes of members are set up for each of such classifications, the efficiency of processing may be improved. These files form a hierarchical structure in the order of block, sub-block and piece. Each file can thus call its constituent members automatically. It is therefore possible to express comparatively simply the mutual relationship between members, one of problematic points in the steel structures. The member file has all items of information at the drawing level and serves to prepare the master file by using parameters for member preparation for the purpose of member processing.

(4) Structure (skelton) file

The structure file serves to set up the framework of the whole steel structure on the three-dimensional coordinate system and handle the relation between the members constituting the framework and basic lines, mutual relationship between members, and coordinate values on the three-dimensional coordinate system. In this file, all members to be handled are such members as may be expressed as line members and these are represented by lines. Those members expressed as line members correspond to the sub-block members in the member file. The relation between the members and general structure is represented by planes and lines while mutual relationship between members is represented by lines and points. Accordingly, in the structure file, all data are expressed by the concept of "plane, line and point" which is, in turn, expressed like a network.

The structure file in the data base is composed of:

- Plane file ... Corresponds to the plan (framing plane) and elevation (framing

elevation) in the general drawing.

- Line group file ... Represents the group of reference lines such as lines of alignment and column reference lines.
- Line file ... Represents all items to be expressed lines such as reference lines and member lines.
- Point file ... Represents the intersections of lines, joint points of members, etc. The coordinate values are stored herein.

The structure file handles the relevant data in the basic framework only and has an information of such degree as may show the mutual relation between members.

The more detailed values are to be given as correction values in the member file.

(5) Master file

While the design and production have conventionally been connected with each other through the graphic information, the master file plays the role of intermediate medium in this system. The master file is divided according to the contents of data itself into two parts, though these two parts are all the same in internal specification. One is for members of the same size and shape as those expressed in the shop drawing and is called the "Pre-master File." The other is the file same as this pre-master file except that fabrication data are added to it and is called the "Master File."

The master file is composed of:

- Head file ... Title of job, job No., date of preparation, etc.
- Inherent file ... Shape of single piece, hole information, fitting information, etc.
- Block file ... The relation of block with single pieces constituting it.

The master file consists mainly of inherent file in which the shape of single piece is expressed by structural lines. This file has all items of information necessary for fabrication of single pieces such as material, plate thickness, hole information and marking information. This file is therefore playing a leading role in the subsequent production processing and serves as a source of information for production.

3.5 Processing program

(1) Drawing processing program

The drawing processing program serves to perform automatic preparation of shop drawings using the data other than those stored in the master file. Somewhat unlike the conventional drawing system, the shop drawing is divided systematically into parts so as to make the information to be expressed on the drawing corresponding to the manufacturing process and to avoid the duplication of information.

- General drawing ... Framing plane, marking drawing, framing elevation
- Assembly drawing, general assembly drawing, partial assembly drawing
- Single piece drawing
- Standard drawing ... Bolt joint standard drawing, welding standard drawing

In this system, the expression method of shop drawing was simplified and the drawing scheme was systematized. Simultaneously with this, files in the data base were made structural, thus finding the possibility of automatic preparation of shop drawings. This idea is an important factor in application of shop drawings and for taking measures against the changing operations as may sometimes occur in production of steel structures.

(2) Production processing and production information

The data registered in the files relating to drawing processing are automatically subjected to member processing and then registered in the master file. Production processing is to output the contents of this master file as production



information.

The production information includes:

- Material list, material arrangement program
- Template
- Butting drawing, template drawing for drilling
- NC tape (drilling, cutting, marking)

Production processing includes:

- Processing for addition of fabrication data
- Material processing (sorting, nesting)
- Butting processing
- NC processing

Since the production information is outputted from the same data as have been checked at the stage of shop drawing, various stages of conventional manual operations can be unified and integrated. The effect of this system as a total system can be realized just by this production processing.

4. POSTFACE

The system as has been described above can realize the following items through the utilization of digital computer for preparation, control and application of all kinds of information in production of steel structures including the shop drawings and by introduction of automatic labor-saving machines such as NC machine tools.

- Unification or integration of production processes at various stages
- Mechanization of fabrication
- Continuity of production
- Higher standardization of operations

This system will be a pioneer in promoting the systematization in the industry of multikind and small quantity production type.

Conception et construction assistées par ordinateur

CAD-System für das Bauwesen

Computer Aided Design and Construction

Stéphane BERNAERT

Directeur
CACT
Paris, France



Stéphane Bernaert, né en 1930, Ingénieur ETP, obtint le degré de master of science de l'Université de l'Illinois en 1956. Après plusieurs années dans l'Illinois et le Texas, il revint à Paris où il dirige depuis 1971 le CACT, société d'informatique spécialisée dans le bâtiment et les travaux publics.

RESUME

L'article présente un système modulaire et évolutif de CAO, fonctionnant sur micro- et mini-ordinateurs et conçu spécifiquement pour les divers intervenants dans l'acte de construire, en vue d'aider à la décision et de faciliter la communication.

ZUSAMMENFASSUNG

Das in diesem Beitrag vorgestellte modulare und entwicklungsfähige CAD-System für Mikro- und Mini-computer wurde speziell für die im Bauwesen tätigen Partner entwickelt. Dieses System hat als Aufgabe bei Entscheidungen und Kommunikation Hilfe zu leisten.

SUMMARY

This paper presents a modular and evolutive CAD system, running on micro- and mini-computers and designed specifically for the various professions in the construction field as an aid to decision and communication.



1 – UN OUTIL SPÉCIFIQUE DE CONCEPTION-CONSTRUCTION

Au cours de ces dernières années, le processus de conception dans des secteurs industriels aussi variés que l'électronique, la mécanique, l'automobile ou l'aéronautique a considérablement évolué grâce à l'apport de la Conception Assistée par Ordinateur (C.A.O.).

L'industrie de la construction (Bâtiment et Travaux Publics) est restée un peu en retrait de cette évolution, pour deux raisons principales :

- Les études de conception ne peuvent pas normalement s'amortir sur d'importantes séries de produits. En construction, la nécessité fondamentale de l'adaptation au site, plus généralement la mise en jeu pour chaque nouveau projet d'un ensemble particulier de contraintes, notamment fonctionnelles, esthétiques, sociologiques, techniques, économiques, font que construire, c'est toujours faire du « sur mesure ».
- La multiplicité des intervenants (maître d'ouvrage, maître d'œuvre, géomètres, projeteurs, mètres, entrepreneurs de gros et de second œuvre, ...) à faire intervenir pour un projet donné, constitue un frein à l'informatisation globale.

Travail sur mesure et multiplicité des intervenants ... Ce sont là des conditions peu favorables à la mise en œuvre des systèmes de C.A.O. développés pour des industries à organisation concentrée des études de conception. Comment amortir de tels systèmes dans un système de production doublement dispersé, pour ce qui est des produits comme pour ce qui est des intervenants ?

Et pourtant ! L'intérêt de l'outil informatique est incontestable en vue de permettre, à chaque stade d'un projet de construction, d'évaluer et de représenter rapidement les conséquences, notamment esthétiques et économiques, des différents choix possibles ! De même, cet outil devrait pouvoir donner à chaque intervenant le moyen d'enregistrer et de transmettre au moyen de supports appropriés, les étapes successives de l'étude ! ... AIDER A LA DECISION, FACILITER LA COMMUNICATION, voilà bien deux fonctions que pourrait utilement jouer l'outil informatique dans le processus de construction.

Le CACT, société de services informatiques spécialisée dans le secteur de la construction (bâtiment et travaux publics), propose à tous les intervenants des logiciels sur micro- et mini-ordinateurs, dans les trois domaines suivants :

- CONCEPTION
- ETUDES TECHNIQUES (calcul des structures, calcul et dessin de charpentes et de béton armé),
- ETUDES ECONOMIQUES, comprenant notamment mètres détaillés, études de prix d'entreprise, planning, gestion des approvisionnements, suivi technique, situations de travaux ...

Plusieurs milliers de logiciels ont ainsi été diffusés et plus de 600 systèmes matériel-logiciel « clé en main » ont été directement installés par le CACT, dans des entreprises de toutes tailles, dans des bureaux d'études, chez des ingénieurs-conseils, des architectes, des mètres ...

La continuité des développements et de la mise à jour des logiciels, ainsi que la formation des utilisateurs et l'assistance sont assurées par notre équipe d'une quarantaine d'ingénieurs en génie civil, d'architectes, de géomètres, de techniciens ayant tous une double formation BTP et informatique.

Deux des trois domaines cités ci-dessus, les ETUDES TECHNIQUES et les ETUDES ECONOMIQUES ont été assez rapidement pourvus en logiciels performants, les cahiers des charges correspondants ayant pu être définis sans difficultés majeures en collaboration entre nos clients et leurs interlocuteurs au CACT.

Pour ce qui est de la CONCEPTION, une réflexion préliminaire approfondie a été menée en vue de définir un outil de C.A.O. spécifique à l'industrie de la CONSTRUCTION, permettant de proposer dans des délais raisonnables des solutions d'AIDE A LA DECISION couvrant les demandes immédiates de divers intervenants, sur des matériels de coût abordable, mais autorisant des évolutions ultérieures, notamment dans le sens de l'AIDE A LA COMMUNICATION.

Le résultat, c'est le système ARCADE, conçu comme un système modulaire et évolutif pouvant, dans une première étape, apporter à chacun des intervenants dans le processus de conception, un outil d'aide à la DECISION adapté à sa tâche actuelle et tenant compte de ses possibilités d'investissement présentes. Dans une seconde étape, les logiciels du C.A.O. doivent pouvoir être reliés progressivement aux logiciels d'application existants en matière d'études économiques puis d'études techniques, en vue de permettre une COMMUNICATION de plus en plus efficace entre les différents maillons de la chaîne constructive.

Le CACT a ainsi retenu pour développer ses logiciels de conception des unités centrales sur lesquelles fonctionnaient déjà ou devaient prochainement fonctionner ses autres logiciels d'application. Ces matériels étaient soit des micro-ordinateurs tels que HP9816 et HP9836, soit des mini-ordinateurs tels que HP1000 (A600 et A700) pour les utilisateurs souhaitant un matériel multi-poste. Par ailleurs, une version de C.A.O. adaptée aux ordinateurs personnels compatibles

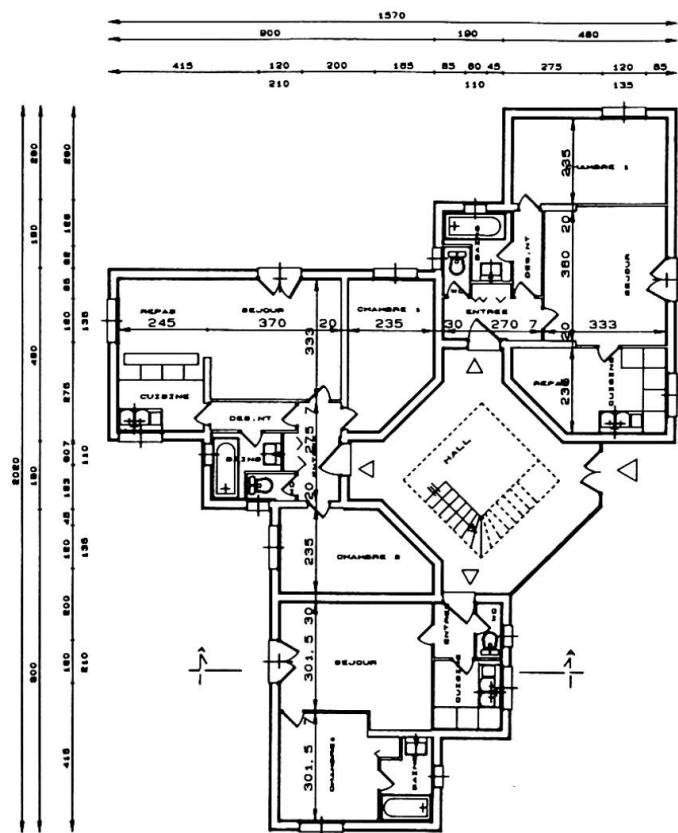


Photo 1 : ARCADE – Vue en plan – Sortie sur traceur

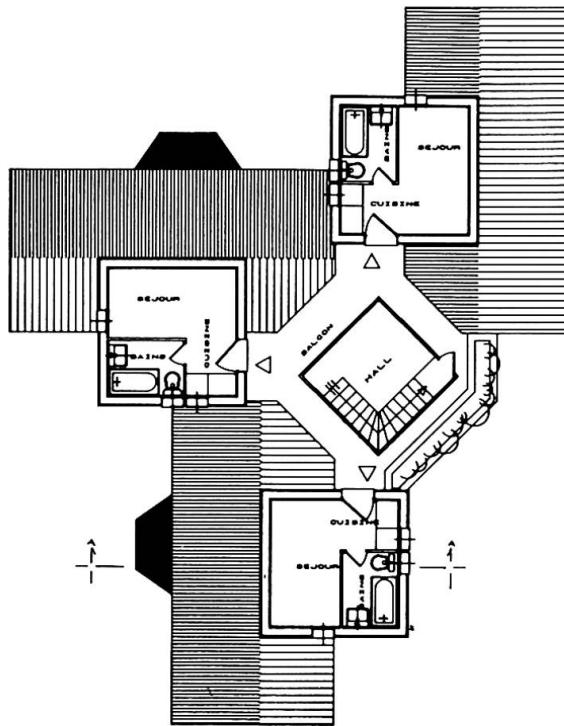


Photo 2 : ARCADE – Vue en plan – Sortie sur traceur



IBM, sur lesquels sont déjà diffusés la plupart des logiciels d'application du CACT, est en développement depuis plus d'un an et sera diffusée prochainement en version anglaise comme en version française.

Ces unités centrales doivent bien entendu être munies des périphériques requis pour les besoins de la conception, particulièrement :

- tablette,
- écran graphique,
- traceur.

2 – L'AIDE A LA DECISION

Concevoir, c'est concrétiser progressivement, selon un processus itératif de traitement de l'information, l'IDEE que l'équipe de conception a pour mission de mettre en œuvre, en convergeant aussi rapidement que possible, à travers des OBJETS INTERMEDIAIRES, vers le projet optimal qui constituera l'OBJET FINAL de l'étude de conception.

Le système informatique d'aide à la décision doit permettre à chaque instant de comparer les performances de l'objet intermédiaire avec les performances visées dans les différents domaines, notamment fonctionnel, esthétique et économique.

Bien entendu, il faut que l'utilisateur puisse notamment faire apparaître à l'écran, à tout instant, les plans côtés, les façades et les perspectives de l'objet étudié, qui peut être un ouvrage isolé ou, dans le cas de la mise en site (que permet une option d'ARCADE), un ensemble d'ouvrages. Mais pour qu'il y ait véritablement conception assistée (C.A.O) et non seulement dessin assisté (D.A.O.), le système ARCADE se charge aussi de métré automatique des quantités et du métré estimatif, à partir des prix unitaires en mémoire.

Avec ARCADE, tout objet intermédiaire est un ensemble homogène d'informations définies de manière unique et modifiable au gré du projeteur. A partir d'une idée donnée, il peut exister un grand nombre de démarches différentes pouvant aboutir au même objet final et qui représentent l'historique du projet. ARCADE ne mémorise pas cette histoire mais seulement le dernier objet obtenu, qui est le résultat, à un instant donné, de la démarche suivie par le concepteur. Le rôle du système informatique est de structurer l'ensemble des informations définissant l'objet, qui ont été introduites par le concepteur, d'effectuer à partir de ces informations les représentations et les calculs demandés et d'en permettre les modifications. Avec des délais de réponse acceptables, même sur les systèmes les plus économiques.

Ces informations sont de deux natures différentes :

- L'information « support » qui est une information géométrique et qui est introduite, au moyen d'un « menu » géométrique sous la forme d'une esquisse « fil de fer », progressivement mise à dimensions. Bien entendu, le concepteur peut choisir de faire appel, comme premier « objet » intermédiaire, à l'objet final d'une étude antérieure similaire, conservée en mémoire externe et dont il peut déformer et compléter à loisir le « fil de fer ».
- L'information « supportée » qui est une information spécifique à l'application construction (matériaux, prix, caractéristiques mécaniques, thermiques, acoustiques ...) et qui est peut être structurée en un grand nombre de niveaux. Cette information est introduite en faisant appel au « menu paramètres » et à une bibliothèque d'éléments constructifs (murs, cloisons, portes, fenêtres, éléments sanitaires, garde-corps ...) créés par l'utilisateur.

3 – L'AIDE A LA COMMUNICATION

Le système de CONCEPTION-CONSTRUCTION développé par le CACT doit permettre les liaisons entre les logiciels de CONCEPTION d'une part, les autres logiciels d'application existants et futurs en ETUDES TECHNIQUES et en ETUDES ECONOMIQUES d'autre part.

Deux types de liaisons ont été retenues :

- liaison entre deux matériels différents, l'un muni des logiciels de conception, l'autre des logiciels d'application, par l'intermédiaire d'un support magnétique,
- mise en place de l'ensemble des logiciels sur le même matériel, qui doit alors convenir de manière aussi satisfaisante que possible, aux deux types d'applications.

La première solution a été expérimentée en entreprise, avec succès, en utilisant, pour la partie conception, un micro-ordinateur HP9836 et pour la seconde partie, un mini-ordinateur HP250.



Photo 3 : Poste de travail C.A.O. sur mini-ordinateur



Photo 4 : les perspectives permettent, dans le cours de la conception, un contrôle esthétique de tout objet intermédiaire

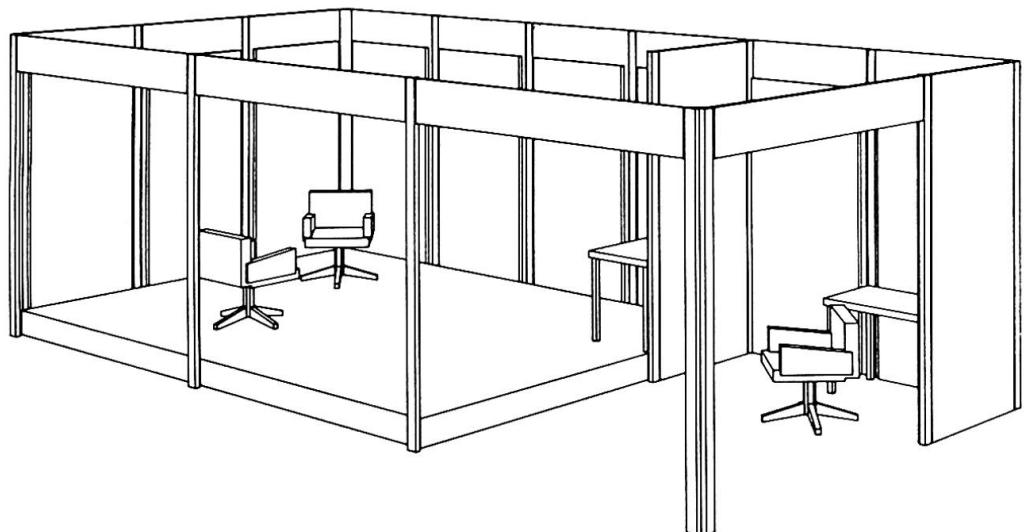


Photo 5 : ARCADE peut être utilisé aussi bien pour la mise en site des ouvrages que, comme ici, pour l'aménagement intérieur

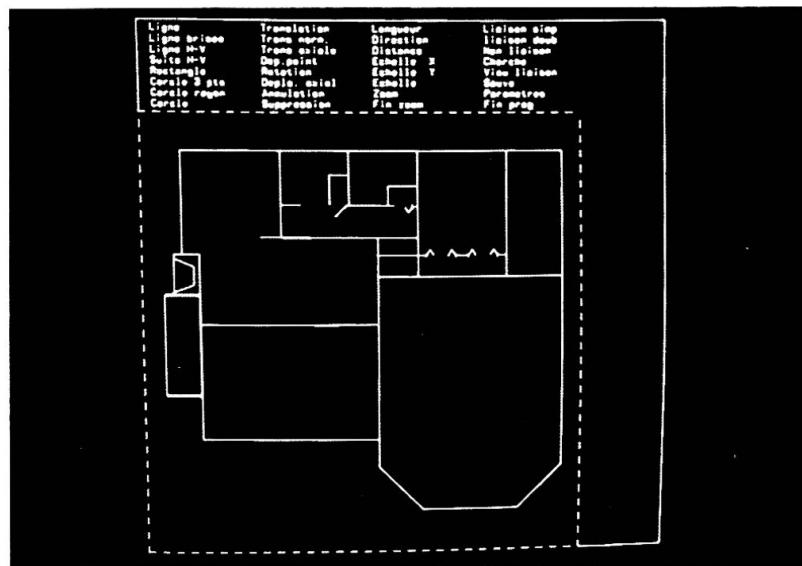


Photo 6 : Esquisse « fil de fer » — Tracé et mise à dimensions au moyen du menu « géométrique »

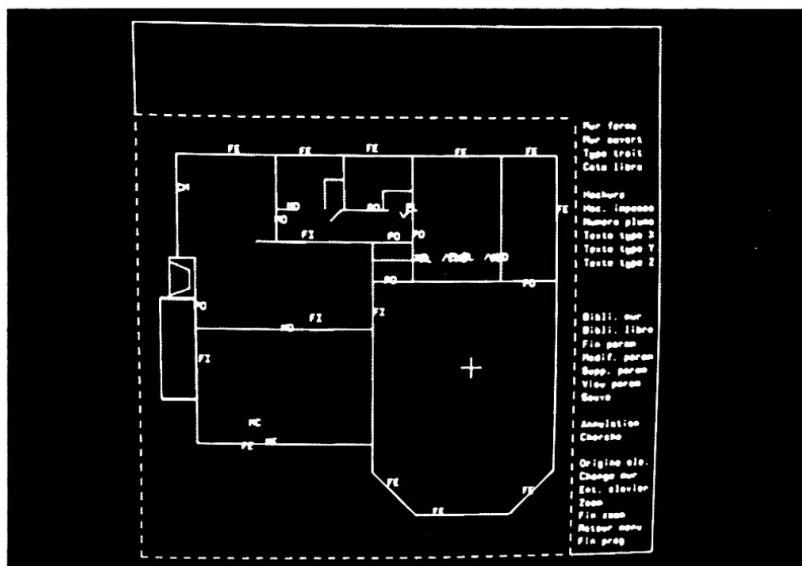


Photo 7 : Esquisse « fil de fer » — Mise en place des éléments de bibliothèque (murs, cloisons et autres éléments constructifs) au moyen du menu « paramètres »

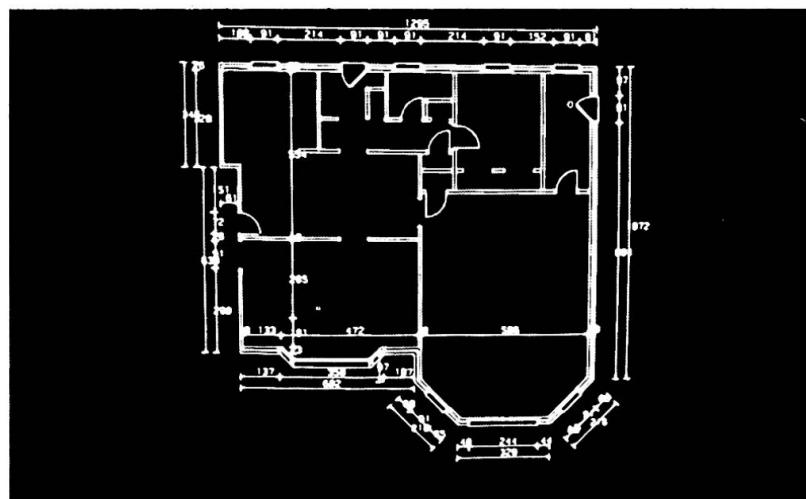


Photo 8 : A tout moment, le concepteur peut faire apparaître le graphisme côté

La seconde solution a nécessité la transposition, sur les matériels utilisés pour la conception, des logiciels d'études économiques, qui fonctionnaient précédemment sur du matériel de gestion (HP 250). Toujours avec le souci de répondre aux besoins divers des utilisateurs, cette transposition a été effectuée, d'une part sur micro-ordinateur (tel que 98XX), d'autre part sur matériel multi-poste (tel que HP1000).

Pour l'instant, la priorité a été donnée essentiellement à la liaison de la C.A.O. avec les logiciels d'études économiques (métrés détaillés, études de prix d'entreprise détaillée, planning, gestion des approvisionnements, suivi technique, situations de travaux ...). Ainsi, les études économiques donnent une dimension supplémentaire à la C.A.O., en permettant notamment à l'entreprise :

- de mieux répondre aux appels d'offres en fonction de son propre savoir-faire et en proposant, le cas échéant, des variantes personnalisées (par exemple faisant appel à un système de construction industrialisée);
- de mieux contrôler ses prix de revient par la comparaison du réel au prévisionnel, pour chaque phase d'avancement du chantier.

En ce qui concerne les logiciels d'études techniques, la liaison C.A.O.-calculs thermiques a été réalisée et cette liaison est en cours de perfectionnement en vue de permettre une meilleure prise en compte de l'environnement de l'ouvrage. Nous travaillons par ailleurs sur la liaison avec les plans de coffrage en vue de permettre ultérieurement le lien avec la descente des charges.

Ainsi se complète peu à peu, tout en répondant au mieux, à chaque instant, à la demande et aux possibilités d'investissement des divers intervenants dans l'acte de construire, un système modulaire et évolutif de CONCEPTION CONSTRUCTION accessible sur des configurations variées de micro- et de mini-ordinateurs et susceptible d'apporter non seulement une AIDE A LA DECISION, mais aussi une AIDE A LA COMMUNICATION de plus en plus efficaces.

Leere Seite
Blank page
Page vide

Conclusions to Seminar II, Computer-Aided Structural Engineering

MICHELE FANELLI, Prof., ENEL, Milano, Italy

From the inception of computer use in an engineering environment, the expectations about what the computer can do for the engineer have been growing at an ever-increasing pace. As a counterpoint, however, so have been growing - perhaps at an even faster pace - the things that the engineer has been required to do for the computer.

Nevertheless, we have constantly been told that better solutions were just around the corner, that the next-generation hardware, or software, would sweep away what little difficulties were still being encountered, etc.

Nowadays the cry and hue in the engineering milieu is all about CAD, or CAM, or more generally CAE. Many powerful corporations are vying and manoeuvering for elbow space in the field, offering "working stations" of ever more glamorous performances.

The hardware people are telling us that they have squeezed the computing power of a "mainframe" into dedicated miniprocessors which form the heart of "intelligent" working stations, giving the user faster response and avoiding the "information jams" that can result in centralized systems with many "stupid" working stations sharing a common large computer. The software people in turn tell us that, above and beyond the "engineering" programs, they have developed "data-base" techniques allowing all informations and decisions about the different elements of a design to be stored, cross-checked and made consistent at every step.

Taken at their face value, such assertions could convey the impression that tomorrow's engineer - if not exactly today's - will just have to relax in his ergonomically designed armchair and let the machine do all the dirty work: he will have still to attend to some little tasks, like getting the input right and keeping an eye on the output drawings, but otherwise he will discreetly fade away into the background and let the computer steal the show.

If you can get the inside view of people having actually tried to kick a C.A.D. system into day-to-day activity, they will, however, quite often tell you a somewhat different story. A story of toil and sweat - not usually of blood, it is true, but quite often of frustrations, maddening delays and amazing stumbling blocks or dead-end alleys.

Why, one is bound to ask oneself, such a dramatic discrepancy between rosy expectations and hard reality? Where does the truth lay? Why, above all, in spite of recurring waves of disillusionments, still the profession is falling for the pleas of the "informatics" man?

I must confess that I have given a great deal of thought to this riddle, and I feel that there is, in fact, an answer. Maybe it is only a part of the truth, but I am convinced that every engineer worth his salt and bread would concur in it.

This rationalization of the "C.A.E. puzzle" hinges, in a nutshell, on the following considerations:

- 1) It is perfectly true that EDP can speed up considerably some phases of the engineering process: all those phases involving



lengthy, complex, but well-defined computations. In all these phases not only the human designer cannot compete: it would be very stupid and inefficient not to step back and let the computer go number-crunching happily.

- 2) It is also perfectly true that not all the phases of the designing process can be formalized (either analytically or logically) so as to be automatized. Engineering, alas, is not an exact science: it is an art. Quite aptly, it could be defined as the art of reaching acceptably "correct" decisions based on insufficient (incomplete) information and taking avail of previous experience. There are, in other words, logical "jumps" in the process of design, which we couldn't, and shouldn't, demand the computerized system to bridge. It is in these "jumps" that a really good practising engineer shows his mettle.
- 3) The logical solution seems, therefore, to lie in a symbiosis between the C.A.E. system and the experienced engineer. For this symbiosis to be effective, several conditions have to be met.
 - 3.1) The C.A.E. software and hardware systems should be flexible enough to allow the expert's intervention at every step of the design process, and to incorporate those interventions smoothly into the "automatic" phases of the process and into the "data-base". This "interactive" character is underlined in the articles by McGuire, Anderheggen.
 - 3.2) The system should "prompt" the user to take the meaningful decisions at the appropriate steps (when the machine finds "holes" in the input data, or reaches a stage where non-formalizable decisions have to be taken). This prompting should be made as "user-friendly" as possible, through well-established interactive techniques such as the presentation of menus of options etc. These features are variously present in the papers by Pfaffinger, Monsarrat, Anderheggen.
 - 3.3) The use of the system should be in the hands of an experienced engineer. A beginner could well use the system to "learn", under the guide of an expert, but should never be let to "design" all by himself an actual project: he will soon run into trouble or produce very doubtful results.
 - 3.4) The system should be "open" for future additions and improvement "without tears" (Monsarrat).

These considerations may look, and sound, rather obvious and anticlimaxing; nevertheless, I think they could be helpful in dispelling many myths about what C.A.E.. systems can do for us, as well as many disillusionments and heartburns about what we engineers are currently required to do for C.A.E. systems. The latter, for sure, are here to stay; maybe in the future the technique of the so-called "expert systems" will further ease the burden that the human partner of the system will have to carry; but that there will always be a burden to carry is a fact of life and the profession had better face it. It is, and will be, a burden of responsibility, intuition, experience and faith. We should be prepared to carry it with pride.