Development of an Interactive Finite Element Analysis Program Within MS Windows 95 Environment

Gustavo E. Pacheco Crosetti, MSCE, PE
Associate Professor, Department of Civil Engineering
Polytechnic University of Puerto Rico
E-mail: bpacheco@pupr.edu

Eduardo J. Veras Jorge, MSME, PE
Associate Professor, Department of Mechanical Engineering
Polytechnic University of Puerto Rico
E-mail: everas@pupr.edu

ABSTRACT

The objective of this project was to develop a highly interactive, graphically oriented, special purpose Finite Element Analysis program, that utilizes the MS-Windows 95 environment as the Graphical User Interface (GUI). The system has been developed for personal computers; Visual Basic 5.0 has been selected as the programming language for its versatility and the advantages of its toolbox to develop applications that follows the MS-Windows standards. The program performs plane stress analysis, using four nodes isoparametric elements (Q4), considering static loads, linear elastic material behavior and small displacements. It is specifically oriented to non-prismatic beam type models (defined by two vertical boundaries at the most left and the most right limits). The designed program minimizes the user's learning process and allows the sharing of numerical and graphical data with other MS-Windows applications.

SINOPSIS

El objetivo de este proyecto fue desarrollar un programa de computadora altamente interactivo y orientado a gráficas para el análisis de elemento finito utilizando el ambiente de Windows 95 como su GUI ("Graphical User Interface"). Además de su versatilidad, el programa de computadora Visual Basic 5.0 fué seleccionado como el lenguaje de programación debido a las ventajas que ofrecen sus herramientas en el desarrollo de aplicaciones que siguen los estándares de MS-Windows. El programa puede llevar a cabo análisis de esfuerzos en el plano utilizando elementos isoparamétricos de cuatro nodos (Q4) considerando cargas estáticas, el comportamiento de material elástico lineal y los desplazamientos pequeños. Está orientado específicamente a los modelos de viga de tipo no prismático (definidos por dos fronteras verticales en los límites del extremo izquierdo y del extremo derecho). El programa resultante minimiza el tiempo de aprendizaje del usuario y le permite compartir los datos numéricos y gráficos con otras aplicaciones en MS-Windows.

I- INTRODUCTION

Within the area of structural engineering there is a great dependence on computer programs developed outside of Puerto Rico. These programs sometimes do not match the local expectations and needs. On the other side, there is a large quantity of general-purpose finite element analysis (FEA) programs, that are either not user-friendly to be applied in specific applications, or too expensive for local engineering companies. This situation causes that most engineers do not use these tools. Instead they perform more simplified analysis, which in turn implies less economical solutions and, sometimes, less secure solutions. This lack of adequate tools also limits the engineers' creativity to propose new structural shapes and designs.

The above mentioned considerations motivated the PUPR to impulse the research and development within the areas of structural modeling, numerical methods, FEA, and computer graphics. Computer Aided Engineering (CAE) is an area of applied research and development that will contribute to the university by providing a more active role in the technological development of the engineering field in Puerto Rico. The selection of a specific application, such as the one presented in this paper, not only allows the practitioner engineer to obtain a useful and flexible tool that meets a particular necessity, but also greatly contributes by developing a research and development area, since it implies a deep inside in programming tools and a deep involvement with key concepts of computer graphics and numerical methods.

The development of computer graphics application programs within the area of structural analysis has been a growing research topic in the last decade. Most university efforts have been oriented to UNIX-based computers, running GUI such as X-Windows and Motif. This fact limits the technology transfer process, considering that PC compatibles are the most commonly used computers in many of the analysis and design engineering studios, and that MS-Windows has become a de-facto standard environment in the PC market. For these reasons, the present program has been developed for PC computers, using MS-Windows as the Graphical User Interface (GUI). Visual Basic 5.0 has been selected as the programming language, due to its versatility and the advantages of its toolbox to develop applications following the MS-Windows standards.

II- PROGRAM DESCRIPTION

The Application is called NPB program, which can be defined as an interactive computer program for the analysis of Non-Prismatic Beams (or similar structures) by means of plane stress Finite Element Analysis (FEA). The program performs plane stress analysis using four nodes isoparametric elements (Q4), considering static loads, linear elastic material behavior, and the small displacements theory. It is specifically oriented to non-prismatic beam type models (defined by two vertical boundaries at the most left and the most right limits). A model analysis may be divided into three implementation phases: a) *Pre-processing Stage*, where the model data is defined, b) *Processing Stage*, where the automatic meshing of the model, and the finite element analysis are performed, and c) *Post-Processing Stage*, where results are consulted. Regarding these phases, the program permits the user, among other features, to:

A- PRE-PROCESSING STAGE

1.1- Define the solid modeling using lines, parabolas and circle arcs

1.2- Assign different support conditions, including elastic supports

1.3- Assign different loading conditions (concentrated, distributed, self weight)

1.4- Verify/modify model data presented in tabular form

1.5- Display the identification of solid modeling elements (Key Points, Curves)

1.6- Perform Zooming options

1.7- Activate/deactivate many visibility options

1.8- Save the solid model (or FEA model) as a bitmap file

1.9- Copy the solid model (or FEA model) as a bitmap to the Clipboard

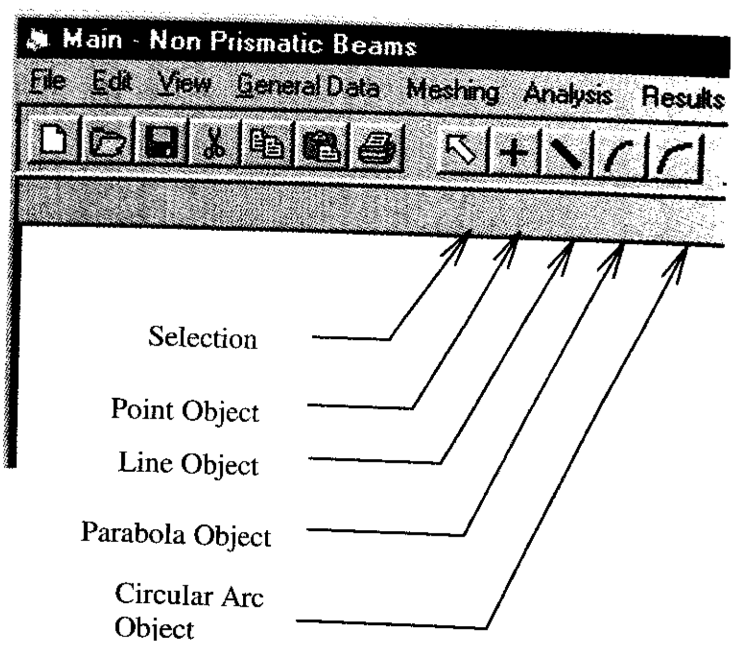


Figure 1: Drawing toolbar.

B- Processing Stage

2.1- Define the parameters to control the number of divisions for the meshing

2.2- Perform the automatic meshing of the solid modeling with quadrilateral elements

2.3- Perform a static linear elastic analysis of the model, using plane stress elements

C- Post-Processing Stage

3.1- Review the output file (nodal displacements, element strains and stresses)

3.2- Future developments will include the graphical post-processing of the results (i.e. deflected shape, stress contours, etc.)

The following sections present a brief description of each of these three stages through the use of examples.

III- EXAMPLES OF PRE-PROCESSING CAPABILITIES

This stage consists of the *Solid Modeling* generation: interactive definition of the beam geometry, support conditions, loads, and any other model data. Special care was taken to include graphical and tabular data entry, zoom options, visibility options (to activate/deactivate toolbars visibility, solid modeling identification, among others), file management (save/open models) and setting options (limits, colors, among others). The program database is easily modifiable to allow the users to be in control of the program execution flow, moving from one stage to the other according to their needs. This allows an iterative analysis (new model proposal and re-analysis), which is a characteristic of the design process.

NPB provides several drawing objects for graphical generation of solid models. These are: Points, Lines, Parabolas, and Circular Arcs. Using these objects, the user is able to generate the boundaries of a model. Many geometries can be represented using these drawing utilities, provided that they represent a closed region (closed regions are geometric constraints imposed by the Finite Element Analysis Meshing procedure). Figure 1 shows the Drawing Toolbox.

A very simple procedure is followed for the graphical generation of any geometry: The user selects a drawing object from the Drawing Toolbox. The extreme points of each type of curve are called Key-Points, and their position may be specified with mouse clicks or by editing a tabular form. Drawing aids are provided for accurate selection of points on the screen (grid lines and snap options). Figure 2 shows an example of a solid model, and Figure 3 displays a Key-Point table.

Support and loading conditions are also assigned using toolbar options. The specific condition is selected (if a numerical value is required, a dialog box is displayed) and assigned to the corresponding Key-Point or Curve. Figure 4

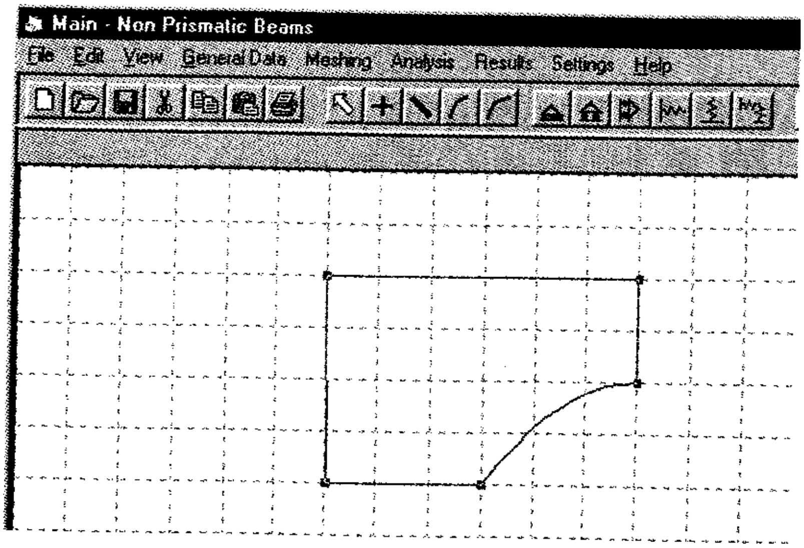


Figure 2: *Solid model generated using NPB.*

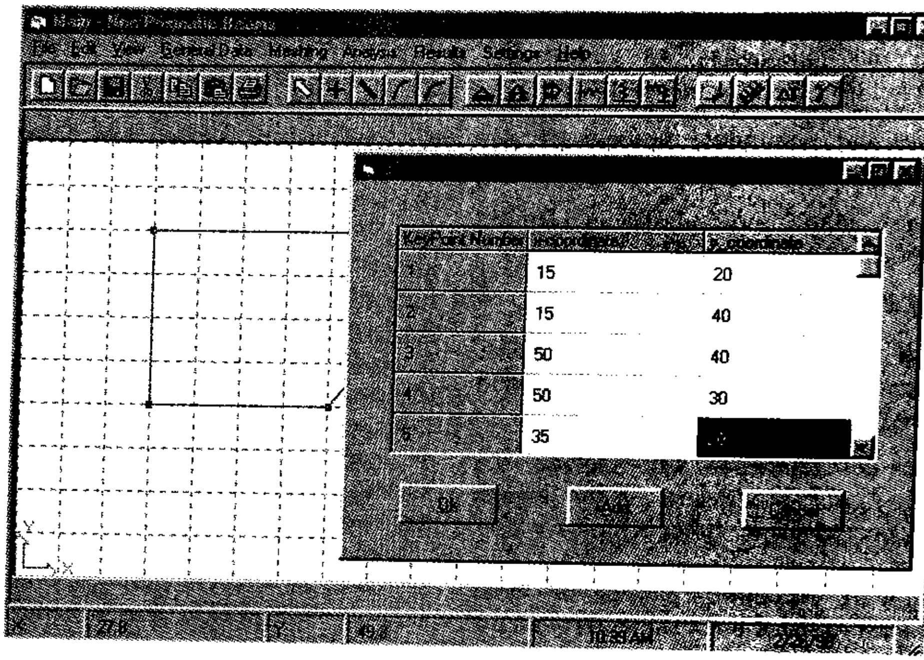


Figure 3: Key-point table.

shows a model with the support conditions applied. It also shows a solid model after activating the key-points and curve identification numbers.

IV- EXAMPLES OF PROCESSING CAPABILITIES - AUTOMATIC MESH GENERATION

The analysis process consists of the following steps: a) Automatic Finite Element Mesh generation, which allows the control of the number of elements used to discretize the model in each direction; b) Identification of the Global Degrees of Freedom; c) Assembling of the global stiffness matrix and load vector (a storage vector of the skyline components of the stiffness matrix was used). 2-D Plane stress elements formulation with four nodes is used for the numerical approximation. Two (2) point Gauss Quadrature is used for integration; d) Solution of the system of linear equations to obtain global

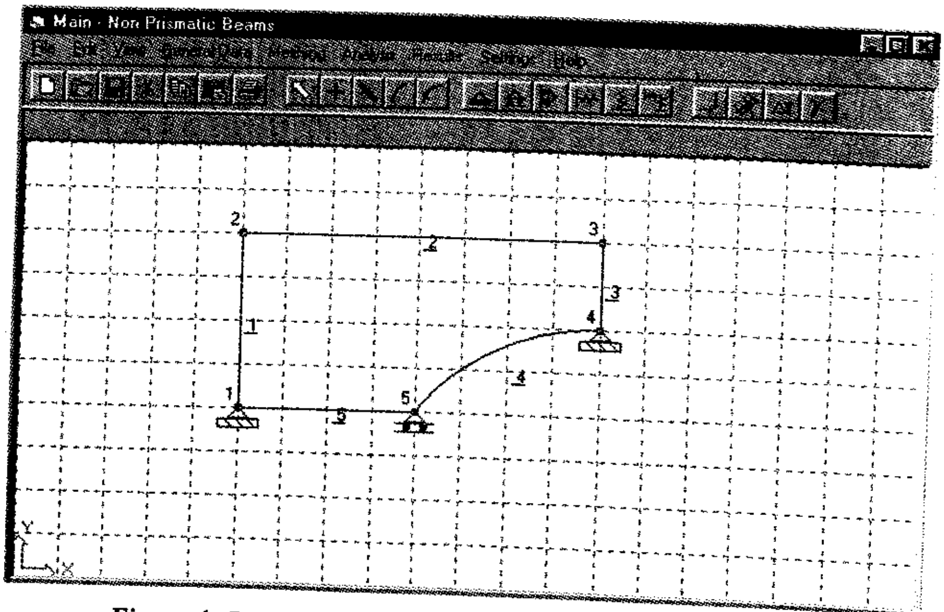


Figure 4: Support conditions. Object identification visibility.

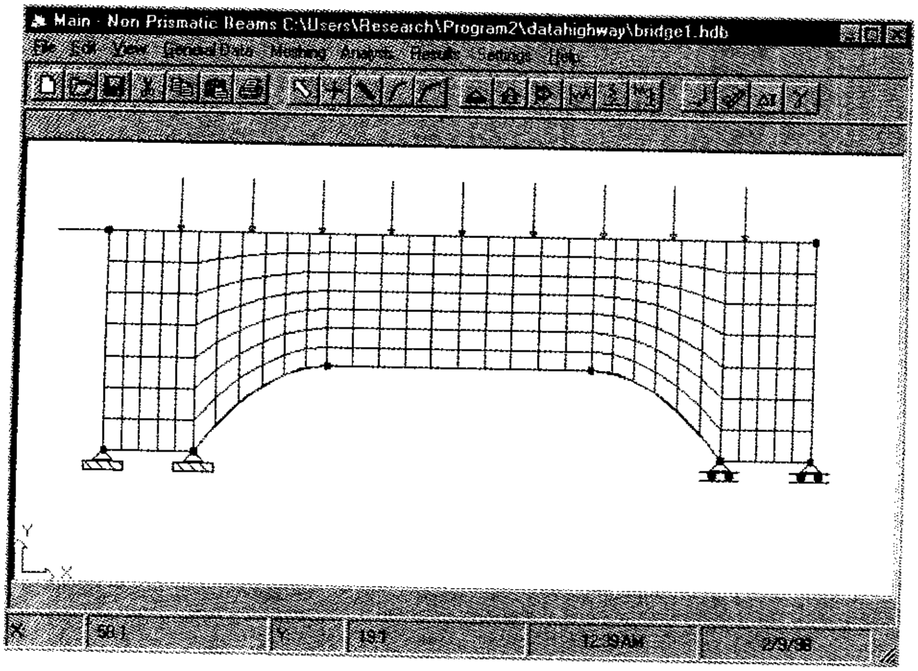


***Figure 5:** Beam model after meshing*

displacements (Gauss elimination is used to solve the system of linear equations); e) Computations of stresses and generation of the output file. Special care was taken in the Finite Element Model generation and reporting, allowing users to control the display of the model (mesh, nodes, elements identification, nodes identification), and to consult the modeling in a tabular form (where nodal coordinates changes are allowed).

Figures 5, 6 and 7 are examples of automatic mesh generation, and FEA model reporting capabilities. Figure 5 shows a view of a beam model after the discretization (generation of the Finite Element Model). Figure 6 displays the elements identification numbers. This information may be too crowded, according to finite elements

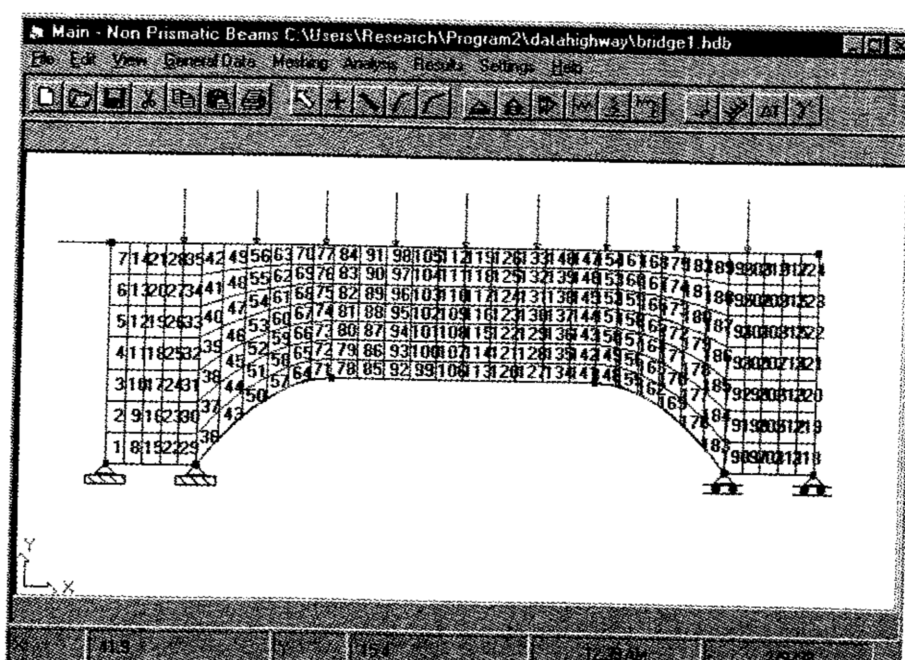


***Figure 6:** FEA model elements identification.*

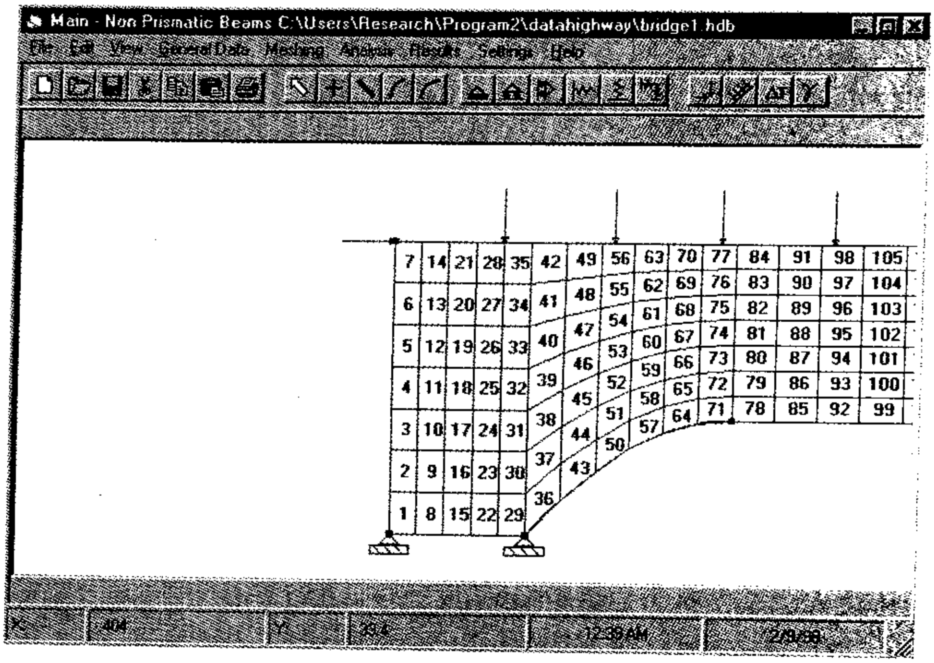


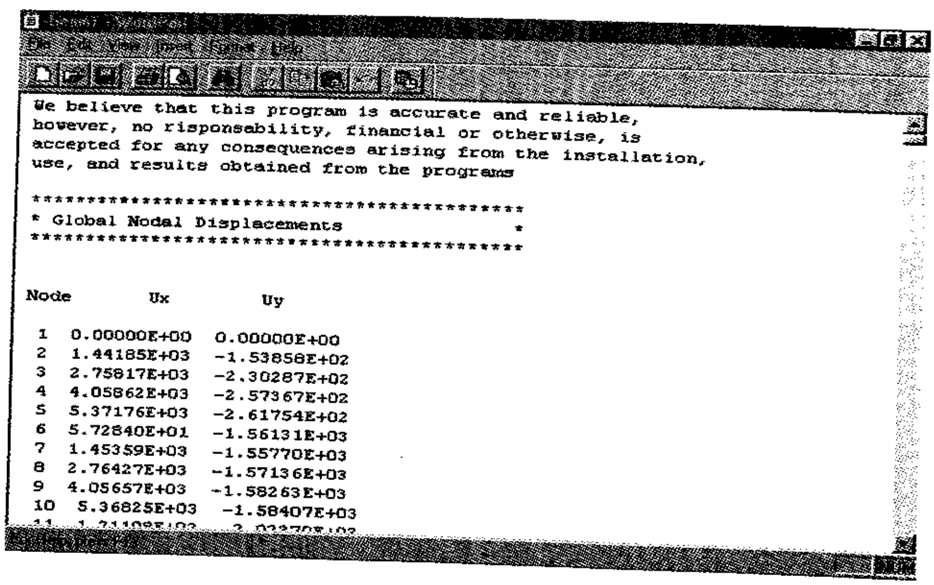
***Figure 7:** Zooming of left part of FEA model.*

density, so zooming options are provided. Figure 7 presents a zooming of the left part of the model to facilitate the verification of the elements' identification numbers.

The automatic mesh generation implemented is based on I-J mapping of physical coordinates in a normalized ε - η plane, as discussed by Beer and Watson (1992).

V- EXAMPLES OF POST-PROCESSING CAPABILITIES

After the analysis, users may consult the results using any of the Windows-based word processors available. Supported word processors are MS Write, MS WordPad, MS Word, and WordPerfect. NPB program also allows to consult the results in tabular form. Graphical post-processing of the results (i.e.



```
We believe that this program is accurate and reliable,
however, no risponsability, financial or otherwise, is
accepted for any consequences arising from the installation,
use, and results obtained from the programs

*******************************************
* Global Nodal Displacements              *
*******************************************

Node        Ux             Uy

 1   0.00000E+00   0.00000E+00
 2   1.44185E+03   -1.53858E+02
 3   2.75817E+03   -2.30287E+02
 4   4.05862E+03   -2.57367E+02
 5   5.37176E+03   -2.61754E+02
 6   5.72840E+01   -1.56131E+03
 7   1.45359E+03   -1.55770E+03
 8   2.76427E+03   -1.57136E+03
 9   4.05657E+03   -1.58263E+03
 10  5.36825E+03   -1.58407E+03
```

***Figure 8:** Output file display.*

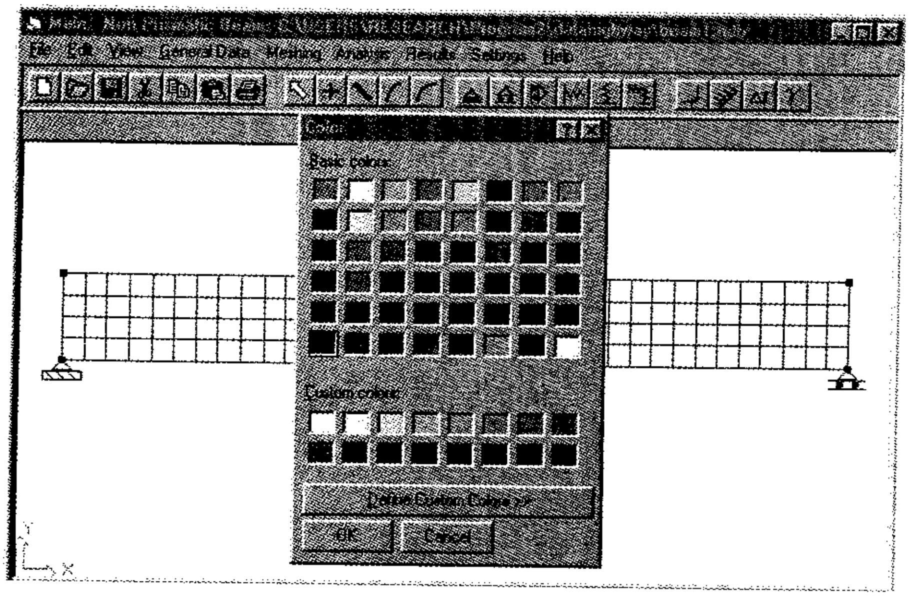


Figure 9: Standard color palette.

deflected shape, stress contours) is planned to be implemented in the next phase of the project development.

To retrieve the results, the NPB program displays a dialog box where users are allowed to check one of the four available word processors. Figure 8 shows an example of an output file loaded in MS WordPad.

VI- VISUAL PRESENTATION FEATURES AND WINDOWS-SUPPORTED UTILITIES

An image of a model may be copied to the clipboard (or saved as a bitmap) with the intention of pasting it into another application. For example, a solid model graphical representation can be pasted into MS Word or MS PowerPoint. This

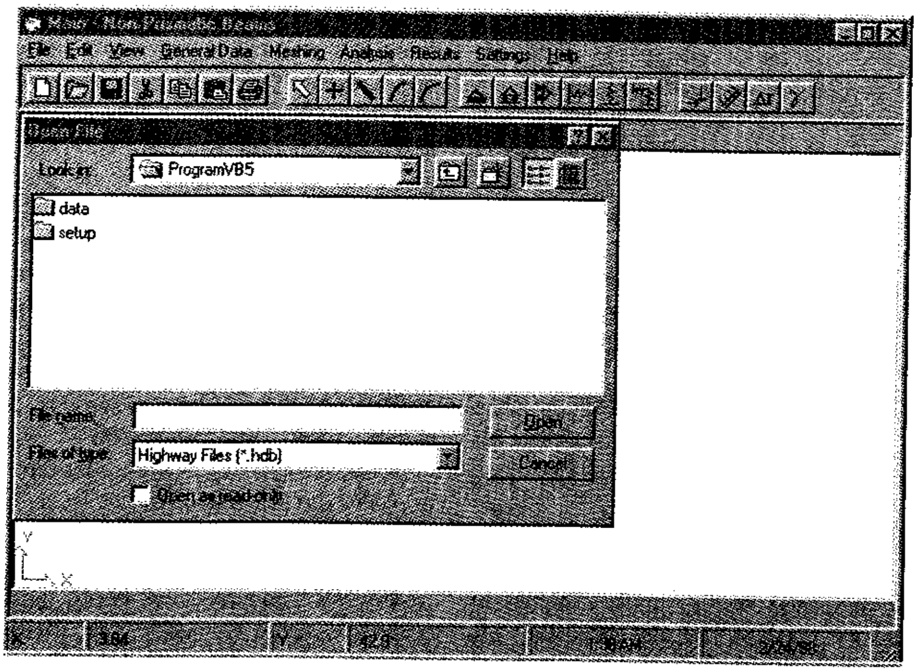


Figure 10: Common dialog box.

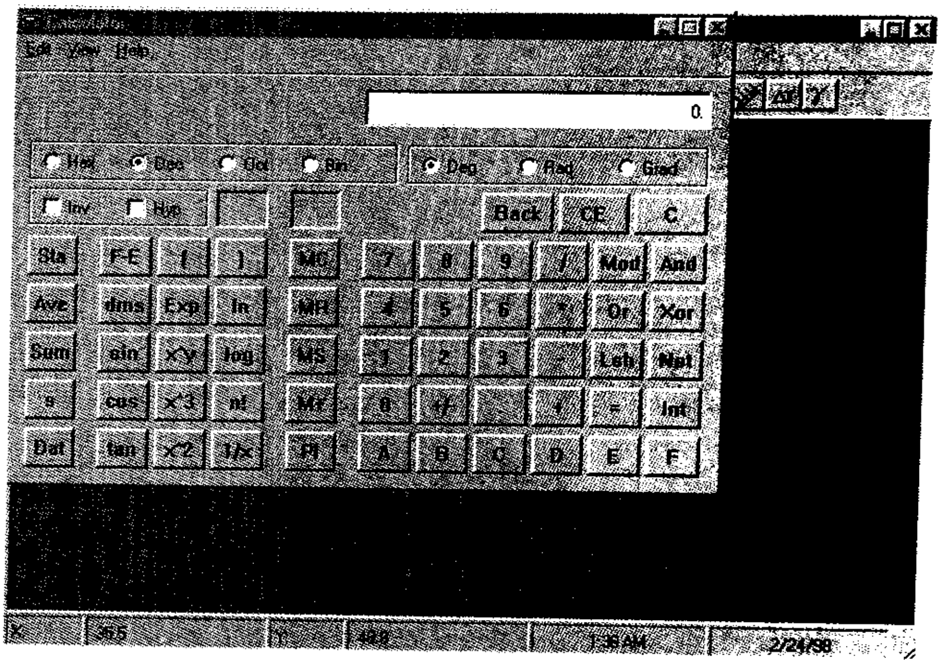


Figure 11: Scientific calculator.

feature is very useful when the results obtained through an NPB analysis need to be included in a report or a presentation. The display and exchange of graphics can be customized assigning different colors to the components of a model: Gridlines, Drawing Objects, Selected Objects, Background, Support Conditions, Applied Loads, and Meshing. Component colors can be changed using the Color Palette. A Restore Defaults option is provided to get back to the original colors defined by NPB.

NPB has been programmed to support the standard Color Palette available for Windows-based programs. The Color Palette is a dialog box that allows the user to choose among the 125 colors that define the color spectrum (see Figure 9). Some graphics adapters (VGA standard, for example) can not display all the 125 color options simultaneously. A dithered color is displayed in this case.

As shown in Figure 10, the NPB program provides a Common Dialog Control Object for managing files, which includes the Open, Save, and Save As options.

NPB provides users with a Microsoft utility for making calculations through an Application Programming Interface (API) call. Calculated values related to a particular model can be easily moved between applications using the Copy/Paste capabilities of the MS Calculator to have access to the result of calculations back in the NPB program. Figure 11 shows the Scientific Calculator dialog box when activated.

VII- CONCLUSIONS

An interactive application for Finite Element Analysis of Non Prismatic Beams and alike structures (defined by two vertical boundaries at the most left and the most right limits) using Plane Stress theory has been successfully implemented. The program is menu-driven, mouse-controlled, allowing most of the data-entry to be performed graphically. Tabular forms are used basically for editing purposes (if desired), and output information. An automatic mesh generation algorithm based on I-J mapping was implemented to speed the Finite Element Model generation, and eliminate errors during the discretization process. Several display options have been included to facilitate model generation and verification, and to allow user customization. Graphical post-processing capabilities are under development.

The program will serve as a basis for future developments of computerized analysis tools for engineering. These developments will always follow the objectives of being highly interactive, specific task-oriented, and within Windows environment and standards. These premises are considered fundamental to minimize the learning time and to assure the effectivity of the technology transfer process.

VIII- ACKNOWLEDGEMENT

This project was supported by the Department of Transportation and Public Works, Puerto Rico Highway and Transportation Authority (State Planning and Research Program - 1996), and by the Polytechnic University of Puerto Rico. The support of the respective offices at both institutions is gratefully acknowledged.

IX- REFERENCES

1- Akin, J. E. (1994) "Finite Elements for Analysis and Design", Computational Mathematics and Applications, Academic Press, 548 pp.

1- Amirouche, Farid M. L. (1993) "Computer Aided Design and Manufacturing", Prentice Hall, New Jersey, 538 pp.

2- Appleman D. (1993), Visual Basic Programmer's Guide to the Windows API, Ziff Davis Press, Emeryville, CA

3- Bathe, Klaus-J. (1996) "Finite Element Procedures", Prentice Hall, New Jersey, 1037 pp.

4- Beer, G. and Watson, J. O. (1992) "Introduction to Finite and Boundary Element Methods for Engineers", John Wiley and Sons, Inc., 509 pp.

5- Cook R. D., et all (1989), Concepts and Applications of Finite Element Analysis, Third Edition. John Wiley & Sons, New York, NY

6- Chandrupatla T. R., Belegundu A.D. (1997) "Introduction to Finite Elements in Engineering", Second Edition, Prentice Hall, New Jersey, 461 pp.

7- Gurewich, N. and Gurewich O. (1996) "Teach Yourself Visual Basic 4 in 21 Days", Sams Publishing, Indianapolis, 960 pp.

8- Krishnamoorthy C. S. (1994) "Finite Element Analysis: Theory and Programming", Second Edition, Tata McGraw Hill Publishing Company, New Delhi, 710 pp.

9- Krishnamoorthy C. S., and Rajeshirke, U. Adaptive Mesh Refinement for Two Dimensional Finite Element Stress Analysis, Computers and Structures, Vol. 48, No. 1, pp 121-133, 1993.

10- Park, Chan S. (1985) "Interactive Graphics Programming", Addison-Wesley Publishing Company, Reading MA. 458 pp.

11- Plastock, Roy A. and Kelley, Gordon (1986) "Computer Graphics", Schaum's Outline Series in Computers, MacGraw-Hill Book Company, 340 pp.

12- Segerlind, Larry J. (1984) "Applied Finite Element Analysis", John Wiley and Sons, Inc., 427 pp.

13- Stephens, Rod (1996) "Visual Basic Algorithms", John Wiley and Sons, Inc., 322 pp.

14- Stephens, Rod (1997) "Visual Basic Graphics Programming", John Wiley and Sons, Inc., 677 pp.

15- Weaver W., Johnston P. R. (1984) "Finite Elements for Structural Analysis", Prentice Hall, New Jersey, 403 pp.

16- Zecher, Jack E. (1994) "Computer Graphics for CAD/CAM Systems", Marcel Dekker, Inc., New York, 412 pp.

17- Zienkiewicz, O.C., and Zhu, J.Z. (1987) A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis, International Journal for Numerical Methods in Engineering, Vol. 24, pp. 337-357.